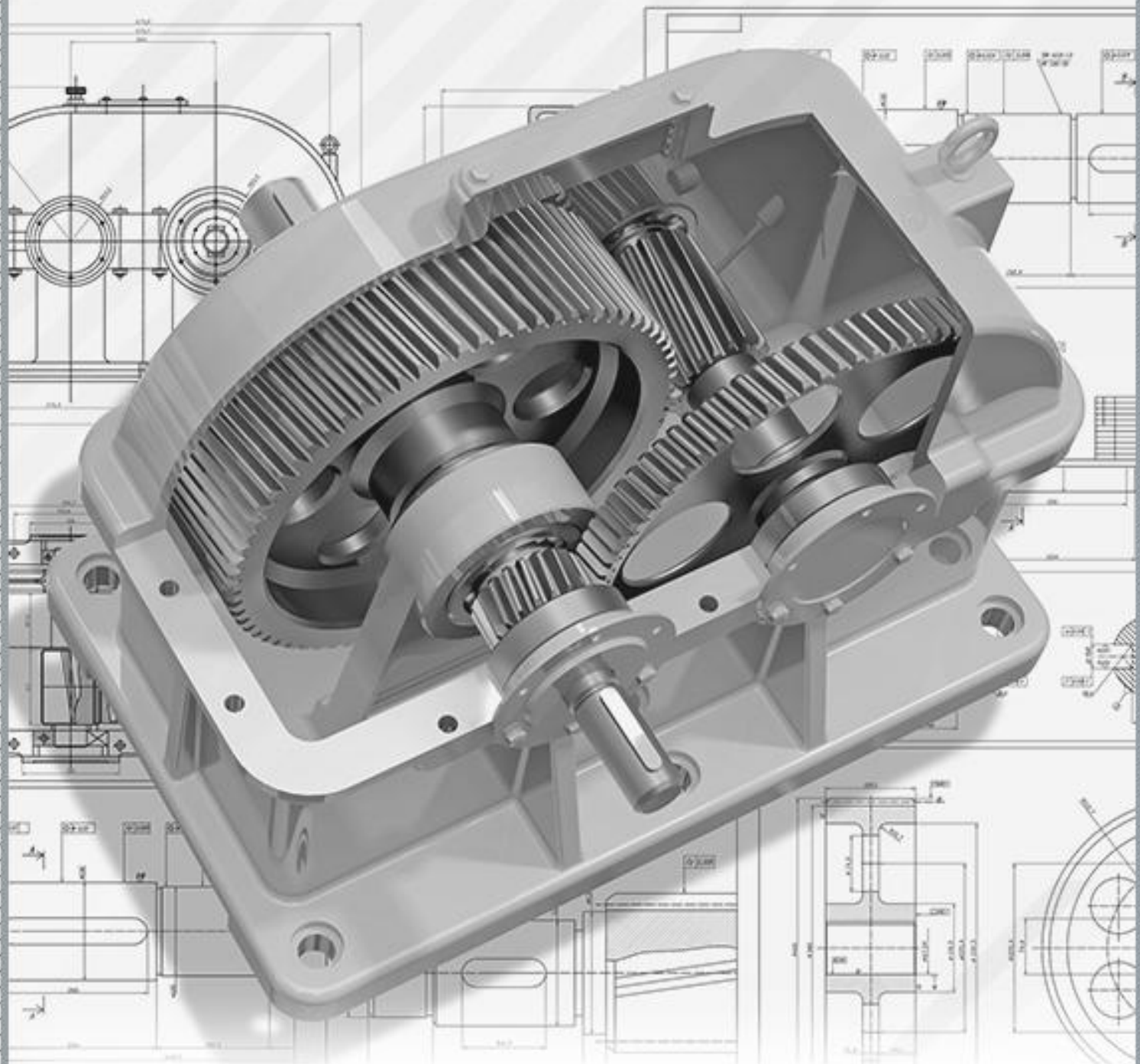


TurboCAD[®]

2023

User Guide



MSI DESIGN[®]

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Getting Started with TurboCAD 2023



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Welcome to the TurboCAD 2023 User Guide.

This section covers what you should know before and during TurboCAD installation, and fundamental concepts of using files, including import / export and page setup.

1.1. System Requirements

System Requirements

TurboCAD 2023 Platinum

TurboCAD 2023 Platinum 32-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7

4 GB RAM

TurboCAD 2023 Platinum 64-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7

8 GB RAM

*Note: Windows 8 does not mean Windows RT compatibility. TurboCAD v27 will not run on tablets like the Microsoft Surface running Windows RT.

Recommended:

Your experience with TurboCAD Platinum will be greatly enhanced with a newer generation, higher speed CPU, 4 GB RAM for 32-bit and 8 GB RAM for 64-bit, and larger display resolution and graphics support.

The optional GPU-accelerated Redsdk render modes require a supported graphic processing unit. The latest video drivers are typically required. Newer boards with more power and VRAM generally provide greater performance.

Recommended GPU-accelerators: NVIDIA® Quadro FX 1000 & up • NVIDIA® GeForce FX & up • NVIDIA® GeForce 6, 7, 8 & up • AMD RadeonHD 2000, 3000, 4000 & up • AMD-ATI Radeon 9500 & up • AMD-ATI Radeon X, HD 2000, 3000, 4000 & up • AMD-ATI FireGL X • AMD-ATI FireGL V 3000 & up • Intel(R) HD Graphics Family & up • INTEL® Integrated graphics series: GMA 3000 & GMA X3000

TurboCAD 2023 Professional

TurboCAD 2023 User Guide Publication

TurboCAD 2023 Professional 32-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7
4 GB RAM

TurboCAD 2023 Professional 64-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7
8 GB RAM

*Note: Windows 8 does not mean Windows RT compatibility. TurboCAD v27 will not run on tablets like the Microsoft Surface running Windows RT.

Recommended:

Your experience with TurboCAD 2023 Professional will be greatly enhanced with a newer generation, higher speed CPU, 4 GB RAM for 32-bit and 8 GB RAM for 64-bit, and larger display resolution and graphics support.

TurboCAD 2023 Deluxe

TurboCAD 2023 Deluxe 32-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7
4 GB RAM

TurboCAD 2023 Deluxe 64-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7
8 GB RAM

*Note: Windows 8 does not mean Windows RT compatibility. TurboCAD v27 will not run on tablets like the Microsoft Surface running Windows RT.

Recommended:

TurboCAD 2023 User Guide Publication

Your experience with TurboCAD 2023 Deluxe will be greatly enhanced with a newer generation, higher speed CPU, 4 GB RAM for 32-bit and 8 GB RAM for 64-bit, and larger display resolution and graphics support.

TurboCAD 2023 Designer

TurboCAD 2023 Designer 32-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7
4 GB RAM

TurboCAD 2023 Designer 64-bit System Requirements:

Microsoft® Windows 10, Windows 8*, Windows 7
8 GB RAM

*Note: Windows 8+ does not mean Windows RT compatibility. TurboCAD v27 will not run on tablets like the Microsoft Surface running Windows RT.

Recommended:

Your experience with TurboCAD 2023 Designer will be greatly enhanced with a newer generation, higher speed CPU, 4 GB RAM for 32-bit and 8 GB RAM for 64-bit, and larger display resolution and graphics support.

1.2. Installing TurboCAD

To install TurboCAD, insert the CD into your CD-ROM. If the installation process does not start automatically, select Start / Run from the Windows taskbar and type D:\setup.exe (where D is the drive letter of the CD-ROM).

After you have installed TurboCAD, the Setup program creates a program folder. If you choose the default settings, TurboCAD is installed in the C:\Program Files\IMSIDesign\TCWP2023 folder. This folder contains several subfolders that contain TurboCAD program files and related files such as templates, sample drawings, and symbols.

Note: The Setup program also creates a program group containing the TurboCAD application icon, as well as shortcuts to the Help and the Readme file. The program group is accessed through the Start menu. Before you start the program, please read the Readme file, which contains the latest information on TurboCAD.

To remove TurboCAD from your computer:

In Windows, select **Start / Settings / Control Panel**.

Double-click **Add/Remove Programs**.

Select TurboCAD from the list.

Click **Add/Remove** and follow the instructions on the screen.

1.3. Registration

If you do not have Internet access, please call +1-800-833-8082 to register. Becoming a TurboCAD registered user has many advantages, including technical support, access to extensive online support databases, and program updates.

1.4. Getting Help

There are several ways to get help on any TurboCAD topic.

Default UI Menu: Help

Ribbon UI Menu:

Forums

Default UI Menu: Help/TurboCAD on the web/TurboCAD Forum

Ribbon UI Menu:

Go to www.turbocad.com and click User Forums. Choose the forum for the version of TurboCAD you have. The first time you enter the forums, you will be asked to register. Registration helps maintain your preference settings, and your real name and email address help us all interact with you better.

The TurboCAD Forums are the place to get the absolute best technical support . All the support comes from other users, including professional experts (architects, engineers, and drafters) from all over the world, some using TurboCAD full-time. There are also a few IMSI Design employees and programmers participating. Ours are perhaps the most

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active forums in the CAD industry. It's free, and everyone is quite eager to help. In fact, forum users help IMSI develop and improve the software!

Feel free to reply to a message, post a new message, or create your own new thread.

Introduce yourself. Formalities are waived, and you are welcome regardless of skill level. This is where we get our questions answered and our struggles resolved on a regular basis.

Online Help

The **Help** menu provides access to the online help, which basically contains the same content as this book. The help is also context-sensitive, which can be accessed in several ways:

- Click the **Help** button (question mark icon), then click on the tool button, menu item, tab, or palette in question.

- Place the cursor over a button, menu item, or palette, or highlight a menu item, and press F1.

- Press F1 while using a tool or window.

- Press the **Help** button on a window.

To see a tool tip for an icon, hold the cursor over the button for a second or two. The tool tip will appear, showing the name of the tool.

Tip: Look in the **Help** menu to see a list of keyboard shortcuts. Look in the **Help** menu for the **Tip of the Day** for helpful hints.

Technical Support

Default UI Menu: Help/TurboCAD on the web/TurboCAD Support

Ribbon UI Menu:

Go to the support section of <https://www.turbocad.com/> for detailed support options.

Technical support is available by calling **1-800-833-8082** .

Technical Support will work closely with you to solve any problems related to our software. Please give our support technicians as much information as possible.

Remember that they are not in front of your computer and that they need your help to diagnose the problem. On occasion, a problem can be traced to hardware, or to another software application. Our technician will supply as much support as possible in these cases, but they are not authorized to support products manufactured or published by another company. You might find that e-mail is a more convenient way to get technical support. If you do send e-mail, provide as much information as you can about your system and about the problem.

Technical Support CheckList

You may already have the information you are looking for. Before calling, check this manual thoroughly.

To receive the fastest response to your technical questions, please be in front of your computer with TurboCAD running, and be prepared to provide the following information before you call or send e-mail:

The type of computer and Windows version you are using.

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The name, version number, and other information about your specific version of the product. (To find this information, select **Help / About TurboCAD** from the TurboCAD menu.)

The exact sequence of events that created the problem. Verify that you can reproduce the problem by following the same series of steps.

The exact wording of any error messages.

Steps you've taken to find the answer to your question, including resources used.

The results of any steps you have undertaken to resolve the problem.

Web Tutorials

Default UI Menu: Help/Getting Started Videos

Ribbon UI Menu:

There are several online demo tutorials that can be accessed from the page that appears in the Internet Palette. You need the Macromedia Flash plug-in to view these tutorials.

These short, animated demos demonstrate some basic concepts about the user interface, and show you how to use some of the TurboCAD tools.

Help on the Internet

Default UI Menu: Help/TurboCAD on the web

TurboCAD 2023 User Guide Publication

Ribbon UI Menu:

Under the **Help** menu there is a sub-menu labeled **Help on the Internet**. Each item in the sub-menu opens to the selected web site to provide help.

1.5. Starting TurboCAD

If you accepted the default installation settings, TurboCAD can be accessed via the **Windows Start/Programs** menu.

Click the **Start** button on the Windows taskbar.

Select **Programs/IMSI Design**, and select the version you have installed.

You can also start TurboCAD by double-clicking the desktop icon. You will see a dialogue to choose the interface.

You can choose the desired interface and press ok.

Splash Screen comes a Startup Screen when Launching TurboCAD for a better interactive user interface. The interactive screen displays recently created files and ability to open/create new file via Create tab, allows user to learn about TurboCAD basics, including new feature videos, via Learn tab, and get access to the latest information about TurboCAD, via News tab.

You can view a **History** of drawings previously opened.

Click on "File" from the menu bar.

Select "Open" from the drop-down menu.

A dialogue box appears which displays the history of drawings.

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You can also browse to folders containing drawings you want to open or open a **Template** by clicking on the drop-down menu at the top left corner of the dialogue box and selecting the desired template. You can also use this window to import files of other types by changing the option in "File of type" and selecting the file type you want to open from a drop-down list.

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Opening New File:

If you want to create a new drawing, click **New**.



You can also close this window by clicking **Cancel**. You can then select **File/New** within TurboCAD to create a new drawing. The New TurboCAD Drawing window appears, in which you specify what type of file you want to open.

New from Scratch: Creates a new blank drawing with the default settings. **New from Template:** Enables you to select a predefined template, which has size, units, and layout defined.

Page Setup Wizard: Opens the step-by-step guide for setting page size and scale.

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Open Drawing: Opens an existing drawing. Browse to the desired folder to locate the file. For acceptable file types, see [Importing and Exporting Files](#)

Open Last Edited: Opens the last file you worked on.

Page Setup Wizard

Default UI Menu: File/Page Setup Wizard

Ribbon UI Menu:

A step-by-step guide to setting page size and drawing scale.

Note: This is not exactly the same as the **Page Setup** used for printing, but any relevant parameters you set in this wizard will appear in the **Page Setup**. See [Page Setup](#)

If you selected **Page Setup Wizard** in the **New TurboCAD Drawing** window, the wizard opens automatically. You can also access it via **File/Page Setup Wizard**. In the first page you can choose between **English** and **Metric** units.

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On the second page, you can set the type of unit you want to use - architectural (1'-3"), decimal (3.75 ft.), scientific, etc. You can also specify the main unit (inches, feet, yards) and the precision of the units.

The third page contains options for paper size and orientation - these are used when printing. You can also enter drawing information such as title, author, and comments.

The fourth page enables you to set the printing scale. This is the size of the objects when printed, relative to their actual size specified in the drawing.

The fifth page enables you to create a default viewport for mapping the Model Space drawings onto the paper sheet as defined in Paper Space. When printing the default viewport from Paper Space, you get a hard copy of the Plan view of the current drawing because the Plan view is the default view for Model Space.

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The sixth and final page enables you to save the settings for future drawing or to re store the initial settings. Settings can be edited in the TC Explorer Palette (see [The TC Explorer Palette](#)).

1.6. Opening and Saving Files

Within TurboCAD you can choose to open a saved file or start a new file. You can also open new and saved files from within TurboCAD, and there are several options for saving files.

Note: The **General** page of the **Program Setup** contains several settings for opening and saving files. See [General Setup](#).

Opening Files

Default UI Menu: File/Open

Ribbon UI Menu:

Hotkey: Ctrl+O

Files of type: By default, you will open a *.tcw file. Use this list if you want to open a file of a different format. See [Other CAD Formats](#).

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Open as read-only: A read-only file can be viewed only; it cannot be edited.

Description: Displays text entered in the **Subject** field of the **Summary Info** window.

See [Summary Info](#).

Show Preview: Displays a thumbnail of the drawing. Only TurboCAD files (*.tcw and *.tct) can display previews.

Setup: When importing files of other formats, provides access to conversion options.

Note: You can select multiple files from the file window to open at the same time. Simply hold down the Shift key to select multiple files.

Purge

Default UI Menu: File/Drawing Utilities/Purge

Ribbon UI Menu:

The purge tool is designed to make your files smaller by eliminating un-used elements that are stored in your file. For example, you can purge un-used blocks, line styles, or object styles.

Select **File | Drawing Utilities | Purge**. The Purge dialog will open.

Select the items you wish to purge. If necessary expand the tree so you can select sub-items. Alternately you can press the Purge All button.

After selecting your items to press the Purge button.

Press Close when you are finished.

Purge Options

Show Used Items- When this option is selected the tree will show the items in the tree. You cannot purge while this option is on. Selecting items in the tree will show purge parameters for that item type. The parameters are displayed in a text box directly below this option checkbox.

Purge default linestyles: If this option is turned on un-used default linestyles will be purged from the drawing. This option is off by default. It is recommended to use this option with caution.

Purge default brushstyles: If this option is turned on un-used default brushstyles will be purged from the drawing. This option is off by default, it is recommended to use this option with caution.

Clear all undo records before action: If this option is turned on all undo buffers will be deleted. This prevents issues with an attempt to undo purged items. This option is on by default.

Confirm each item to be purged: If this option is on you will be presented with a

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confirmation dialog for each item that is being purged from the drawing. This option is on by default.

Saving Files

Default UI Menu: File/Save

Ribbon UI Menu:

Hotkey: Ctrl+S

Saves the current drawing to disk. If the file was not previously saved, the **Save As** window will appear.

Save as type: By default, you will save the file as a *.tcw file. Use this list if you want to save (export) the file to a different format. See [Other CAD Formats](#).

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Setup: Enables you to set parameters for the saved *.tcs file (see [TurboCAD Formats](#)).
When exporting to another file format, provides access to conversion options.

Save:

Drawing: Saves the entire drawing.

Selection: Saves the selected objects only.

If **Prompt for Summary Info** is checked in the **General** page of the **Program Setup**, the **Summary Info** window will appear before saving.

Tile Area

TurboCAD splits a large image into pieces (horizontal stripes) in order to save memory while rendering, and then stitches these fragments into a single image.

The filter settings dialog bitmap (bmp, jpg, gif, png) now contains an option called **Tile Area**, which allows users (based on the amount of memory) to establish the image fragmentation settings

Minimal (default) value - 1 megapixel, corresponds to the image size of 1000 x 1000.

Maximal value - 256 megapixels, corresponds to the image size of 16000 x 16000

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(maximum resolution).

If you set "**Tile Area**" = 256, then for any size of image, fragmentation will not work.

Please note: When installing large values "Tile Area" is very likely to be used throughout the computer's memory which can lead to system instability.

Undo/Redo File Buffer

With this Save file option, the Undo/Redo buffer of a file is saved when the file is closed or when the application closes. The buffer is re-loaded when the file is reopened and can be used from the last point.

Some operations are not saved within the Undo/Redo buffer.

Summary Info

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Records general information to be saved with the drawing.

If **Prompt for Summary Info** is checked in the **General** page of the **Program Setup**, the **Summary Info** window will appear when you save the drawing.

File Name and Directory: Information for files that have already been saved.

Keywords: Text used when searching for this file.

Subject: Descriptive text that appears in the **Open** and **Save** windows. This text can also potentially be used by Windows file search utilities.

Tip: If a drawing is being saved as a symbol, the **Title** should be a short description of the symbol, and **Subject** contains a more detailed description. See [Loading an Individual Symbol into the Library](#).

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Save Preview Graphics with Drawing: Sets the type of graphic image that will be stored with the drawing, so that it can be previewed in TurboCAD and in other Windows programs.

None: No preview graphics will be saved.

Bitmap: Suitable for large files because the bitmap will use the same amount of space no matter how complex the drawing is.

Metafile: Suitable for small files because it displays more detail. Symbol previews are generally stored using this option.

Create Preview by: Saves the preview as the plan view or by the current view and render mode.

Time Stamp

The time stamp feature adds system information about the date/time of drawing creation, the date/time of the last modification, total time spent editing the drawing and the total time of the last editing session.

The information is available in "Summary Info" dialog (File | Summary Info...). The information is stored in the following file formats: TCW, DWG, DXF.

ePack

File|ePack

Default UI Menu: File/ePack

Ribbon UI Menu:

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The ePack utility is designed to encapsulate in one location all the external files associated with a drawing. This includes Xrefs, material images, linked images, fonts, and other referenced files.

Once activated you must specify the following:

Package type: This can be either a Folder or a Zip.

Include options: You may choose to include or exclude:

- SHX fonts - a standard for DWG files
- True Type Fonts - Standard for Windows files.
- Textures - Images which are used to create materials
- File References - Xrefs and all other referenced files

Action: With two options:

Save to folder - This is the location where the new folder or zip will be created.

Send via email - the entire packet is sent by email to the designated parties.

Lastly, you may use the Current Drawing tree and specifically select/deselect items that you wish to be packed. When you click the Save button the folder or zip will be created and all files, including the drawing, will be packed.

IFC Export and BIM



Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

IFC stands for Industry Foundation Classes (IFC). This is a standard data model used to describe data for the building and construction industries

It is a platform-neutral, open file format specification commonly used collaboration format for Building information modeling (BIM). The following Items are exported to IFC:

1. Architectural objects [Architecture Tools](#)

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Custom Properties [Custom Properties, Database, and Reports](#)

Property Sets [Property Set Definitions](#)

The data is mapped as follows:

TC Representation	IFC Element
	IfcBooleanResult
box	IfcBoundingBox
	IfcManifoldSolidBrep
sweep	IfcRevolvedAreaSolid
	IfcSolidModel
	IfcSweptAreaSolid
sweep	IfcSweptDiskSolid
line	IfcAxis1Placement
	IfcBoundedCurve
	IfcConic
	IfcElementarySurface
ellipse	IfcEllipse
	IfcGeometricRepresentationItem
	IfcPlacement
	IfcPoint
	IfcRepresentationItem
	IfcNamedUnit
	IfcPresentationLayerAssignment
	IfcColourSpecification
	IfcSimpleProperty
	IfcGeometricRepresentationContext
	IfcGeometricRepresentationSubContext
	IfcRepresentationContext
	IfcShapeModel
	IfcStyleModel

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	IfcLoop
	IfcTopologicalRepresentationItem
group	IfcGroup
	IfcObjectDefinition
	IfcPropertyDefinition
	IfcPropertySetDefinition
	IfcRoot
	IfcTypeObject
	IfcTypeProduct
	IfcBuildingStorey
	IfcElement
wall	IfcCurtainWall
beam	IfcMember
slab	IfcPlate
railing	IfcRailing
slab	IfcRamp
slab	IfcRampFlight
stair	IfcStairFlight
IFC Object	TurboCAD Object
IfcBeam	Beam
IfcColumn	Column
IfcProject IfcSite IfcBuilding IfcBuildingStorey IfcPropertySet(Pset_ProjectCommon) IfcPropertySet(Pset_BuildingCommon) IfcPropertySet(Pset_BuildingStoreyCommon)	Drawing
IfcWallStandardCase	Wall
IfcMaterial	Wall style
IfcOpening IfcWindow	Window
IfcWindowLiningProperties IfcWindowPanelProperties IfcPropertySet(Pset_WindowCommon) IfcPropertySet(Pset_DoorWindowGlazingType)	Window style
IfcOpening IfcDoor	Door

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IfcDoorLiningProperties IfcDoorPanelProperties IfcPropertySet(Pset_DoorCommon) IfcPropertySet(Pset_DoorWindowGlazingType)	Door style
IfcSlab	Slab
IfcMaterial	Slab style
IfcStair	Stair
IfcPropertySet(Pset_StairCommon)	Stair style
IfcRailing	Rail
IfcPropertySet(Pset_RailingCommon)	Rail style
IfcRoof	Roof
IfcMaterial	Roof style
IfcBuildingElementProxy	Landscape
IfcBuildingElementProxy	Other 3D objects
IfcPresentationLayerWithStyle	Layer
IfcPropertySet(TC_Pset_Custom)	Custom properties
IfcPropertySet(property set definition name)	Property set

IFC Import

IFC files can be imported. Geometry is converted to architectural objects where possible. Where not possible geometry is converted to grouped surfaces. To enable IFC Data

Go to Tools and select **IFC BIM Data**. The IFC BIM Data dialog appears

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- 1. Click the **Enable IFC BIM data.**
- 2. Click **OK.**
- 3. The **IFC BIM Data** dialog is open

This dialog Allows you to view add or alter IFC BIM data as needed.

Following features are now available

- Common
- Buildings
- BuildingStoreys
- Spaces
- Persons
- Organizations
- Applications
- Addresses

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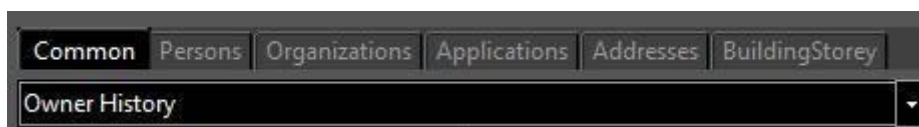
Clicking on some fields will present a Drop-down list.

Clicking on some fields will give you the option to Append or Remove a duplicate of the field

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Clicking on all other fields will allow you to enter data directly.

The pages of the dialog are sorted using the tabs at the top, and the drop-down list below the tabs.



The Tabs are: Common, Persons, Organizations, Applications, Addresses. The contents of the drop-down alter depending on which tab is selected. Here are all of the available pages:

Common

Persons

Organizations

Applications

Addresses

[Click here to see the ISO Standard for IFC data](#)

Building Storey

Object Data

Non-geometric data is mapped to custom properties for standard IFC data, and to Property Sets for expanded AEC data.

TC Representation	IFC Element
	IfcBooleanResult
Box	IfcBoundingBox
Set of curves	IfcGeometricCurveSet
Set of curves and surfaces	IfcGeometricSet
	IfcManifoldSolidBrep
Sweep	IfcRevolvedAreaSolid
	IfcSolidModel

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Sweep	IfcSurfaceCurveSweptAreaSolid
	IfcSweptAreaSolid
Sweep	IfcSweptDiskSolid
Line	IfcAxis1Placement
	IfcBoundedCurve
	IfcConic
	IfcElementarySurface
Ellipse	IfcEllipse
	IfcGeometricRepresentationItem
	IfcPlacement
	IfcPoint
	IfcRepresentationItem
	IfcNamedUnit
Material	IfcSurfaceStyleRendering
	IfcPresentationLayerAssignment
	IfcColourSpecification
Curve	IfcArbitraryOpenProfileDef
Region	IfcArbitraryProfileDefWithVoids
	IfcArbitraryClosedProfileDef
Polyline	IfcAsymmetricIShapeProfileDef
Polyline	IfcCShapeProfileDef
Polyline	IfcCircleHollowProfileDef
Ellipse	IfcEllipseProfileDef
Polyline	IfcIShapeProfileDef
Polyline	IfcLShapeProfileDef
	IfcParameterizedProfileDef
	IfcProfileDef
Region	IfcRectangleHollowProfileDef
Polyline	IfcRoundedRectangleProfileDef
Polyline	IfcTShapeProfileDef

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Polyline	IfcUShapeProfileDef
Polyline	IfcZShapeProfileDef
	IfcSimpleProperty
	IfcGeometricRepresentationContext
	IfcGeometricRepresentationSubContext
	IfcRepresentationContext
	IfcShapeModel
	IfcStyleModel
	IfcLoop
	IfcTopologicalRepresentationItem
Group	IfcGroup
	IfcObjectDefinition
	IfcPropertyDefinition
	IfcPropertySetDefinition
	IfcRoot
	IfcTypeObject
	IfcTypeProduct
	IfcBuildingStorey
	IfcElement
Wall	IfcCurtainWall
Beam	IfcMember
Slab	IfcPlate
Railing	IfcRailing
slab	IfcRamp
Slab	IfcRampFlight
Stair	IfcStairFlight

IFC BIM Data and BIMTool

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(Available in Platinum)

Default UI Menu: Tools/IFC BIM Data

Ribbon UI Menu:

IFC stands for **I**ndustry **F**oundation **C**lass, and these are the international standards for types of objects used in CAD software, design, and BIM (Building Information Management) software.

By enabling IFC BIM data from an imported drawing, you can to use the property sets that are part of IFC standard.

You can choose property set from the Selection Info palette, under the General option.

You can also set or retrieve data about the IFC file itself, including the creator and the owner of the file, and the organization associated with the creator.

Contact Manager

Default UI Menu: File/Share/Run Contacts Manager

Ribbon UI Menu:

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The Contact Manager is used to hold a list of contacts you can use to share your files with.

Add New Contact

Clicking the Add New Contact button opens the dialog.

Enter data into all of the relevant fields.

Select the appropriate groups for the contact.

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If an image is available click the Load Image button, navigate to the image, select it, then click OK.



4. Click OK to finalize the Contact.

Edit Contact

Clicking the Edit Contact button opens the dialog for the currently selected contact.

Enter data into all of the relevant fields.

Select the appropriate groups for the contact.

If an image is available click the Load Image button, navigate to the image, select



it, then click OK.

4. Click OK to finalize the Contact editing.

Delete Contacts

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To delete a contact, select all of the contacts you wish to delete. Then click the Delete Contacts button. You will be prompted to confirm the deletion. Click Yes to confirm the deletion.

Import Contacts

Clicking the Edit Contact button opens the dialog for the currently selected contact.

Contact Info

This shows the information for the currently selected contact. If multiple contacts are selected it will show the last contact selected.

Contact List

This shows the list of all of your contacts.

Search

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Searching is automatic. Just type into the Search field.

Once a search is entered you can clear it by clicking the X in the Search field.



Sort

This sorts the contact list into alphabetical order. If pressed again it will sort the list into inverse alphabetical order.

Show Groups

This flips the interface from Contact mode into Group mode.

Add Group

Clicking the Edit Contact button opens the dialog for creating a group.

Type in a name for the Group and click OK.

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To add a contact to a group, edit the contact and select the required group in the Groups list.



Delete Groups

To delete a contact, select all of the contacts you wish to delete. Then click the Delete Contacts button. You will be prompted to confirm the deletion. Click Yes to confirm the deletion.

Import Contacts

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Clicking the Edit Contact button opens the dialog for the currently selected contact.

Group/Contact Info

This shows the information for the currently selected group or contact. If multiple contacts are selected it will show the last contact selected.

Group and Contact List

This shows the list of all of your Groups and Contacts.

Search

Searching is automatic. Just type into the Search field.

Once a search is entered you can clear it by clicking the X in the Search field.



Sort

This sorts the contact list into alphabetical order. If pressed again it will sort the list into inverse alphabetical order.

Show Only Contacts

This flips the interface from Group mode into Contact mode.

Share

Default UI Menu: File/Share/Share with Contacts

Ribbon UI Menu:

The Share tool uses the Contact manager to send your current file to your contacts.

To share a file:

Select the contacts that will receive the file.

You will then be prompted for the manner in which the file will be sent: **TCW** - sends a standard TurboCAD file **JPG** - sends a screenshot image. **DWF** - sends a Design Web Format **PDF** - sends a Standard PDF version of the CAD file. **ePack** - Uses the ePack utility to zip up all the necessary components into ePack

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file. See [ePack](#) **No Attachment** - Sends an empty email to the designated contacts. This allows you to just send an email.

3. Then click OK.

1.7. Importing and Exporting Files

You can import and export data from other TurboCAD formats, as well as formats of other CAD systems. To import from another file type:

Select **File / Open** or Ctrl+O.

Under **Files of type**, select the format you want to import.

If you want to specify conversion parameters, click **Setup**.

Browse to the file, and double-click it or click **Open**.

To export to another file type:

Select **File / Save As**.

Under **Save as type**, select the format to which you want to export.

If you want to specify conversion parameters, click **Setup**.

Browse to the folder and assign a file name.

To import or export only certain components of a file, use **File / Extract From** or **Ext ract To**. For example, you can choose to import a drawing's layers and blocks, or export only print styles. To insert the contents of another file as a block, create an external reference (xref). See [External References](#)

For a list of all supported CAD and Graphics file formats, see [Other CAD and Graphics Formats](#)

File Converter

(Plugin)

Opens the **TurboCAD File Converter** window, in which you can select a source and destination folder for batch conversion. Use the **Files of type** field to select the conversion format. Click **Run** to convert all files in the selected folder and place the converted files in the target folder.

Other CAD and Graphics Formats

The following formats can be read by TurboCAD, either for import, export, or both. Some formats have a **Setup** in which you can set certain parameters for conversion. If you do not specify conversion parameters, the defaults will be used. (If no **Setup** is indicated, the conversion requires no additional input.)

Import Only ✓ *	Export Only ✓ **		
Platinum	Professional	Deluxe	Designer
✓	✓	✓	✓
✓	✓	✓	✓
✓	✓	✓	✓
✓			
✓	✓	✓	
✓ *	✓ *	✓ *	✓ *
✓			
✓ *			
✓ **	✓ **	✓ **	✓ **
✓	✓	✓	✓
✓	✓ **	✓ **	
✓ *	✓ *	✓ *	✓ *
✓	✓	✓	
✓	✓	✓	

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✓ *	✓ *	✓ *	
✓	✓	✓	
✓ *	✓ *		
✓ **	✓ **	✓ **	✓ **
✓	✓		
✓			
✓ **	✓ **	✓ **	✓ **
✓ **			
✓ *	✓ *	✓ *	
✓	✓		
✓ **	✓ **	✓ **	✓ **
✓ **			
✓ **			
✓	✓	✓	✓
✓ **	✓ **	✓ **	✓ **
✓ **			
✓	✓	✓	✓
✓			
✓			
✓ **	✓ **		
✓	✓	✓	
✓			
✓	✓	✓	
✓			

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SVG	Scaleable Vector Graphics	✓ **	✓ **	✓ **	
TAP	TurboApps® packaged file format	✓ *	✓ *		
TCT	TurboCAD® drawing template	✓	✓	✓	✓
TCW	TurboCAD® native format	✓	✓	✓	✓
U3D	Universal 3D	✓	✓ **		
WMF	Windows® Metafile	✓	✓	✓	✓
WRL	VRML Worlds	✓	✓	✓	✓
WRZ	VRML Worlds	✓	✓	✓	✓
XYZ	Terrain Data	✓ *	✓ *		
3MF	3D Manufacturing Format	✓ **	✓ **	✓ **	
ASM	Pro/E/Creo/Solid Edge Assembly	✓ *			
CATPART	CATIA V5/V6	✓			
CATPRODUCT	CATIA V5/V6	✓			
IAM	Autodesk Inventor Assembly File	✓ *			
3DXML	3DXML	✓ **			
IPT	Autodesk Inventor Part File	✓ *			
MODEL	CATIA V4	✓ *			
PAR	Solid Edge Part	✓ *			
PRT	UG NX/Pro/E/Creo	✓ *			
SLDASM	SolidWorks Assembly	✓ *			
SLDPRT	SolidWorks Part	✓ *			
VDA	VDA-FS	✓			
X_B	Parasolids binary	✓ *			
X_T	Parasolids text	✓ *			

Note: IGS is a file format for a public domain called IGES which is actually a neutral file format intended as an international standard for the exchange of data between different

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CAD/CAM systems. IGES is an ANSI standard maintained by the IGES/PDES Organization (IPO), under the direction of the National Institute of Standards and Technology (NIST). The National Computer Graphics Association (NCGA) acts as the administrator of the standard.

SKP File Import Setup

Use "ByLayer" color for entities without material: Assign the SketchUp layer color to any entity that does not have an assigned material.

Import Hidden Entities: Imports entities which are hidden in the SKP. If this option is off hidden entities will not be imported.

Import dynamic components: Dynamic block are imported as PPM objects.

SKP materials are imported and supported for both RedSDK and Lightworks.

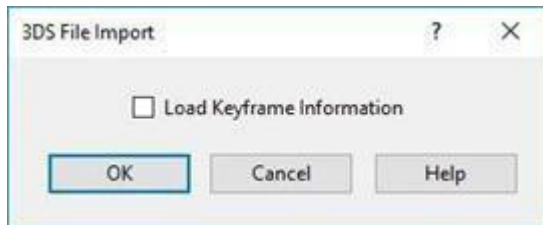
SKP File Export Setup

SketchUp Version: Specify the version of SketchUp SKP to be used in the file format: 5, 6, 7, 8, to 2023.

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Note: The 2023 format is only supported on 64-bit versions of TurboCAD.

3DS File Import Setup



Load Keyframe Information: Accepts or ignores the keyframe 3DS file data.

Materials can also be imported in 3DS format.

Additions made to import RedSDK materials.

Issue with lost materials is now fixed.

Issue with lost texture coordinates is now fixed.

Without Texture:

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With Texture:

Demo Video:

<https://youtu.be/LHfhtw6TAK>

3DS File Export Setup

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Preserve loaded 3DS materials: Saves unique materials from a 3DS file. If not checked, saves TurboCAD materials.

Save Keyframe: Saves the hierarchical structure of entities in a file. Do not check this option for 3DS files that you want to use as 3DS symbols in TurboCAD.

BMP Export Setup

Output Range: Creates an export file either out of the entire drawing (**All**) or only the part that is currently in view (**View**).

Use Render: Available only when **Render** or **Hidden Line** mode is active. If checked, the exported image will show the render displayed on the screen.

Size of Rectangle: Sets up the resolution for the exported graphics. By default, these values are 400 x 400 but you can customize the height and width

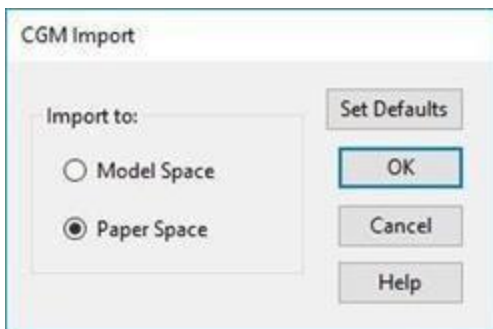
Maximum resolution: The drawing will be defined by a rectangular matrix of 2000 x 2000 logical pixels. If there is at least one image object, this is set to 1000 x 1000. **Keep View**

Rectangle Aspect Ratio: Keeps the same ratio of length to width.

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Note: You can determine an object's type by using the Selection Info Palette.
See [Selection Info Palette](#).

CGM Import Setup



Choose to open the file either in **Model Space** or **Paper Space**.

DGN Import Setup

View Number: By default, the view number is 1.

Line Widths: You customize a set of drawing line widths identified by the indexes (0, 1, 2, etc.). To do this, use the two-column **Line Width** control, where the **Index** column provides a reference to particular line width, and the **Width** column is a data entry field. You can scale the line width values, using the **Width Scale** data entry field.

Fonts: As with the drawing line widths, the import file's fonts are indexed. For every index in the **Index** column, you can select a font from the corresponding drop-down list in the **Font** column. In addition to the options of the **Fonts** control, there is the **Default**

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Font entry field whose drop-down list lets you select a common font for the ones you haven't changed.

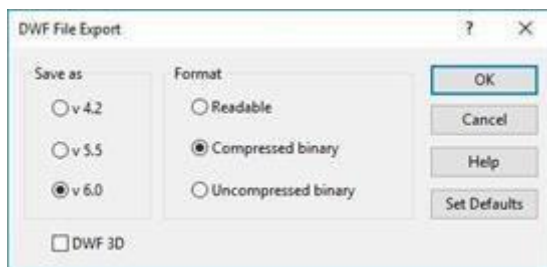
Set Defaults: Instead of customizing the imported file options, you can apply the default settings for the DGN file import. To do this, click **Set Defaults**.

DGN Export Setup

Version: Choose between Versions 7 and 8.

Default font index to export: The DGN format requires the export file's fonts to be indexed. For every listed font, you can select an index. In addition to the options of the **Fonts** control, there is the **Default Index** entry field whose drop-down list lets you select a common index for the fonts you have not changed.

DWF Export Setup



Output Format: If **Readable** is selected, you can later open the exported file to view and/or edit via a text editor.

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Skip Invisible Layers: Invisible layers will not be included in the export file.

DWF 3D: DWF files of 6.01 format will be created. Files will contain all elements from Model Space, including 3D.

Note: TurboCAD does not support importing from this format. DWF files created with this option can be viewed either in Autodesk DWF Viewer (v6.01 and up), or in other applications that support this format.

DXF (DWG) Import Setup

Open mode:

Open without audit: Opens the file without checking it. If the file has errors, it will not be opened.

Open with audit: Opens the file and corrects errors when possible. The report is displayed only if errors are found. The file is checked is after it is loaded into memory, though if the file is very corrupt it may not be able to be loaded into memory.

Recover: Opens the file and corrects errors. The report is automatically displayed, regardless of errors. The file is checked before loading it into memory. **File Units:** Replaces the units of an imported file with the selected units.

Default text font: Sets a default import font.

Lineweight Unit and Value: Defines how objects using the AutoCAD default line weight

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are treated on import.

Overwrite existing entries for: AutoCAD stores the settings (or values) for its operating environment and some of its commands in system variables. Checking this box overwrites these settings. In TurboCAD, these settings include World units, the numerical display format, the angular system, etc.

You can overwrite system variables for objects under the following conditions: the DXF/DWG drawing is inserted into an open file, and both files (the imported file and the receiving file), have system variables (views, blocks, layers and/or line styles) of the same names.

DWG and DXF file formats do not support unit types, only a designation as Imperial or Metric. All Imperial units are converted to inches in DWG, all Metric units to mm.

DXF (DWG) Export Setup

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Save As: Select the AutoCAD version. TurboCAD now support DWF export till 2023.

Precision: The number of decimal places for numerical values of the exported file.

Convert Xrefs to AutoCAD Drawing: Converts attached Xrefs into DWG files.

Rendered Viewports as Images: All viewports that have renderings will be converted to images, to preserve the appearance of the renderings.

Rewrite embedded images: Relevant for files that contain image objects.

Choose whether to replace images.

Groups as Blocks: Displays objects included in groups as blocks when imported into AutoCAD.

Unblock: Explodes blocks into its constituent objects.

Preserve Hatch Associativity: Retains hatch association in the resulting DWG file.

Explode Text: Explodes any text into its constituent parts.

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Explode Architectural Objects for DWG: Explodes any TurboCAD Architectural objects into data comprehensible to AutoCAD.

Convert By World Scaled Arcs into Polylines: All arc using the By World setting for line scaling will be converted into arc segments of polylines with a supported AutoCAD scaling method.

Set Fixed Height for Text Styles: (On, Off, Don't check this option): Locks Height for Text Styles.

DWG and DXF file formats do not support unit types, only a designation as Imperial or Metric. All Imperial units are converted to inches in DWG, all Metric units to mm.

EPS Import Setup

Import to: Choose whether the imported file will open in Model Space or Paper Space.

EPS Export Setup

Explode Text: Explodes any text into its constituent parts.

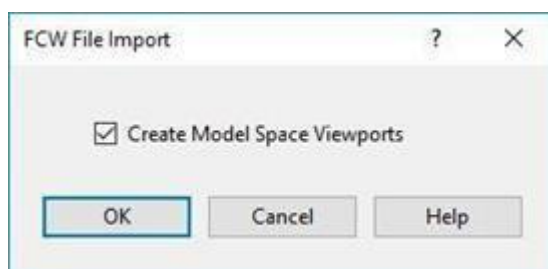
Save block as procedure: Saves the drawing's blocks as procedures in the PostScript language.

Resolution: Sets the horizontal and vertical resolutions for the two-dimensional presentation of your drawing. Check **Max Resolution** to set the maximum 1600x1600 resolution.

Output file: Exports to a binary or readable format.

Header: Specify whether to include a preview image, and in what format.

FCW Import Setup



Create Model Space Viewports: Specify whether to create viewports for the Model Space presentation of your drawing.

FBX Import / Export

FBX loading and writing is totally automatic.

This file format now supports textures and materials.

GIF Export Setup

Optimized: Color palette optimization. If not checked, the initial color palette will imitate the Halftone palette, in which the color of a pixel is adjusted by mixing colors of adjacent pixels. If checked, the initial color palette will be optimized, containing no more colors than the number specified in the **Maximum Colors** field.

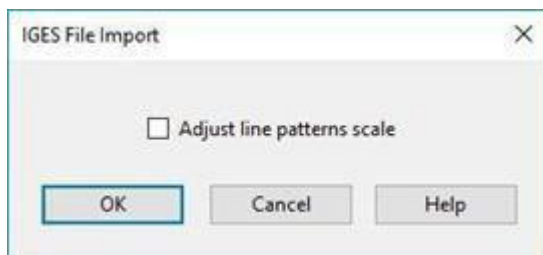
Maximum Colors: Number of colors in the optimized palette (from 2 to 256).

Grayscale: The initial color palette will contain shades of gray.

Transparency Color: Sets the color that will be transparent. Available for the optimized palette.

Layout - without Margins: Uses only the area within the margins to generate the GIF when the image is generated from a layout.

IGS Import Setup



Adjust line pattern scale: Line style for improving line display. Because IGS files can be created in a variety of ways, either option (on or off) may work to improve precision during import.

JPG Export Setup

Output Range: Creates an export file either out of the entire drawing (**All**) or only the part that is currently in view (**View**).

Use Render: Available only when **Render** or **Hidden Line** mode is active. If checked, the exported image will show the render displayed on the screen.

JPEG quality factor: Your goal is generally to pick the lowest quality setting, or smallest file size, that decompresses into an image indistinguishable from the original. This setting can vary, but here are some rules of thumb. For good-quality, full-color source images, the default quality setting (Q 75) is very often the best choice. This setting is about the lowest you can go without expecting to see defects in a typical image. Try Q 75 first; if you see defects, then increase. If the image was less than perfect quality, to begin with, you might be able to drop down to Q 50 without objectionable degradation. On the other hand, you might need a higher quality setting to avoid further loss. This is often necessary if the image contains dithering or moire patterns. Except for experimental purposes, avoid values above Q 95; Q 100 will produce a file two or three times as large as Q 95, but with negligible quality improvement.

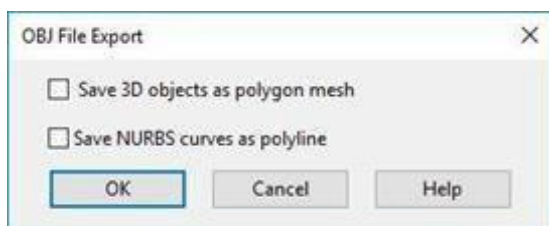
Size of Rectangle: Sets the resolution for the exported graphics. By default, these values are 600 x 800 but you can customize the height and width

Maximum resolution: The drawing will be defined by a rectangular matrix of 2000 x 2000 logical pixels. If there is at least one image object, this is set to 1000 x 1000.

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Note: You can determine an object's type by using the Selection Info Palette. See [Selection Info Palette](#).

OBJ Export Setup



Save 3D objects as polygon mesh: When 3D objects are converted into a mesh, smoothness may be lost, but the exact appearance will be preserved.

Save spline curves as polyline: When NURBS curves are converted to polylines, smoothness will be lost, but the exact appearance will be preserved.

PDF Export Setup

Embedded fonts: Fonts will be embedded in the PDF file, increasing the file size. If not checked, the PDF viewer (such as Adobe Reader) will select a font that exists in the

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system.

True type as geometry: True-type text will be exploded into a collection of lines.

SHX text as geometry: SHX text will be exploded into a collection of lines.

Enable simple geometry optimization: When checked, smaller PDF files will be created.

This may affect the PDF quality.

Zoom to extents mode: Zooms into the closest point, showing the full model.

Enable layer support (pdf v1.5): Add layers to the PDF. You can also opt to export Off layers as well.

Workspaces to Export: Select Active to export only the active Space or All to export all Paper Spaces and Model Spaces.

Paper Width: set the paper Width of the PDF.

Paper Height: set the paper Height of the PDF.

PDF (Adobe PRC Based)

Export setup is fully automatic.

PDF (Adobe U3D Based)

Export setup is fully automatic.

Export named and standard views.

Publish to PDF

(Available in TurboCAD Platinum, Professional, and Deluxe)

Default UI: File/Publish to PDF

Ribbon UI:

Publish to PDF Combine functionality of 2D and 3D PDF filters have been integrated into a single tool

“Publish to PDF”, which has been added to the File menu.

This tool supports all features currently present in PDF filters with the following options: -Model space - as 2D/3D with 3D options setup -Watermarks Setup for exported PDF -Paper spaces - Viewports as 2D/3D - Select separate Paper spaces in active drawing to export -Select separate spaces in several open drawings to export.

Pages:

This tab allows the user to select the pages from the drawing to be exported to PDF.

Settings:

This tab provides settings for the pdf to be exported including settings for text, zooming, hyperlinks, layers export, vector resolutions, and color policy.

3D Settings:

User can adjust 3D settings for exporting 3D PDF via this tab. This includes options to select/deselect 3D Model and 3D Viewports. Scene views and Scene hierarchy selections can also be made via this tab.

Info:

This tab lets the user to enter PDF file information including Title, Author, Subject, and Keyword s.

Watermark:

This tab provides settings for placing a watermark in PDF export. Watermark settings can be applied including watermark text, text position, opacity and font formatting.

PNG Export Setup

Bits Per Pixel: The number of bits per pixel in the saved image. A value of 8 means a 256-color palette; a value of 24 means True Color palette. **Gray Scale:** The initial color palette will contain shades of gray.

Maximum Colors: Number of colors in the palette (from 2 to 256). Available if **Bits Per Pixel** = 8.

Color: Selection of transparency color. When **Bits Per Pixel** = 8, the pixels that have this

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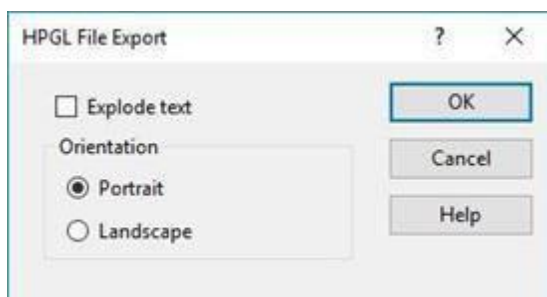
color will be transparent. When **Bits Per Pixel** = 24, the pixels that have this color will get the level of opacity specified in the **Opacity Level** field. If **Color** is off, the level of opacity will be set for all pixels of an image.

Opacity Level: Level of opacity (alpha channel: from 0 to 255). **Available if Bits Per Pixel** = 24.

PLT Import Setup

Import to: Choose whether to open the imported file in Model or Paper Space.

PLT Export Setup



Explode text: Explodes any text into its constituent parts.

Orientation: Sets the paper orientation to **Portrait** (vertical) or **Landscape** (horizontal).

SAT Import Setup

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Mode: Choose one of the modes for representing SAT objects.

Default: The choice of the polygon type is determined by the surface type in the solid object model (planar, cylindrical, spherical, toroidal, etc.)

3, 4, n-sided polygons: The number of sides of polygons used for object representation. If you wish to check what type of polygon is used for the object presented in an imported SAT file, do the following.

Once the SAT file has been imported, select an object in the drawing, make a block of it, and bring this block onto the paper of the TurboCAD desktop.

Open the **Properties** window for this block, open the **Block Insertion** page, and assign non-zero values to the rotation coordinates.

When a 3D view of your object is displayed, select the object and explode it.

After this, you can display individual elements (polygons) of the object by selecting the different parts of objects.

Existing ACIS SAT file import/export has been updated to support curves such as lines, arcs, circles, ellipses, and splines.

Previously in TurboCAD, ACIS file formats (SAT and SAB), only 3D objects were supported, along with their colors. However, now 2D entities allow the user to import the entire ACIS drawing. Geometry and Color are both supported for 2D entities.

SAT Export Setup

Select the SAT version for export.

SAB Import Setup

The Standard ACIS Binary (SAB) format is used by the ACIS modeling kernel from Spatial to share CAD data. Choosing SAB is especially beneficial when sharing with other ACIS based CAD systems where data is shared and not recalculated between different modeling kernels. TurboCAD supports both reading and writing SAB files.

Mode: Choose one of the modes for representing SAT objects.

Default: The choice of the polygon type is determined by the surface type in the solid object model (planar, cylindrical, spherical, toroidal, etc.)

3, 4, n-sided polygons: The number of sides of polygons used for object representation. If you wish to check what type of polygon is used for the object presented in an imported SAT file, do the following.

Once the SAT file has been imported, select an object in the drawing, make a block of it, and bring this block onto the paper of the TurboCAD desktop.

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Open the Properties window for this block, open the Block Insertion page, and assign non-zero values to the rotation coordinates.

When a 3D view of your object is displayed, select the object and explode it.

After this, you can display individual elements (polygons) of the object by selecting the different parts of objects.

SAB Export Setup

Select the SAB version for export.

SHX Export Setup



SHX Export Quality Factor: A high value creates shapes that more accurately represent the selected geometry. However, large values tend to create shapes that contain more coordinate data and can be a processing burden during regeneration. Low values create less accurate shapes that are more quickly processed. You may need to experiment with different values.

Note: SHX export does not create an *.shx font file. Once created, the results cannot be edited.

STL File Export Setup

Binary: Saves to a binary file.

Color Binary: Saves to a color binary file.

ASCII: Saves to an ASCII text file.

3D Print Check:

A new 3D Print Check option has been added when exporting an STL file. This tool checks part for the following common topological errors such as:

- non-manifold edges

- duplicate facets

- collapsed facets

- consistent normal orientation

- open edges

- shells

The tool also displays geometry information such as extents, number of facets/vertices, area, and volume. When saving an STL file, the user is given the option of performing a print check on the 3D model.

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After saving the file, A dialog will appear for each object giving general information of the object as well as a result of 3D print tests. Please find the attached reference images.

Demo Video:

<https://youtu.be/diFs5v1V3oo>

WMF Import Setup

Prohibition of palette change: Prevents TurboCAD from changing the color palette.

Insert to extents of: Sets the size of the imported drawing.

View: The size is based on the current view.

Drawing: The size is based on the entire drawing.

Page: The size is based on the paper sheet.

Custom: Define your own size.

Custom extents: Enter coordinates and dimensions of the rectangle that encloses the drawing.

Open in Model Space: Opens the imported drawing in Model Space. The drawing is placed on the default **By World** workplane.

WMF Export Setup

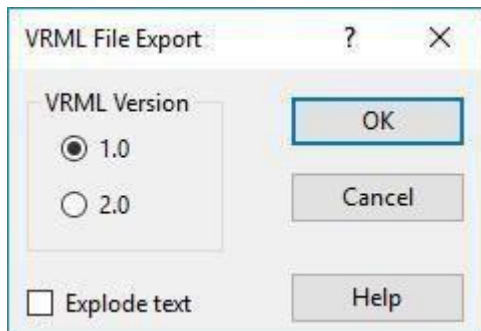
Output Range: Creates an export file either out of the entire drawing (**All**) or only the part that is currently in view (**View**).

Size of Rectangle: Sets the resolution for the exported graphics. By default, these values are 400 x 400 but you can customize the height and width

Maximum resolution: The drawing will be defined by a rectangular matrix of 8000 x 8000 logical pixels. If there is at least one image object, this is set to 1000 x 1000.

Note: You can determine an object's type by using the Selection Info Palette. See [Selection Info Palette](#).

VRML Export Setup



Select a version of VRML file to save. You can also explode the drawing text while creating the file for export.

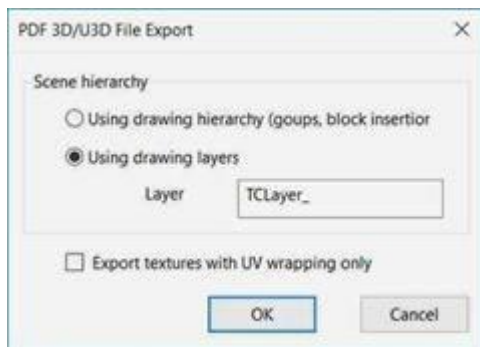
3MF Export Support:

TurboCAD supports import and export of the 3MF file type used in manufacturing. This includes support of Textures.

U3D Export Setup

Give an allow to control layer prefixes: Now each layer in U3d file has predefined prefix: "TCLayer_". User wants to change (edit) it.

Save drawing hierarchy in U3D file. In this case, Adobe pdf reader shows it.



Export the drawing hierarchy into a U3D file (groups, block insertions):

User will have the ability to choose export the drawing hierarchy or layers into a U3D file.

User will have the ability to change the layer prefix.

This options will be available from the export setup dialog.

Demo Video:

<https://youtu.be/tCNR4wz6EUo>

Following CAD Formats have been Improved

OBJ filter. Import materials

Added import RedSDK and Lightworks materials.

Supported features:

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diffuse color

transparency

reflection

texture map

bump map (RedSDK only)

reflection map (RedSDK only)

Support of Rhino 5 f iles (3DM) f iles

The latest openNURBS toolkit is incorporated to TurboCAD 2023.

SAT/SAB/IGES/STEP - Support for Interop Face Color

When reading ACIS objects (SAT/IGES/STEP/SAB import), if ACIS object has colored faces, then the appropriate material is assigned to these faces.

Import STEP format improvement

Import text attributes from a STEP file

User can access the text information and would like to read it out with VBA (TurboCAD-VBA-Object-Catalog) and store the text into the “Info” Property.

User can now view the information about the object used in the file. The information will be displayed in the ‘info’ field of “selection-info” palette.

Note : It supports both .STEP & .STP and .SAT files as well.

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New IFC Object types now supported in TC2023

Please find the list from the below link:

<https://docs.google.com/spreadsheets/d/14HbZ25DpZJcumg7IT0cfSbOF5xYwVQZQ5jEkhlofew/edit#gid=0>

KMZ and KML

When a KMZ or KML file is imported it may include a Google map. Any data points will be represented by push-pin indicators which are purely a representation of the

geolocation of the data point.

TurboCAD Formats

There are three TurboCAD file formats, TCW, TCT, and TCX, that you can use for saving vector graphics from within TurboCAD.

TCW: (TurboCAD for Windows) - a file format for saving vector graphics from within TurboCAD.

Note: If you need to save a drawing that is to be opened by an earlier version of TurboCAD, you can use Save As and save to one version back i.e. In TurboCAD 20 You can save as TurboCAD 19.

TCW, TCT Import Setup

Restore render view: Opens the drawing in render mode. The file must have been saved in render mode.

TCW, TCT Export Setup

Compression: Saves the file in the TCT compressed format. If the speed of operation is more important than the file size, leave this option disabled.

Full Precision: TurboCAD stores your objects with the maximum possible accuracy. **Full Precision** enables the maximum depth of mathematical calculations performed to generate and save entities. When disabled, entities will be calculated and saved using only four decimal points of precision. This results in faster execution and smaller files but may compromise accuracy.

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Note: Less than full accuracy can affect the ability of the program to correctly display and edit objects at extremely small scales. Unless you are using a slow machine, and know you do not need extreme accuracy, you should leave **Full Precision** on.

Save proxy objects: saves ACIS solids and a number of other TurboCAD Pro object with proxies so that TurboCAD Deluxe or Designer can open the file and see representations of objects that are not supported by those applications. Proxies increase the size of the file, but do allow file to open more quickly.

Save Undo: The Undo/Redo performed in the file are also saved when this option is enabled in setup while exporting.

A progress meter reports the progress while reading/writing STEP and VDA files.

1.8. AutoUpdate

The AutoUpdate feature is designed to allow occasional updates to the application. This allows for ongoing bug fixes, data updates, minor feature deployment and feature enhancements.

Internet access is required for AutoUpdate to function.

When a update is available you will be prompted when the application opens.

Select the Download Now link to begin the download. The IMSI/Design Update Center runs from your system tray, so you can opt to download even if the application is closed.

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The update center will track the download and the time.

You can stop the download at anytime by selecting the Stop Download link.

Once the download is complete you will be prompted to Install the update by clicking the Install Now link. You will have to close the application before running the installation of

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the update.

1.9. TurboCAD Classic UI in TurboCAD Pre 20

When you start TurboCAD 27 for the first time, you will notice that the program user interface is now a dark grey in color and that the many of program icons have been re-worked and re-colored.

If you prefer to work with the 'Classic' user interface found in v18 or earlier, it is quite simple to change back to this.

Please follow these steps:

Click on 'Tools' then 'Workspace' and select the 'Customize' option.

When the Customize menu appears, Click on 'Styles and Themes'.

Tick the first radio button 'TurboCAD Classic', then click 'Close' - you will see the interface change to the familiar, lighter grey color.

Click 'Close' a final time to exit from the Customize menu - the Classic interface should now fully appear.

Optionally you can use the slider to set the icon size to 16 x 16 by sliding it all of the way to the left

2. User Interface

This section provides information on your working environment in TurboCAD. In addition to descriptions of the UI and its components, information is provided about program and drawing setup, customization, and windows.

Object properties, and the various means of setting and changing properties, are also covered.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

2.1. Default User Interface

Default User Interface:

(Available in all Variants)

Note: The screenshots below are taken from TurboCAD Platinum other variants will contain less menus and tools

File Menu:

Edit Menu:

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View Menu:

Insert Menu:

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Format Menu:

Tools Menu:

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Draw Menu:

Dimension Menu:

Constraints Menu:

Architecture:

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Modify Menu:

Modes Menu:

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SDK Menu:

AddOns Menu:

TurboCAD 2023 User Guide Publication

Ruby.Net Scripts Menu:

Options Menu:

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Window Menu:

Help Menu:

2.2. Ribbon User Interface

Ribbon User Interface:

A new "Ribbon" interface is now included in TurboCAD 2023. The Ribbon interface is optional . The Ribbon interface can be selected the first time you use TurboCAD through a Pop-up which appears at the start-up of TurboCAD (See below) from where user is asked to choose the User Interface. User Interface setting can be changed either via this Pop-up, if left enabled, or through the Customize tool in Page Setup of the Workspace Menu of the Ribbon.

Note: If you subsequently install an optional TurboCAD Plug-in such as Animation Lab or CAM, a new top level menu item called 'Add-ons' will appear. The plug-in may be accessed from the Add-ons menu

File menu:

Edit menu:

View menu:

Workspace menu:

Draw menu:

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Insert menu:

Architecture menu:

Format menu:

Tools menu:

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Modify menu:

Add-Ons menu:

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Modes menu:

Option menu:

Window menu:

Help Menu

2.3. CADCourse User Interface

CADCourse User Interface

(Available in all Variants)

Standard

Line

Arc

Dimension

Text

Architecture

3D Object

3D Tools

Modify

Copy

Constraints

Desktop

Align

Tools

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Options

View

Camera

Insert

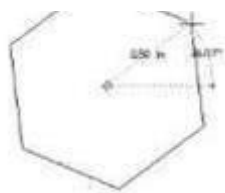
2.4. Using LTE Dynamic Input



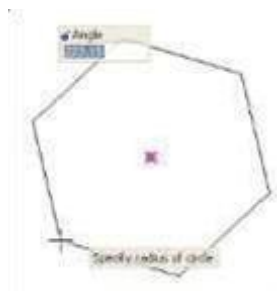
Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

This section covers dynamic dimensions and dynamic input fields.

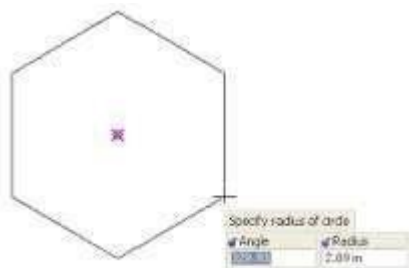
Dynamic dimensions inform you about the size and location of an object before it is created. These are the same values that appear in the Inspector Bar. You can display one, two, or all available dimensions.



If you choose to display one or two dynamic dimensions, the order of dimensions follows the fields in the Inspector Bar. For example the first field for the **Polygon** tool is **Angle**.



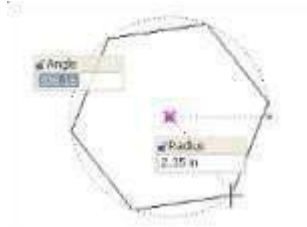
Input fields enable you to enter geometric data in the drawing window, instead of the Inspector Bar. In this example, Input fields are displayed next to the cursor, and dynamic dimensions are turned off.



If you don't want the input fields to follow the cursor, you can choose to place them at the bottom of the drawing window. Input fields and dynamic dimensions can work in tandem. If both are displayed, and you choose to place input fields on the dynamic

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dimensions, you will see input fields in place of the dimension itself.



Any input field that does not have a corresponding dynamic dimension, like the **Sides** field above, will be placed near the cursor, or at the bottom of the drawing window, depending on the option you select. The **Customize** window also has options for controlling the transparency, width, and title bar display of the input fields. To use the input fields, you can place your cursor in them. Or you can press Tab to enter the first field, then press Tab to jump from field to field.

Tip: If you enter a command alias in an input field, it is the same as entering the alias in the Command Line.

2.5. Menu Navigation

The two following maps show the menu structure to illuminate navigation.

Menu Structure

This menu structure is for Default User Interface

File

New

Open

Close

Extract From...

Save

Save As...

Save All

ExtractTo

ePack...

Publish To HTML...

Page Setup...

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Page Setup Wizard...

Print Styles...

Print Style Manager

Underlay Manager

Print Preview...

Print...

Print Queue...

Summary Info...

Store Thumbnail

Send...

Share ►

Share with Contacts

Run Contacts Manager

Drawing Utilities ►

Purge...

Recent File

Exit

Edit

Undo

Redo

Cut

Copy

Copy In Place

Paste

Paste Special...

Isolate Objects

Hide Objects

End Isolation

Select

Select All

Select By ►

Query...

Entity Type...

Color

Layer...

Attribute...

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Fence

Similar...

Similar Options...

Select by Similar

Select By Similar - Shape

Clear ►

Selection

All

Construction

All Constructions

Links...

Edit Tool

Highlight End Points

2D Edit Mode

View

Redraw

Regen

Model Space

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Zoom ►

Previous View

Zoom Window

Selection

Zoom In

Zoom Out

Full View

Extents

Printed Size

Uniform for all views

Pan ►

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Middle

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2.6. Command Line



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum and Professional)

Note: The Command Line allows for support of the Dark UI theme, and for scaling of the UI to better support visibility of icons and menus when using high resolution, 4K monitors.

The Command Line enables you to activate a command by typing its alias, and/or to input geometric data or command information by typing. The Command Line can help you avoid using the mouse or menus. To enable LTE workspace

Select Tools → Workspace → Customize...

Select LTE workspace from the drop-down list for Default style.

Press Close.

Note: LTE Workspace is not a feature of TurboCAD Deluxe

Command Line Options

(Available in the LTE Workspace only)

The Command Line console is divided into three sections:

The **Command Line** itself, which is located to the right of the word Command. This is where you enter command names (aliases).

The **Command History**, above the Command Line. This is where you see prompts and options for the current tool, as well as a history of previous tools.

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The **Actions List**, to the left of the Command History. This is where you can see a list of tools used, points or other geometry defined for the tool, and other actions such as zooming.

The Actions List shows three different types of information. Tools (**Line**, **Circle**, etc.) are shown in red. Points are shown in blue inside each tool. These are the geometric data used for each tool. Commands are shown in green; these include non-geometric actions like zoom or redraw.



You can double-click on any object in the Actions List to repeat the command or input. The icons below the Actions List control what is displayed there. The first icon can be used to show or hide the Actions List.



The other icons can be used to hide the tools, points, and commands.



The **Recent Commands** icon displays the tools last used, in reverse order. You can select one of these to activate it.

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The **Recent Input** icon displays a list of points entered. You can select one of these to use it for the current tool.



Using the Command Line

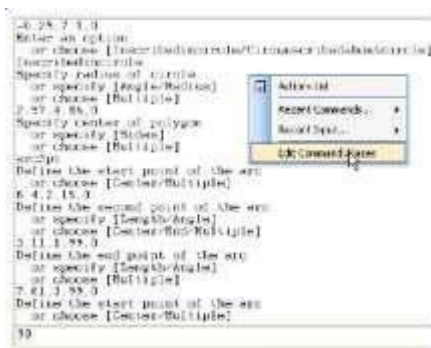
(Available in the LTE workspace only)

This example shows how to use the Command Line to create a polyline.

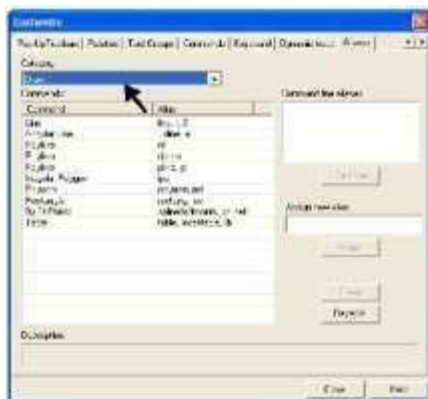
You could activate **Polyline** by clicking its icon or selecting it from the main menu, and still use the Command Line for entering data. But this example will use the Command Line for everything. In the Command Line, a tool is referenced by its "alias."

To see the list of aliases, right-click in the command history area (above the **Command** field) and select **Edit Command Aliases**.

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This opens the **Aliases** page of the **Customize** window. The commands are listed according to their appearance in the menu bar. **Polyline** is in the **Draw** menu, so choose the **Draw** category.



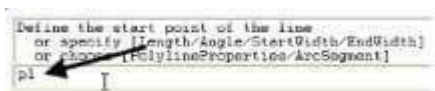
Note: You can also open this window by selecting **Tools / Workspace / Customize**.

Highlight **Polyline**, and its default alias, "pline", appears on the right side.

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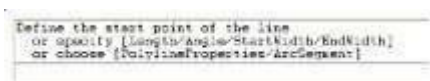
To make it easier to type, you can name the alias to something easy like "pl."
Then click **Assign**, then close the window.

To access the Command Line, you can place your cursor there, or press Tab to enter the Command field. Enter "pl" and press Enter.



Note: If dynamic fields are displayed, pressing Tab will place the cursor in the first dynamic field. You can type an alias in any of these fields, or press Tab until the cursor moves to the Command Line.

The default action is to "Define the start point of the line."



You can click a point on-screen or use the dynamic input fields (see "Using Dynamic Input" on page 87). To define the start point in the Command Line, type the coordinates using the format "0,0" then press Enter.

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```
Define the start point of the line  
or specify [Length/Angle/StartWidth/EndWidth]  
or choose [PolylineProperties/ArcSegment]  
0,0
```

For the end point of the first segment, type "20,0."

```
Define the next point of the polyline or polygon  
or specify [Length/Angle/StartWidth/EndWidth]  
or choose [PolylineProperties/ArcSegment]  
20,0
```

This creates a horizontal segment 20 units long.



To use the local options, you can open the local menu by right-clicking. To activate a local option in the Command Line, type the first letter of the option name. So to define the next segment by "Length" type "L" and press Enter.

```
Define the next point of the polyline or polygon  
or specify [Length/Angle/StartWidth/EndWidth]  
or choose [PolylineProperties/ArcSegment/Finish/Cancel]  
L
```

Then type "15" and press Enter to set the length.

```
L  
Specify Length value:  
press <Ctrl+L> to unlock value  
15
```

The prompt now mentions that you can lock this value by pressing Ctrl+L, so press Ctrl+L.

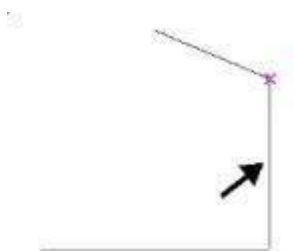
```
Specify Length value:  
press <Ctrl+L> to lock value  
>Lock* length value 15  
L
```

To complete the line segment, enter "A" then type "90." Press Enter twice.

```
A  
Specify Angle value:  
press <Ctrl+L> to lock value  
90
```

This completes the second line segment. If you move the mouse around, the preview line for the third segment is locked at 15 units.

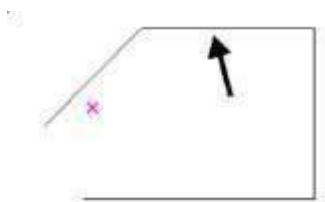
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For the next segment, enter "A" again, then type 180 and press Enter twice.



This finishes the third segment, which is also 15 units long.



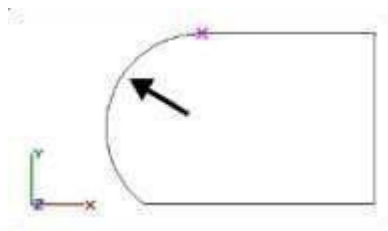
To make the next segment an arc segment, look in the Command History for option names. When an option has two words, such as "Arc Segment," you enter the first letter of each word. So type "AS" and press Enter.



The last step is to complete the polyline by closing it. But if you enter "C," that will invoke the **Circumference** option which is listed first. To invoke Close, type the entire word: "close."



The polyline is now closed with an arc segment.



2.7. Components of the UI

The main area of the TurboCAD screen is the drawing area. When creating and inserting 2D and 3D objects, you typically work in Model Space, and when putting your model on paper, Paper Space is used.

By default, the background color is white, but you can change that by selecting **Options / Background Color** and choosing another color.

Nearly all components of the screen can be customized. See [Customizing the UI](#).

Coordinate Fields

When creating or sizing objects, you can define points by entering them directly into the Coordinate Fields. By default, this toolbar is located at the lower right corner of the screen.



When moving the cursor, the values in the fields update dynamically. The type of coordinates depends on the coordinate system you are using. In the absolute and relative coordinate systems you use X, Y coordinates. In the polar coordinate system you enter an angle and distance. See [Coordinate Systems](#).

You can access the numeric fields via the cursor, or by pressing Shift+Tab or Ctrl+R. You can also press Tab to scroll between Coordinate fields.

Tip: If you want to jump from the Inspector Bar to the Coordinate Fields, first press Esc, then Ctrl+R.

If you enter a value and press Enter, Any Ortho or snap modes are ignored. You can also enter the results of mathematical expressions from the **Calculator**. See [Calculator Palette - Variables Palette](#)

Locking: You can lock a coordinate by clicking the lock box. This means that no matter where you place the cursor, the value will remain fixed.

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You can also lock coordinates by using the **Lock** options in the **Modes** menu.

Tip: The Coordinate Field locks are particularly useful when you lock just one field. If you use a locked value in one field, you can then set the second coordinate using the mouse. This makes it easy to define a series of points along a specific horizontal or vertical line.

The Coordinate Field also provides access to snap modes. Disable the SNAP button to temporarily turn off any running snaps, and disable the GEO button to temporarily turn off any running geometric aids. Right-clicking on either the SNAP or GEO fields opens the **Drawing Aids** window. See [Snaps](#)



Drawing Area - Model Space and Paper Space

For creating and laying out your drawing, TurboCAD provides you with two drawing environments: Model Space and Paper Space.

Model Space is the environment in which you create your drawing, usually called a model. In this area you do drafting and design work, creating two-dimensional drawings or three-dimensional models.

Paper Space is the environment in which you create the final layout of your drawing for printing or plotting it on paper. In Paper Space you usually arrange the drawing's elements on a sheet of paper.

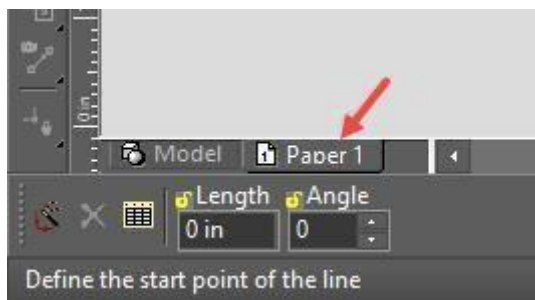
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Note: Floating Model Space enables you to use Model Space tools within a viewport in Paper Space. See [Floating Model Space](#)

You can use the small button at the top left corner, at the intersection of the rulers, to toggle between Model and Paper Spaces.

Note: If the rulers are turned off, this button will not appear. Go to **Program Setup/Desktop** to toggle the ruler display.

There are also workspace tabs at the lower left corner of the screen. By default, each file starts with one tab for Paper Space. Click the tabs to switch spaces.



Note: If scroll bars are turned off, these tabs will not appear. Go to **Program Setup/Desktop** to toggle the scroll bar display.

Inspector Bar

Enables you to create objects by defining some or all of its numerical parameters. By default, the Inspector Bar is located immediately below the drawing window.



Note: You can customize the Inspector Bar appearance, by choosing **Classic** or **Custom** in the **Desktop** page of the **Program Setup**. You can also display or hide the Inspector Bar on this page.

For most tools, the Inspector Bar is divided into three parts. On the left side are three icons, **Auto Add Constraints**, **Cancel** and **Properties**.

Auto Add Constraints: When enabled, it automatically applies constraints while drawing objects.



Cancel: Ends an operation without completing it. For some tools, once the operation is complete the tool remains active. You must click **Cancel**, or activate another tool, to exit the tool. The hotkey for **Cancel** is the Esc key or space bar.



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Properties: Opens the **Properties** window for the object you are creating. See [Object Properties](#).



The middle section of the Inspector Bar contains the numeric fields. For example, when creating a line segment in the Polyline tool, you can set the **Length**, **Angle**, and **Start** and **End Widths**. For a circle, the fields would be **Radius**, **Diameter** and **Circumference**. When moving the cursor to position or size an object, the values in the fields update dynamically.

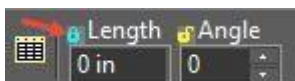


You can access the numeric fields via the cursor, or by pressing Tab or Ctrl+E. You can also press Tab to scroll forward to the next field. Return to the drawing by pressing Esc or clicking in the drawing.

Tip: If you want to jump from the Coordinate Fields to the Inspector Bar, first press Esc, then Ctrl+E.

If you enter a value and press Enter, Any Ortho or snap modes are ignored. You can also enter the results of mathematic expressions from the **Calculator**. See [Calculator Palette - Variables Palette](#)

Locking: For some fields you can lock their values by clicking the lock symbol. This means that no matter where you place the cursor, the value will remain fixed.



The right side of the Inspector Bar contains options relevant for the tool or mode.

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Note: All options on this section of the Inspector Bar can also be accessed from the local menu, opened by right-clicking.

There are two common local menu options that appear for many functions: **Finish** and **One Step Back**. **Finish**: Completes the operation and exits the tool. For instance, when creating a polyline, you can click **Finish** after creating the last segment. The hotkey for **Finish** is Alt+F.



Note: For many tools, double-clicking when creating the last component of an object is equivalent to clicking **Finish**.

One Step Back: For tools that require multiple steps, this enables you to scroll backward through the steps already completed until you reach the desired start point, then continue working.



When in **Select Edit** mode, the Inspector Bar has a different appearance. The numeric fields enable you to change the size, scale, and rotation of the selected objects, while the local menu options include **Make Copy**, **Selector** properties, etc. **Size** and **Position** fields for 3D objects are turned off by default. See [Select Edit](#)



Note: The options that appear on the Inspector Bar in **Select Edit** mode depend on whether you are in 2D or 3D Selector mode, and can be changed. See [2D-3D Selector](#)

Mathematic operations, i.e. add, subtract work in the Inspector Bar fields.

Customizing the Inspector Bar

You can remove any of the fields or local menu option icons by right-clicking anywhere on the Inspector Bar (except the numeric fields). This opens customize dialogue which lets you add or delete any of the controls. You can select icon from inspector bar and drag it away to exclude from inspector bar and you can add commands by searching them in commands tab and selecting it and dragging into the inspector bar. You can also change the width of any of the numeric fields by dragging either side with the mouse.

Local Menu

The local menu, sometimes called the context menu, popup menu or context-sensitive menu, is opened by right-clicking the mouse. The contents of this menu differ depending on where you right-click, and in which tool you are working.

Note: You can customize the contents of the local menu via **Tools / Customize**.

See [Customize Popup Toolbars](#)

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When working in a tool, options on the local menu also appear in the Inspector Bar. See [Inspector Bar](#). In addition to customizing local menu items, you can add popup toolbars icons that will appear each time you open the local menu. See [Customize Popup Toolbars](#).

The following items are found on most local menus:

Cancel: Cancels the current operation, and exits without completing.

Undo: Undoes the last operation and return to the previous condition. The last operation is displayed here.

Redo: Reverses Undo.

Properties: Opens the **Properties** window. See [Object Properties](#).

Local Snap: Sets the snap mode for the next point only. See [Snaps](#) and [Snap Modes](#).

Menu Bar

The menu bar is at the top of the screen. (Most tools are available in toolbar icons as well).



Palette Area

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By default the palette area is on the right side of the workspace.

For details on each palette, see [Palettes](#)

Rulers

You can work with rulers to have a visual representation of the current dimensions.

Rulers are only displayed for orthographic views (top, left, right, etc.) and display the current Model Space or Paper Space units (see [Space Units](#)).

Tip: Another handy visual aid, which can be used in tandem with rulers, is the grid. See [Grid](#).

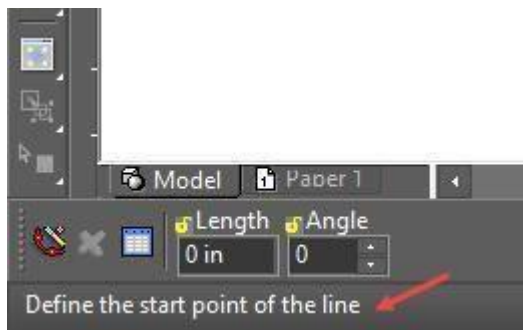
Scroll Bars

Your drawing can be larger (much larger!) than what you see on the screen. The scroll bars can be used to pan to a different area of your drawing. Click on the arrows at the ends of the scroll bars to move a small amount. Drag the scroll box itself to move faster.

Status Bar

Located at the lower left corner of the screen, the Status Bar lets you know the current status of the model.

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For example, if you have just activated the **Line** tool, the Status Bar will contain the prompt "Define the start point of the line."

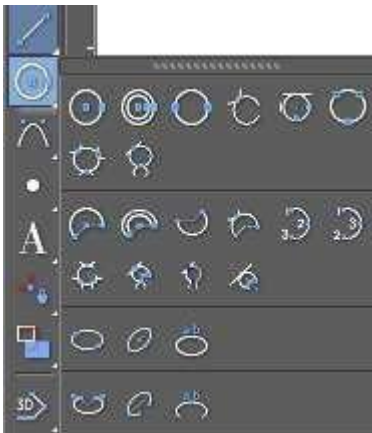
Toolbars

Toolbars are groups of related icons. They can be located on top and below the screen, and to either side of the screen.

By default, the **Standard** toolbar appears at the top of the screen, and the **Property** toolbar (see [Property Toolbar](#)) appears just below it. **Drawing** tools appear along the left side. Icons that have a small yellow or black triangle in the corner are fly-outs. If you click one and hold the mouse button...



... the entire toolbar will "fly out," and you can select the desired icon. The color of the triangle is dependant upon UI settings.



You can also create tabbed toolbars and popup toolbars that appear when you open the local menu. You can modify a toolbar's icons and create your own toolbars.

See [Customize Toolbars](#). In addition to the default toolbars you see when you first start TurboCAD, there are numerous other toolbars you can display. There are two ways to access additional toolbars: Open the **Toolbars and Menus** page of **Program Setup**. This opens the **Toolbars** tab of the **Customize** window, which contains a list of all available toolbars; check the ones you want to see. Right-click in any toolbar area. The local menu that appears contains the list of all available toolbars; select a toolbar to add or remove from the screen.

Moving, Docking, and Rolling Toolbars

2.8. Customizing the UI

Default UI Menu: Tools/WorkSpace/Customize

Ribbon UI Menu:

You can customize toolbars, menus, keyboard, and your IntelliMouse (if you use one). You can invoke the **Customize** window at any time by right-clicking in any blank toolbar area.

Note: If you prefer an older interface for customizing toolbars and menus, select **UI Themes**, and select the classic interface.

Whenever you make changes to the UI settings it is best to save the Workspace file (under the Options tab accessed via Tools|Customize) in order to accurately retain your changes.

Customize Ribbon

TurboCAD (All variants)

Enables you to create Ribbon for any desirable and familiar configuration.

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In Customize Ribbon, the following options are available **Add:**

Adds a selected tool into the tab that has been selected.

Remove: Removes a selected tool from a tab.

Rename: Rename a selected menu tab using this button.

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New Tab: This button adds a new Menu tab.

New Group: Creates a new group in a selected menu tab.

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Hide Tab: Unchecking the tab will hide tab from the Ribbon.

Move Tabs: Tabs can be repositioned using up/down arrow buttons.

Drop-Down List: To add a drop-down list to a menu, toolbar, or ribbon, you can use the “ **New Popup Menu**” or “ **New Popup Toolbar**” items on the “Commands” page of the Customize dialog box.

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Then drag the required tools Icons from the “ **Commands**” list. If you hold the mouse over a popup control, a list of its sub elements will open.

Import Ribbon: Manually created custom(.xml) configuration files can be imported and used in TC 2023 Ribbon UI

Currently Available On: Allows the user to see the position of a particular command available in the menu, UIs and toolbars.

Find Control in Custom Ribbon Page

The Find feature allows users to search or find Command/Group/Pop-ups/Tabs in the Ribbon tree.

Ribbon Theme Support

TurboCAD (All variants)

Ribbon UI of TurboCAD supports TurboCAD theme e.g. Classic, Default or Custom Themes

Customize Options

Enables you to manage the configuration (Config) files, control the look of the TurboCAD window, and specify external devices.

Available Space Icon Sizing

Users can change tool icon size/icon text and the icon image along with many other customizations. To open this menu, right-click on any tool icon. To access this menu, Customize Window should be open.

Changing the Styles of Ribbon Icons with lack of free space

When using the Ribbon Interface, as you change either the UI Scale or Icon size through the Options/Toolbars and Menus /Options menu, the Size and Style (with or without text) of the icon will dynamically change.

Default Scale and Tool Icon Size Settings with Ribbon Display:

After: UI Scale changed to 150% and Toolbar Icon size increased:

Styles and Themes

TurboCAD (All Variants)

These functions allow you to select from several different Workspace/UI looks, or "skins."

Default: uses the "new" (TurboCAD 2023) dark interface with new icons.

TurboCAD Classic: uses a light background with TurboCAD's classic icons.

Custom Theme: uses the new icon set, but allows you to choose a skin to customize the look and feel of the interface.

Other: Various settings for how icons and tool-tips are displayed.

Fonts: Select the font and its size for the selected control.

Colors: Set color for the selected control.

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Reset All: Resets all the fonts for the selected control.

UI Scale Slider

TurboCAD 2023 (All variants)

Now TurboCAD is fully compatible with 4K monitors. Menu, toolbars and Ribbon updates accordingly when "UI Scale" is changed.

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This will scale the "Toolbar icon size" and "Others Icon size" together. This option is implemented to make TC 2023 compatible with 4K display.

Workspace

Select the functionality and layout of tools:

Default Workspace - Uses the standard TurboCAD interface and layout

LTE Workspace - Uses the LTE Workspace layout which includes Dynamic input and the Command Line.

To Load a Config:

Select a file from the Load From drop-down list.

Click the Load From button.

To Create/Save a Config:

Type or Select a name in the Save In drop-down list.

Click the Save In button.

Locate/Browse Config Files:

Click the Browse button to open the file location of your configuration files.

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To Use the Default Configuration:

Click the Default button.

Other

Toolbar icon size (X, X): Sets the size of the main toolbar icons in pixels X-by_X the default is 24 by 24. The range goes from 16x16 to 64x64.

Other icon sizes (Inspector bar, Palettes etc.): Sets the size of the other application icons in pixels X-by_X the default is 24 by 24. The range goes from 16x16 to 64x64.

Show Command Icons in the Pull-down Menu: Designates whether icons will be shown in the pull-down menu.

Show Command Icons in the Local Menu: Designates whether icons will be shown on the local menu.

Menu Animation: Sets the type of animation the will be performed by menus.

System (Default): use the windows setting

Random

Unfold

Slide

Fade

None

Show ScreenTips on toolbars: When checked, Tooltips are shown when the cursor hovers over toolbar items.

Show shortcut keys in ScreenTips: When checked, shortcut keys are shown when the cursor hovers over toolbar items.

Customize Commands

The **Commands** page enables you to customize toolbars by adding and removing icons. Display the toolbar you want to change, and open the **Commands** page. Locate the commands you want to add to the toolbar and drag it from the **Commands** list to the toolbar itself.

To remove an icon from a toolbar, drag it away from the toolbar into the **Commands** page.

Search Option:

You can also search for a command. Type a command in the "find" option to search, the first suitable item is highlighted. To browse through the commands, buttons next to the input field highlight and can be used to show the previous or next suitable item.

Successful search options are added to the list and saved between sessions.

Customize Keyboard

The **Keyboard** page enables you to view current keyboard shortcuts (hotkeys), to modify them, or to create new ones.

Whenever you make changes to the UI settings it is best to save the Workspace file

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(under the Options tab accessed via Tools | Customize) in order to accurately retain your changes.

Tip: Select **Help/Keyboard** for a list of all shortcuts.

Locate the **Category** and then the command for which you want a shortcut. If a shortcut already exists, it will be listed in the **Key assignments** box.

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You can click **Remove** to delete any unwanted shortcuts.

To assign a shortcut, type it (for instance, press Ctrl+E) and it will appear in the **Press New Shortcut Key** box.

Click **Assign** to implement the shortcut.

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Search Option:

You can also search for a command. Type a command in "find" option to search, the first suitable item is highlighted. To browse through the commands, buttons next to the input field can be used to show the previous or next suitable item. Successful search options are added to the list and saved between sessions.

Multiple Keyboard Shortcuts:

(Available in all Variants)

Unlike in the previous versions where there were not enough Ctrl-Alt-Shift key combinations to cover every command, users can now create multiple key combinations for customizable keyboard shortcuts.

Another use of this feature is that the user doesn't have to create/use a key combination which is difficult to apply with one hand. The ideal case would be that the user uses left hand to apply the keyboard shortcut and the right-hand to control the mouse.

Example:

To assign shortcut such as "tt" to any command.

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Select the command in Customize Keyboard to which you want to assign the shortcut.

2. In the **Press New Shortcut Key** box, press the Control key and then press 't' twice. Then ,press the Assign button and close.

To invoke the command, press control and release control key, then press 't' twice.

Demo Video:

<https://youtu.be/E2d31x07Grs>

Devices

Relevant if you have an Intellimouse connected to your computer.

Wheel assignment: Shows the command that will be executed when you rotate the mouse wheel forward, back, or click with it. To change it, highlight the desired command, select **Forward**, **Backward**, or **Click**, and click **Assign** to implement the

change.

Built-In Folder

Occasionally the configuration INI files may become corrupted causing loss of functionality, and no amount of customizing will restore it.

To fix this problem the standard procedure is to purge the Built-in folder by deleting it.

WARNING: It is best to do this only after consulting with IMSI/Design support or technical services.

To Delete the Built-in folder:

Navigate in your Hard Disk to

C:\Users\AppData\Roaming\IMSIDesign\TurboCAD 2023<<Your Version of TurboCAD>>\Profiles

Select the Built-In folder. Then select Delete from the local menu to delete the folder. **Repeat WARNING: It is best to do this only after consulting with IMSI/Design support or technical services.**

Close the file window, then close the Customize dialog.

Restart the application.

Customize Palettes

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The **Palettes** page enables you to control which palettes appear and how they can be displayed and moved.

Auto activate palettes: Opens the palette relevant for the current option. For instance, when using hyperlinks, the Internet palette opens. **Auto-hide palettes:** Hides the palette when not in use.

Docked palettes: These options are relevant while moving or docking palettes.

Show Caption: Displays the palette title.

Use Splitter Tracker: If disabled, the palette can be resized dynamically (in real time).

Alpha Docking Context: If disabled, an outline of the palette appears while moving a palette. If selected, a transparent blue rectangle appears while moving.

Docking Stickers: Displays the blue controls that appear while moving or docking.

For details on the various palettes available, see [Palettes](#)

Customize Popup Toolbars

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Popup toolbars are icons that appear with the local menu, which is opened by right - clicking. This is useful if you use certain icons frequently and want convenient access to them. There are three popup toolbars defined by default:

If you want to add additional popup toolbars, simply check them on the list. If you want to change the existing popup toolbars, first display them as standalone toolbars. This is done in the **Toolbars** page of the **Customize** window, or by right-clicking in any toolbar area and selecting the popup toolbar you want to display.

To customize the toolbar, click the arrow shown. This invokes options which enable you to add or remove icons.



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You can also use **Customize Commands** to add or remove icons. See [Customize Commands](#)

Search Option:

Whenever you make changes to the UI settings it is best to save the Workspace file (under the Options tab accessed via Tools|Customize) in order to accurately retain your changes.

You can also search for a command. Type a command in the "find" option to search, the first suitable item is highlighted. To browse through the commands, buttons next to the input field highlight and can be used to show the previous or next suitable item.

Successful search options are added to the list and saved between sessions.

Customize Toolbars

The **Toolbars** page enables you to select which toolbars are displayed.

This page can also be opened by right-clicking in any blank toolbar area.

You cannot delete default toolbars, but you can create new toolbars by clicking the **New** button. Toolbars you create this way can later be deleted.

Search Option:

You can also search for a toolbar. Type a toolbar name in "find" option to search, the first suitable item is highlighted. To browse through the toolbars, buttons next to the

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input field highlight and can be used to show the previous or next suitable item. Successful search options are added to the list and saved between sessions.

Customize a Toolbar:

To customize a new toolbar first, make sure it is displayed. Then click the arrow icon shown; the shape of this arrow depends on whether or not the toolbar is docked.



This invokes options which enable you to add or remove icons.

You can also use **Customize Commands** to add or remove icons. See [Customize Commands](#).

Note: If you prefer an older interface for toolbar customization, select **Tools/Workspace/UI Themes**, and you can select TurboCAD classic or Custom theme.

Whenever you make changes to the UI settings it is best to save the Workspace file (under the Options tab accessed via Tools|Customize) in order to accurately retain your changes.

Customize Tool Groups

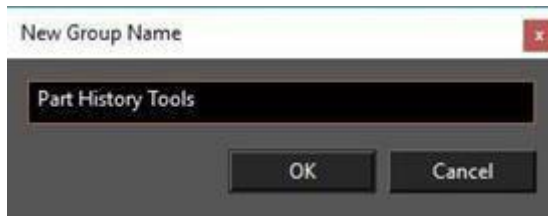
Tool groups are used to specify what palettes and popup toolbars appear for certain groups of tools.

1. To create a tool group, click the "+" icon.



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2. Assign a name to the tool group.



In the top part of the window, select the toolbars that contain the tools in the group. In the lower part, select the palette (or palettes) and popup toolbars that will appear while you are working in any of these tools.

In the above example, the Selection Info palette will be the default open palette when you are working with any **3D Modify** tool. Another example could be to open the Calculator palette when using any Dimension tool.

2.9. Program Setup

Default UI Menu: Options/Program Setup

Ribbon UI Menu:

Contains pages in which you can control TurboCAD program settings. You can access each page directly from the **Options** menu. The options that you set in this window are saved when you exit TurboCAD and will remain in effect the next time you start the program. The **Program Setup** pages can be accessed from the **Options** menu.

You can also display the **Program Setup** toolbar by right-clicking in any toolbar area and selecting **Program Setup**.

Tip: You can also use the **System Defaults** in the TC Explorer Palette to set program setup options. See [General Settings](#)

Auto-Naming

Default UI Menu: Options/Program Setup/Auto-Naming

Ribbon UI Menu:

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Controls how groups, blocks, and symbols are named. See [Auto-Naming of Groups, Blocks, Symbols](#)

Prompt for name: You will receive a prompt each time a new item is created.

Generate names: Names will be automatically assigned.

Prefix: If names are automatically generated, you can enter a string that appears before the item name. The "@" character is a placeholder for the automatic number.

Insert blocks when creating: Each block will be inserted into the drawing once it is created.

Prompt for library location: If not checked, all saved symbols will be stored in the default folder. See [Loading Symbol Folders into the Library](#)

Note: If you turn off both the **Prompt for name** and **Generate names** options, names will not be assigned to groups. However, names will be assigned to blocks and symbols.

Color Palette

Default UI Menu: Options/Program Setup/Color Palette

Ribbon UI Menu:

Enables you to add, modify, or delete colors from the TurboCAD palette.

To add a new color, type the name and click **New**. You will then be prompted to select a color from the color wheel.

You can modify a color by selecting it from the list and clicking **Modify**. Delete a color by selecting it and clicking **Delete**.

When you have the colors you want, they can be displayed for easy selection in the Colors and Brushes palette. See [Colors and Brushes](#)

The Color Wheel

You can set colors by RGB values, or by Hue, Saturation, and Value numbers.

Red, Green, Blue: Sets the amount of each color in the light. Values can be a maximum of 255.

Hue: The color value, where 0 is red, 60 is yellow, 120 is green, 180 is cyan, 200 is magenta, and 240 is blue. If you change the hue, the values for red, green, and blue will be changed to match.

Sat: The saturation level for the color (amount of color), up to a maximum of 240.

Lum: The luminosity (brightness) of the color.

Constraints

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Default UI Menu: Options/Program Setup/Constraints

Ribbon UI Menu:

Settings and controls for geometric and dimensional constraints.

Auto Add Constraints: If checked, the **Auto Add Constraints** tool is enabled. Any constrainable geometry you create while this is active will have constraint automatically assigned. See [Constraining Geometry](#) and [Constraining Dimensions](#)

Update Connected Objects while Dragging: Dynamically updates the position, shape, and size of constrained objects as you drag them within the **Edit** tool.

Connected Objects Editing:

DCM Default: Changes to any part of a set of constrained objects can affect all objects equally.

Use Priority Level: Constraint changes to any part of a set of objects will affect that part first, with minimal changes to the remaining objects.

Incremental Evaluation: Constraint changes are constantly checked and changes are being made. If disabled, the results will be checked only after changes are made. For large scale changes, this option should be enabled.

Constraint Markers and Dimensions

Place marker to the same layer with object: Constraint markers are placed on the same layer as the object that is constrained. Otherwise, they are placed on their own layer.

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Show variable name in dimension text: Displays the variable name in parentheses after the dimension value.

Printing: Choose whether constraint markers and constrained dimensions will be included when printing.

Desktop

Default UI Menu: Options/Program Setup/Desktop

Ribbon UI Menu:

Controls what appears on the screen.

Workspace: Controls the display of the working area

Status Bar: Displays or hides the Status Bar.

Scroll bars: Displays or hides the scroll bars.

Rulers: Displays or hides the rulers.

Inspector Bar

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Inspector Bar: Displays or hides the Inspector Bar. See [Inspector Bar](#)

Classic: Set Inspector bar fields and labels to be in the classic side by side orientation,

Compact: Set Inspector bar fields and labels to be in the compact over/under orientation,

Field width: Specify the field width.

Desktop default scheme

Desktop Scheme: Customizes the color of paper and background, and floating palettes.

Click **Modify** to change an element's color.

Tip: You can also click the different elements in the preview window.

Show margins: Displays page (printer) margins.

Show background: If checked, the background behind the paper will be displayed.

If unchecked, the background will be white.

Apply to opened drawings: Apply changes in the scheme to all opened drawings.

Apply to active drawing: Apply changes to the currently active drawing.

Tip: You can switch off the paper until you're ready to print, then use the **Page Setup** (see [Page Setup](#)) to place your drawing on the page.

OpenGL Hardware Acceleration: Available if your video card is equipped with an accelerator, take this characteristic into account while performing a render.

Render new window: When you render your drawing, the render view will open in a new drawing window.

Background Render: With this option, TurboCAD tools work faster in Render mode;

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rendering will "interfere" less with other tools. This setting can only be applied when working with one window.

Multithreading Quality Render

Use automatic number of threads: If checked the system automatically use the full number of available threads.

Threads: Specify the number of threads to be used for rendering if **Use automatic number of threads** is off.

Multithreading LightWorks: If checked multithreading will be used for LightWorks if the plug-in is installed.

Visualize Preferences

(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Options/Program Setup/Visualize Preferences

Ribbon UI Menu:

Enables you to set visualize preferences.

Device Type:

Device mode can be selected out of: OpenGL ES2, GDI, and OpenGL.

OpenGL ES2 is most advanced and most productive device mode. OpenGL, GDI device modes can be used for virtual machines and low (embedded) video cards.

Device Options:

The Device Options dialog allows you to setup Visualize device options.

Caching:

Use scene graph

Enables usage of the optimized Scene Graphics Manager, which is an additional render layer that builds its own internal hierarchy of scene graphic objects and optimizes them in the background of multiple threads to provide better rendering performance. This option can increase memory usage up to two times but rendering performance increases up to 10 -30 times.

By default: **On**

Enabled: If **Device Type** is **OpenGL ES2**

Use Blocks Cache

Enables caching of a hierarchical block structure.

By default: **Off**

Enabled: **Always**

Use TTF Cache

Enables usage of the TrueType font cache. It is useful for rendering drawings that contain a lot of text. If TrueType font cache is enabled, each font character used inside the drawing is cached as a separate metafile.

By default: **Off**

Enabled: **Always**

Visualization:

Use Geometry Shaders to optimize rendering of

Sets bit flags that instruct the vectorization device to use Geometry Shaders (if this feature is available for the GPU) to optimize rendering performance for specific geometry features. Values:

- **None:** Geometry Shader is not used.
- **Lineweights:** The renderer invokes Geometry Shader to optimize rendering of lineweights.
- **Linetypes:** The renderer invokes Geometry Shader to optimize rendering of linetypes (includes plot style related linetypes and highlighting by dashed patterns).
- **Lineweights and Linetypes:** The renderer invokes Geometry Shader to optimize rendering of lineweights and linetypes.

Range: **[None, Lineweights, Linetypes, Lineweights and Linetypes]**

By default: **Lineweights and Linetypes**

Enabled: If **Device Type** is **OpenGL ES2**

Discard Back Faces

Description: Enables back-face culling. By

default: **Off**

Enabled: If **Device Type** is **OpenGL ES2**, and **OpenGL**

Display:

Use Overlay Buffers

Enables the usage of overlay buffers during rendering processes.

By default: **On**

Enabled: If **Device Type** is **OpenGL ES2**

Double Buffering

Enables an additional intermediate off-screen buffer to avoid a blinking effect during on-screen rendering.

By default: **On**

Enabled: **Always**

Regeneration:

Enable Multithread

Enables multithreading.

By default: **Off**

Enabled: **Always**

Anti-Aliasing:

Visualize supports two types of Anti-Aliasing: Line Smoothing and Fast Approximation Anti-Aliasing (FXAA) Anti-Aliasing is available for device type OpenGL ES2.

Materials:

ODA Visualize SDK render does not yet have its own materials in TurboCAD as Lightwork or RedSDK.

The structure of ODA materials is very similar to the materials of other render engines, therefore added the ability to import materials from Lightwork or RedSDK. Currently ODA render supports color, texture, transparency map, bump map, these parameters can be imported from Lightwork or RedSDK materials.

If the options 'Lightwork' and 'RedSDK' (from Program Setup\ODA Visualize page) are not set, then the default material (with graphic's color) will be used. If the options are set, then the material that has the texture will be used.

Note: RedSDK materials can be used even if the plugin is not installed, but editing of these materials is available only if the Native Draw mode is RedSDK.

Regeneration:

When changing the view (zoom, rotation), primitives can change automatically or by pressing a key F5 (Menu View\Redraw or View\Regen) Drawing speed may slow down when auto-regenerate is enabled.

The user can specify which types of primitives will be updated automatically when the view changes.

Regeneration Options:

The dialog allows you to setup regeneration options.

Non-Zoomable objects

Non-zoomable objects (camera, light, blockmark, unscaled text) keep their size when zooming.

By default: **On**

Enabled: **Always**

Non-Rotated text

Non-Rotated text objects keep their orientation automatically.

By default: **Off**

Enabled: **Always**

2D/3D objects

Turn on/off automatic regeneration of 2D/3D object (ADT objects, Landscapes) switching when changing camera orientation.

By default: **Off**

Enabled: **Always**

Line styles

Objects with some line styles will be updated automatically (as in GDI mode when zooming and camera rotating).

By default: **On**

Enabled: **Always**

Curves, Arcs

Curves and arcs keeps their smoothness when zooming.

By default: **Off**

Enabled: If **Device Type** is **OpenGLES2** or **OpenGL**

Miscellaneous:

Display FPS: If this option is on, a display appears in the upper right corner of the drawing space. This display track the Frame Per Second to indicate the speed of the Visualize drawing engine and Device Type.

Show Normals: The visualization of the normals make it easier to estimate the smoothness and orientation of the 3D objects, and to find artifacts/defects on those objects. Supported in Hidden Line and Draft renderings. This option is Off by default.

Default Render: Enables usage of Visualize rendering modes by default in Draft and Hidden Line renders. This option allows the user to switch on Visualize Draft and Hidden Line render in one click.

Keep data until drawing close: Keep the Visualize contents in cache to optimize switching between paper/model spaces and Visualize/non – Visualize rendering modes.

By default: On.

Use Arcs in ACIS isolines: Enables usage arcs in ACIS isolines instead of polylines.

By default: On.

This option allows user to draw contours in the same way as GDI, and use arches to get an accurate representation of the contours.

Use view for ADT 2D/3D switching:

This option allows user to draw object 2D/3D in the same way as GDI (options is off), and draw object 2D/3D the same as in ACAD (options is on).

If the options is on switching to object 3D state occurs if the view differs from the plan view, and in this case, the orientation of the object is not taken into account

By default: On.

Changing this option requires reloading the Visualize render, so the corresponding warning dialog is shown.

Visualize Switching

(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Options/Program Setup/Visualize Switching

Ribbon UI Menu:

Enabled in model space if camera wireframe mode is set.

The command turns on / off Visualize camera mode in wireframe

File Locations

Default UI Menu: Options/Program Setup/File Locations

Ribbon UI Menu:

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Enables you to specify the folders where TurboCAD places several types of program files.

Templates (.tct)* - Files in which you save settings as well as block libraries, symbol libraries, and scripts, that you can use to start a new drawing.

Backups (.bak)* - Backups that are created if you check **Make Backup Copies** option in the **General** page.

AutoSave Files (.asv)* - Files that are saved automatically, if this option is turned on in the **General** page. These files allow you to restore work after a system crash.

Symbol Library Files (.slw)* - Sets of related symbols, saved to one common *.slw file or a set of drawing files saved to a specified folder.

Drawings (.tcw)* - Drawings and models you create in TurboCAD.

Publish to HTML Files - Files created using **File / Publish to HTML**.

Print Styles: See [Print Style Options](#)

Shapes (.shx):* Shape files (*.shx) used in the creating of line styles. (This is not the same as *.shx font files.) See [Line Styles](#)

Tip: If you have two disk drives (or access to a network drive), it is wise to have your backup files automatically saved to a second drive.

General Setup

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Default UI Menu: Options/Program Setup/General

Ribbon UI Menu:

Sets file opening and saving parameters, as well as user information

New document from: Select how files will be opened when using **File / New**.

Default template for new files: If no template is specified when creating a file, a default template will be used. This option sets the default template.

Prompt for template name: You will be prompted for a template each time you start a new file. If you want to use the default template without being prompted, clear this box.

Open all files as read-only: If checked, files cannot be edited.

Make backup copies: If checked, select **On First Save** or **On Every Save** to determine how often backups will be made. Backup files have the extension .bak. By default, backup files are saved in the same folder as your primary drawing files, but you can change this on the File Locations page.

Save drawings every: If checked, autosave copies (.asv files) will be made at the specified intervals. After a system crash, TurboCAD will open the autosave version of

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your drawing as soon as you launch TurboCAD. Autosave files are stored in the AutoSave folder by default, but you can change this on the File Locations page.

Prompt for summary information: The **Summary Information** window enables you to record user information connected to your drawing. If checked, this window will be displayed the first time that you save any drawing to disk, and whenever you use **Save As** to save a new copy of the file.

Automatic Backup and Restore: Backups the file and restores is automatically after a specific time interval.

Assign material to color faces during importing interOp data: (Available in Platinum only) If checked, when reading ACIS objects (SAT/IGES/STEP/SAB import), if ACIS object has colored faces, then the appropriate material is assigned to these faces.

Healing of SAT insertions: For imported files, heals gaps between faces and holes within faces.

Save desktop on exit: If checked, the desktop settings (toolbars, etc.) will be saved when you exit the program.

Load at startup: You can choose to load the last working file and the last window layout (see [Multiple Windows of the Same File](#)) each time you start TurboCAD. **Help Settings:** Specifies how help will be accessed.

User Information: Basic user information

Preferences

Default UI Menu: Options/Program Setup/Preferences

Ribbon UI Menu:

General program options, such as cursor shape, axis color, zoom factor, and display of coordinate system icons.

Coordinate system icon:

You can choose to display the User CS and/or World CS symbol/s.

For both, you can set the color of the axes and size of the icon.

For the World CS, select the corner of the screen where it will be displayed: TopLeft, TopRight, BottomLeft, BottomRight

Drawing cursor

Cursor Shape: Select one of the three shapes provided.

Show crosshairs: Overrides the normal cursor, displaying a screenwide horizontal and vertical line intersecting at the cursor position.

Show I sometric: Overrides the normal cursor, displaying all three axes.

Show Snap Aperture: Displays a circular area around the cursor whose radius is the Aperture Size. Aperture is used while snapping, to detect objects.

Aperture size: Specify the radius of the aperture in pixels.

You can also use the View | Display | Cursor menu to change the cursor display.

Undo/Redo

Undo/Redo depth: Sets the number of operations stored in the Undo buffer.

Include Model / Paper space switch in Undo buffer: If checked, switching between Model and Paper spaces will be included in the Undo buffer.

Selection

Use Choice if several entities in aperture while selection: If more than one object lies within the aperture area during a selection, a small window appears from which you can select the desired object.

Visual Effect Settings: Settings for the various "highlighting" methods. See Selection Visual Effects Settings.

Symbol Libraries

Default UI Menu: Options/Program Setup/Symbol Libraries

Ribbon UI Menu:

Displays the folders containing files that can be viewed in the Library palette.

Click **New** to add more folders. You can also modify the name of a library folder or file, or delete or modify symbol library files and/or folders. You can also access the symbol libraries from the Library palette. See [Library](#)

Toolbars and Menu

Default UI Menu: Options/Toolbars and Menus

Ribbon UI Menu:

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Open the **Customize** window to the **Toolbars** page to modify toolbars. See [Customize Toolbars](#)

Warning Dialogs

Default UI Menu: Options/Program Setup/Warning Dialogs

Ribbon UI Menu:

Controls the display of TurboCAD warning messages. If you received a warning message with the "Don't show this message again" checkbox, the warning message will appear in this table. Use the **Show** column to show or hide messages. UI enhancements of the Warning dialogs page

Show/hide list columns.

Right click on the header of the list to open the local menu. This makes it possible to show/hide list columns. **NOTE: The first three columns are always visible.**

Setting the value in the 'Default' and 'Log' columns

Left clicking in a cell in the 'Default' or 'Log' column will cause a drop-down, combo box to appear, thereby allowing you to change the setting of that cell.

Sorting Columns in the appropriate order

Columns can be sorted in two ways:

- Via left click at the header of the appropriate column

- Via local menu command. This local menu can be called via a right click within the list.

For the Time Column, the default sets the time in ascending order.

Filtering the contents of the list

The ability to filter the contents of the list is available via the local menu. This local menu can be called via a right click within the list. The filtering commands are:

Show diagnostic - in addition to displaying normal content, the list also shows diagnostic messages. FALSE by default.

Show fixed visibility - in addition to the normal content of the list also show messages for which the 'Show' state cannot be changed. FALSE by default.

Show only appeared - Show only those warnings which have already appeared. If this option TRUE the content of the list is equal to the contents of this list in the all previous TurboCAD versions. TRUE by default.

Changing Multiple 'Log' or 'show' Settings

Since the cells in the 'Log' column can have only two values ('Yes' and 'No'), it is convenient to change the value for the multiple selected cells (it is also possible to do

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this with the 'Show' column). This is possible through the local menu.

For Log:

Simply use Click and SHIFT to select the cells, then open the local menu and select the appropriate yes or no option.

For Show

Simply use Click and SHIFT to select the cells, check or uncheck the option box.

Confirm replace Warning Message

TurboCAD (Platinum, Professional, and Deluxe)

Now, if you tick the checkbox program will remember the last selected option and will not show the dialog again.

This dialog could be enabled again through Warning dialogs in Options.

Native Draw

Default UI Menu: Options/Program Setup/Native Draw

Ribbon UI Menu:

The Native Draw dialog allows you to specify which display/drawing engine will be used . you can select between standard GDI and Redsdk. GDI uses standard Windows CPU based drawing technology and CPU based OpenGL hardware acceleration. Redsdk is an OpenGL accelerated drawing engine that can substantially accelerate the speed at which entities are drawn.

GDI Options

Flicker Free Draw: When this option is turned on the redrawing of entities is made smoother, however, this can marginally slow down zooming and panning of drawings.

Redsdk Options

Hardware: Creates an anti-aliasing effect on 2D graphics elements. The values available are: Off, 1x, 2x, 4x, 8x, 16x. Turning this on will make the display smoother,

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but lowers the performance.

Vertical Synchronization: Makes the display looking better, while moving, however it may higher memory usage.

Text Smoothing: Turning this on will make the display of text smoother, but lowers the performance.

Engine:

Wireframe - Sets the engine anti-aliasing in Wireframe mode. Range - [1-5]. By default - 1. Enabled if Redsdk is On.

Hidden Lines - Sets the engine anti-aliasing in Hidden Line mode. Range - [1-5]. By default - 2. Enabled - if Redsdk is On.

Draft - Sets the engine anti-aliasing in Draft Render mode. Range - [1-5]. By default - 2.

Show Video System Info: Pressing this button will display the current video card setup on your computer. If your current driver for the video card is not the optimal driver for Redsdk you will be prompted to download the correct driver via a link within the window. If you do not download the optimal Redsdk will use a default driver.

Regeneration

To optimize performance Redsdk will delay the regeneration of some drawn elements during movements such as panning or zooming.

Regeneration Mode: You can specify that regeneration is Manual, or automatic. If set to manual all delayed elements will not be regenerated until you press F5 or select Regen or Redraw from the menu. The Advanced settings for Regeneration allow you to specify which items are optimized and which are not.

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Image cache after: Specifies the size (in MB) after which rendered images will be cached.

Advanced: Settings for **Regeneration** allow you to specify which items are optimized and which are not.

Miscellaneous

Display FPS: If this option is on a display appears in the upper right corner of the drawing space. This display tracks the Frame Per Second to indicate the speed of the Redsdk drawing engine.

Delay Surface Load: Delays the loading of surfaces and objects until rendering to optimize memory usage.

Caching of Model Space: Store the contents of the model space in the cache to optimize presentation.

Disable GDI/RedSDK quick switching (recommended): When checked, the program will be forced to restart when switching between **RedSDK** and **GDI** (either direction). This is the recommended setting. If **unchecked**, you can switch without restarting, but minor display problems, etc. have occurred when doing so.

Show Normals: When on this option causes Normals to be displayed on all 3D and Surface objects.

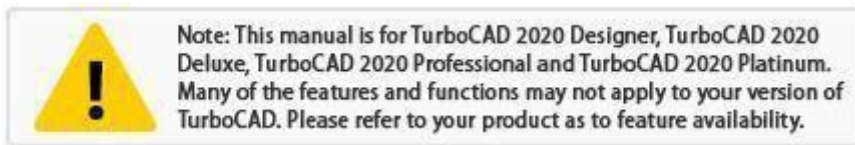
Note: Redsdk is dependant upon video cards which support OpenGL acceleration, and the features of that card. If you do not have an OpenGL video card you will not be able to use Redsdk.

Recommended: Switching between GDI mode and Redsdk will require a restart of TurboCAD.

Note: Redsdk conflicts with GDI's native support for OpenGL hardware acceleration If

you switch to Redsdk you will be prompted to allow TurboCAD to turn native OpenGL hardware acceleration off. With Redsdk turned on the Camera Properties "Renderable objects" is always disabled, "Suppress Hidden Line" is always enabled for all pages except the Wireframe page in Camera Properties Dialog.

Show Normals



The Show Normals setting is found under Program Setup/Native Draw. It is Off by default.

The visualization of the normals makes it easier to estimate the smoothness and orientation of the 3D objects and to find artifacts/defects on those objects.

When this option is On the normals are drawn in red color at the vertices of the 3D objects.

Normals are drawn in blue at the center of facets and show the direction each facet is facing. The direction of each normal is determined by the path that the edges of each facet take as they wind around the face. The calculation uses the right-hand rule to determine the normal.

The size of the normals is approximately 1/10 the size of the surface bounding sphere's radius (the smallest sphere that will enclose the object). In other words, the size of the normal is directly proportional to the size of the object.

The direction of the normals take into account the option **Reversed Normals** which is found in the 3D properties dialog of each object.

The base polygon of the normals are displayed takes into account the option **Double Sided** which is found in the 3D properties dialog of each object. Double-Sided objects show a double polygon.

Normals for Double and Single sided primitives

Render Manager Options

Default UI Menu: Options/Program Setup/Render Manager

Ribbon UI Menu:

General

You can specify how the Render Manager is displayed by selecting either Classic or Palette-Like

Classic

Variable sized preview.

Tree can be hidden.

Each category of components is displayed on separate tabs.

Palette-like

Fixed square preview area (240x240)

The tree cannot be hidden.

While resizing the dialog, only the tree's height, and the properties area size change.

All properties of the main shader are shown on one page. Components of sub shaders are shown on separate pages.

The Palette-like option is always used when the Render manager is accessed from object properties.

Palettte-like	Classic

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Low Level: Loads the minimal set of components - Active components - Active components as well as those for the components in only the current type's (Material, Luminance, Environment or Render Style) category.

Medium Level: Loads a subset of components - Active components as well as all categories for only the current Material, Luminance, Environment or Render Style type.

High Level: Loads a full set of components - All Active components as well as all types and categories.

How these function:

The Filter options in the respective palettes (Materials, Luminances etc.) will further control what will be loaded into the Render manager. As follows:

Choosing the Render Engine for loading in Render Manager

pressed)	pressed)	
Yes (if "RW" filtration button is pressed)	Yes (if "LW" filtration button is pressed)	Yes (if "All" filtration button is pressed)

Advanced Preferences

Default UI Menu: Options/Program Setup/Advanced Preferences

Ribbon UI Menu:

Advanced options for program control.

3D Boolean operations

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Transfer material from operand on its facet: When checked, material from one object is transferred onto the other during a 3D Boolean operation (3D add, 3D subtract, etc.). When unchecked, the materials are not affected on either object.

Shader Sets

Copy shaders set to the application database: Sometimes it will be desirable to copy shaders from a drawing into the Application Shaders Manager. To resolve this issue there is an option on the 'Preferences' page. By default, this option is off. When this option is active when a file is opened all shaders sets will not only be added to the Model Shaders manager but also will be copied to the Application Shaders Manager. It is not advisable to keep this option always active because if it is the Application Shaders Manager will gradually turn into a "trash bin". So the best way to use this option is to turn it on before opening the desired file and then turn it off again.

Shared shaders set: This option defines the scheme of work with the render attributes (material, luminance, environment, render style). By default, it is off.

Note: After changing this option it is necessary to restart TurboCAD.

If this option is ON, then all operations with the render attributes occur according to the "old scheme", which was used before TurboCAD 2016. Namely, the changing of a shader attribute in one drawing leads to the changing in all drawings and in the application database.

Also, the local menus in the palettes and in the Render Manager for that attribute are changed. For example in the local menu of the materials palette the two items are removed (marked by green):

In the Render Manager, the local menu for the material from the drawing will be absent.

Miscellaneous settings

Nudge Step: Sets the distance that a nudge will move an object. Values are specified in Pixels.

Zoom factor: Controls how much you zoom in or out of the drawing when you use View / Zoom In or Zoom Out. The default zoom step setting is 2, meaning that zooming in doubles the size. See Zoom In and Zoom Out.

Zoom In/Out Centralize: When this feature is On (default), rolling the mouse wheel will zoom in/out and centralize the view & cursor at the center of the drawing area. When this is Off, rolling the wheel will zoom in/out at the location of the cursor.

Fixed relative origin: If checked, the relative origin will be fixed while in relative or polar coordinates. If you need to relocate the origin, it must be done manually. See Relative Coordinates and Polar Coordinates.

Geometric select edit: If checked, objects will be selected by their geometric extents, rather than their cosmetic extents. This permits geometrically accurate scaling of objects like double lines and lines that have nonzero width. See Geometric and Cosmetic Select Modes?.

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Apply general property changes to all tools presets: Enables the Property toolbar to change all styles for all tools.

Intelligent Scale

Hatch: When a hatch is created it is automatically scaled to an appropriate size relative to the current view. This means in particular to the level of zoom.

Text : When a text is created it is automatically scaled to an appropriate size relative to the current view. This means in particular to the level of zoom.

Dimension: When a dimension is created it is automatically scaled to an appropriate size relative to the current view. This means in particular to the level of zoom.

Custom Coefficient: This is a scaling multiple that is applied to the intelligent scaling. For example, 2 would double the size whereas 0.5 would halve the size.

Draw by Layer

Options| Advanced Preferences

The Draw by Layer functionality is activated by selecting the **Apply general property changes to all tool presets** check box. This affects how tools relate to general properties such as Layer, Line Color, Line Width, Line Pattern etc.

When active, changing any of those properties, while using a tool, will change that property for all tools in the same way.

When inactive the application will behave in its traditional manner. This means that the

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layer setting for other tools will be unaffected, and changes to other tool properties, like Line Color, will only affect related tools. For example, changing the color of the Line tool affects the color of all line tools.

2.10. Drawing Setup

Default UI Menu: Options/Drawing Setup

Ribbon UI Menu:

Sets properties related to the current drawing.

The **Drawing Setup** pages can be accessed from the **Options** menu, or from the flyout icon on the **Standard** toolbar.



You can also display the **Drawing Setup** toolbar by right-clicking in any toolbar area and selecting **Drawing Setup**.

Tip: You can also use the **Defaults** for a drawing in the TC Explorer Palette to set drawing setup options. See [Drawing Settings](#)

ACIS®



Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Drawing Setup/ACIS

Ribbon UI Menu:

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Controls faceting of ACIS 3D solid (not surface) objects, thereby the accuracy of the model representation. Faceting generates polygonal representations of object faces while maintaining the edge consistency between adjacent faces.

Note: The faceted representation of a face is also called a mesh.

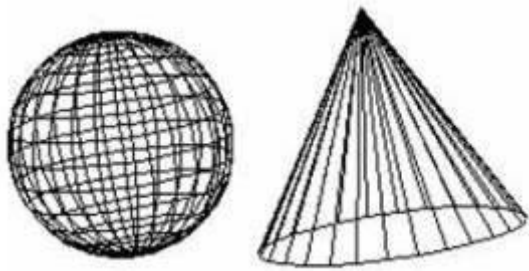
Faceting is used during rendering, and faceting refinements may have a significant impact on the rendering process. Tighter facets (higher in number) produce a smoother rendered surface but slow down the rendering process. **Degenerative**

Faceting: Reduces the number of edges without generating facets. Should be used if your model contains a significant number of 3D objects because the drawing loading time will be reduced. (For files of tens of megabytes, the loading time reduction may be more than ten times.) However, rendering such a model may take much longer.



With degenerative faceting

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Without degenerative faceting

Note: To see the effects of degenerative faceting, check to **Draw form-building edges** on the **Display** page of the **Drawing Setup (Options / Display)**.

Allow Axis Scale: ACIS solid objects can be scaled by entering different scale values in the Inspector Bar for each axis. If not checked, scaling will be uniform, and this uniform values must be entered in the scale fields for at least two axes.

Note: This parameter is only relevant for ACIS solids. Use the Selection Info palette if you are unsure about an object's type. For example, if you create a solid box, you will need to explode it once to make it an ACIS solid.

Multithreaded faceting for render: When checked, rendering is done faster for drawings containing ACIS solid bodies.

Creating Editing History: Enables you to edit 3D objects, after they are created, in the Selection Info palette. See [Editing 3D Objects using Selection Info](#)

Show Selected Axes for Solid Objects: Applies to objects that will be assembled by axis. See [Assemble by Axis](#)

Faceter

The level of facet refinement which is used to display ACIS objects can be set by the Faceter Quality slider.

For greater customized control, select the Expert Faceter Parameters option, and then specify the desired values.

Expert Faceter Parameters

Normal Tolerance: Specifies the maximum angle (in degrees) between surface normals at points on a facet.

Surface Tolerance: The Surface Tolerance represents the maximum distance between a facet and the true surface definition. By default this value is a predefined fraction of the body diagonal. However, using the Expert Faceter Parameters, you can override the default behavior and precisely control the deviation between the facet and true surface. For example, by specifying a value of 0.05, facets will deviate no further than 0.05 inches/mm from the true surface. This is particularly useful to remove facet display anomalies due to tolerances.

Max Edge Length: Constrains the maximum length of a side of a grid cell in object space. More precisely, the maximum edge length value constrains the maximum length of a diagonal of a grid cell, because the length of a grid cell diagonal provides an upper limit for the edge length of a facet. Therefore, this parameter also constrains the maximum edge length of a facet.

Min Grid Lines U: Specifies the minimum number of u grid lines (i.e., grid lines in the u-direction).

Min Grid Lines V: Specifies the minimum number of v grid lines (i.e., grid lines in the v-direction).

Grid aspect ratio-Specifies the maximum ratio of the long side to the short side of a grid cell in 3D space. The aspect ratio value does not guarantee that triangles will have a

particular aspect ratio; it applies only to the aspect ratio of grid cells. If the value of the grid aspect ratio parameter is less than or equal to 0.0, the grid aspect ratio is ignored. If the value of the grid aspect ratio parameter is greater than 0.0 and less than or equal to 1.0, a value of 1.0 is used. The default grid_aspect_ratio value is 0, which means the grid aspect ratio is ignored.

Use Grid to Edges: Specifies whether a grid is used and whether the points where the grid cuts the edge is inserted to the edge.

Object Specific Faceting

Faceting can be controlled on an individual object basis via the Properties of the object. Object specified faceting overrides the global setting in the ACIS settings.

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Two entities with new faceter options:

Advanced Grid Options

Default UI Menu: Options/Drawing Setup/Advanced Grid Options

Ribbon UI Menu:

Advanced controls for the frequency and locations of grid lines. See [Grid](#)

Note: **Set Grid Origin** is also accessible from the **Grid** toolbar.

Perspective grid: Relevant when the grid is displayed and you are working in **Perspective** mode.

By Drawing: An infinite grid will be displayed.

By Size: The size of the displayed grid is determined by Perspective grid extents. This value sets the grid extents in both directions from the WCS origin, i.e. a value of 3

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inches will produce a 6 x 6 grid.

Base: It is an advanced base angle - grid rotation angle. This option works for all types of grids.

1. Enable Grid.

Select Advanced Grid Options from Options/Drawing Setup. Enter a value for example 30 in Base and press OK.

Base angle can be both; positive or negative. Positive base angle rotates the grid in an anti-clockwise direction and negative base angle rotates it in a clockwise direction.

Angle Options

Default UI Menu: Options/Drawing Setup/Angle

Ribbon UI Menu:

Controls the measurement and display of angles, as well as control over the Ortho angle.

Angular System: Select degrees, degrees-minutes-seconds, grads, radians, or the surveyor system.

Base angle: The default base angle is 0 degrees (right quadrant point). You can change this value to start angle measurement from another base angle. **Precision:** The number of decimal digits.

Direction: Choose whether to measure angles clockwise or counterclockwise.

Ortho Angle: By default, Ortho lines are 0 and 90 degrees. You can change this by entering new values here.

Base angle: Sets the angle from which the Ortho angle is measured.

Step angle: The angle to which Ortho constrains lines. If you set the step angle to 15, for example, the line will be constrained to angles 15 degrees apart. The default step angle is 90 degrees.

Background Color

Default UI Menu: Options/Drawing Setup/Background Color

Ribbon UI Menu:

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Changes the background color, which is white by default.

Foreground color conversion: This option is designed to prevent objects from disappearing into a background of the same color (e.g. White on White) This is accomplished by inverting the display color of the object, but not the actual color setting. Color conversion will occur if this is checked. This can be controlled at the object level as well: [Properties Window](#)

Apply to all existing objects: Overrides the setting in individual objects and sets them to the 'By Drawing' default.

Display Options

Default UI Menu: Options/Drawing Setup/Display

Ribbon UI Menu:

Options for adjusting the quality and redraw speed of the display, and options for displaying block attribute values.

Performance

When working with large files or a slow computer, you may be able to increase your display speed by selecting one or more of these options. The display speed can sometimes be slowed by heavy text, styled lines, and fill patterns. **Quick text:** Displays all text as small boxes.

Quick line styles: Simplifies the display of line styles.

Quick vector brushes: Simplifies the display of hatch and fill patterns.

Display

Solid Fill: A line of a specified width will be drawn as solid.

Draw form-building edges: Available in **GDI** mode. Draws form-building edges of 3D surfaces.

Associative hatch: When you modify a hatched object, the hatch pattern will update to fit the new shape.

Block Attribute

Options relevant for blocks that contain block attribute definitions.

Force invisible: Hides all attribute values.

Normal: Shows the attribute values as they were defined while creating.

Force visible: Shows all attribute values, even those defined as invisible.

Defaults

Suppress display of objects smaller than: Sets the size at which TurboCAD will draw a simplified representation of objects. This size is measured in device units (the "device" is your computer's display) so that one inch will be equal to approximately one inch of space on your screen. At smaller sizes, TurboCAD will display objects using a simplified representation, increasing display speed. As you zoom in, you get the detail required for precise editing; as you zoom out, you get faster redraw.

Device pen width when zero value is used: Controls the printed width of lines set to zero width.

Grid Options

Default UI Menu: Options/Display Options/Grid

Ribbon UI Menu:

Controls the grid type, size, and display. You can set up different grid properties for Model Space and Paper Space. See [Grid](#)

Show grid: Equivalent to displaying the grid using the **Grid** menu or **Grid** toolbar.

Hide grid when finer than: This value applies to distances on your screen which are not related to World or Paper units.

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Type: These grid types are based, by default, on a horizontal baseline. This can be changed on the Advanced Grid page.

Ortho: Orthogonal grid.

Isometric: Aligned along lines measured 30° and 150° from the horizontal axis. This type of grid is used in isometric drawing, which uses perspective drawing techniques to represent 3D objects.

Polar: Points in the polar grid radiate from the origin, and are aligned by their radial distance from the origin and their angular distance from the horizontal axis.

Style: Select **Points** (dots at each grid point), **Crosses** (crosses at each main grid point), or **Lines** (graph paper).

Spacing: Sets the distance between all grid lines, major and minor, without respect to frequency. Select X and Y spacing, or angular and radial values for a polar grid.

Layers

Default UI Menu: Options/Drawing Setup/Layers

Ribbon UI Menu:

Enables you to create new layers, set the current drawing layer, set the properties of layers, and delete layers.

These are legacy functions, use the Design Director or Layer Manager instead.

Geo-Location

Default UI Menu: Options/Drawing Setup/Geo-Location

Ribbon UI Menu:

Geo-Location contains three primary functions: Location, Orientation, and Display.

Location

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Latitude - set the latitude of the drawing in degrees, to six decimal places.

- **South/North** - specify whether the latitude angle is South or North

Longitude - set the longitude the drawing in degrees, to six decimal places.

- **East/West** - specify whether the longitude angle is East or West

Coordinates and Elevation

X, Y, Z - specify the location in the drawing which corresponds to the geo-location. Units are in the model space defined Space units.

Elevation - specify the elevation of the drawing at that geo-location.

Marker

Show - When on this shows a crosshair marker within the drawing, located at as specified by the X, Y, Z values. This maker can be snapped to as a drawing reference.

Size - Sets the relative size of the to the crosshair marker.

North Direction

Angle - Specify the Angle of orientation for North relative to the drawing.

Marker - Displays a compass rose for drawing orientation

On Screen Position - Specify the location of the compass rose:

Not Display, Top Left, Bottom Right, Top Left, and Bottom Left.

Size-- Sets the relative size of the compass rose: Tiny, Small, Normal, and Large.

Sun Position



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

The Sun control is designed to provide a universal control for the display of sunlight in all draft and quality renders.

Use Sun - If on a Sun light is used in all draft & quality renders. The direction of the sunlight points to the 0,0,0 point of the drawing.

Intensity: sets the power of the sunlight It takes values from 0 to 1,000,000 (or 1E + 06), with a default of 1.

Shadows - If checked the sunlight casts shadows.

Use System time and Geo-Location: If selected the date and time is derived from the computer clock and the location is specified by Geo-Location

Use Specified time and Geo-Location: If selected the date and time and time zone (GMT+/-) are user-specified and the location is specified by Geo-Location

Date: Sets the date.

Time: Sets the time.

Timezone: Sets the timezone as relative plus or minus to Greenwich Mean Time (GMT)

Custom: If selected the user may directly specify the Azimuth and Angle.

Azimuth: The user can specify the Azimuth from 0.00 to 360.

Angle: The user can specify the Angle from -90 to 90.

New Sun and Sky shader

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(Available in TurboCAD Platinum only)

New properties "sun dir by geo-location" and "dir by geo-location" added to "Sky" and "Sun" shaders correspondingly.

Lightworks

Default UI Menu: Options/Drawing Setup/LightWorks

Ribbon UI Menu:

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The options on this page are relevant to Render Styles. See [Creating a Rendered View](#)

Transparency Shadow: Specifies whether a render created with a render style will support shadows cast by transparent objects.

Line Styles

(Available in Platinum, Professional)

Default UI Menu: Options/Drawing Setup/Line Styles

Ribbon UI Menu:

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View predefined line styles, modify or delete them, and create new ones.

An object's line style can be set in the **Property** toolbar (see [Property Toolbar](#)) or in the **Pen** page of its **Properties** (see [Properties Window](#)).

If you click **New** or **Modify**, you will access the **Line Style Editor**.

Add to Defaults: New line styles will be saved so that they can be used in future drawings. Otherwise, the line styles will be used only in the current drawing.

Tip: You can use the TC Explorer Palette to view line styles defined for any open drawing. See [Drawing Settings](#).

Applying a Line Style

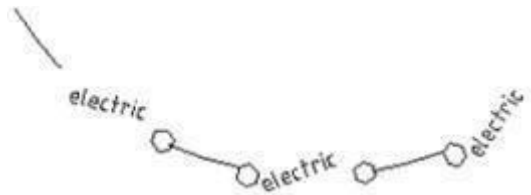
To apply the line style, you can use the **Property** toolbar. You can also set the line style, and adjust its scale, in the **Pen** page of the object's **Properties**. See [Properties Window](#).

1. Create the object, which appears in the default line style.



2. Select the desired line style from the **Property** toolbar.

The line style is applied.



Tip: You can use the TC Explorer Palette to view line styles defined for any open drawing, and to import line styles from one drawing to another. See [Drawing Settings](#).

Line Style Editor

Enables you to edit an existing line style or define a new one.

The example is shown above (line style "Utility") has four components: one dash, one text, and two shapes (hexagons). You can also add dots and spaces. Either click **Add Item** to add another component, or click the relevant button (**Dash**, **Dot**, etc.)

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For each type of line style component, you can set various parameters. For **Shape** you can choose the actual shape and its size and spacing. For **Text** , you can choose the font and angle. The **Preview** window helps you see how the line style will look; use it as a guide when adjusting the parameters.

Note: Shapes are read from *.shx files (not to be confused with *.shx font files), found in TurboCAD's Shapes folder. For Example: My Documents\TurboCAD 2023 Professional x64\Shapes

Once the line style has been created, it appears in the **Line Styles** page of the **Drawing Setup**.



The new style also appears on the **Property** toolbar.

Print Style Table

Default UI Menu: Options/Drawing Setup/Print Styles

Ribbon UI Menu:

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Select the print styles you want in your drawing.

Print styles are specific pen and brush settings you can apply to objects when printing.

See [Print Style Options](#)

Redsdk 6.0

(Available in TurboCAD Platinum, Professional and Deluxe Only)

Note: RedSDK is now offered as only an optional plug-in to TurboCAD 2023 Deluxe, Professional and Platinum. Enhancements described below are only found after the plug-in has been installed. LightWorks has been reintegrated into TurboCAD and will no longer be an optional plug-in

The Redsdk settings allow you to control how object appear when using the Redsdk for draft rendering and hidden line rendering.

Shading

Use background color: When it is turned off (On by default), the option for editing the internal color of objects will become available (by default the color is white).

Front line

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Width by graphic: When this option is on the Width of the visible part of graphic's line is taken from the graphic itself. When it is turned off, the field for editing the width of the visible part of object lines will become available (by default the width is = 0).

Color by graphic: When this option is on the color of the non-hidden part of an object's lines is taken from the graphic itself. When the option is off the option for editing the color of the non-hidden part of an object's lines will become available (by default the color is white).

Backward line

Width by graphic: When on the Width of the hidden part of an object's line is taken from the graphic itself. When it is turned off, the field for editing the width of the hidden part of object lines will become available (by default the width is = 0).

Note: the width of hidden lines must be less than or equal to the width of visible lines.

Color: This option defines the color of hidden lines (by default it is grey; R=G=B=128

Transparency: This option defines the transparency of hidden lines, with the range of 0 to 1 (by default it is 0.1)

Stipple: This option defines the type of dashed line used for hidden lines, with the range of 0 to 255 (by default it is 255)

Draft Mode Settings

Ambient: This field defines the light intensity factor for all light sources of Ambient type (by default it is 10).

Diffuse: This field defines the luminance factor (by default it is 75).

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Specular: This field defines the glare factor (by default it is 50).

Exponent: This field defines the glare attenuation parameter (by default it is 10).

Soften: This field defines glare 'softness' (by default it is 0).

GI Cache - Baking

By baking, we mean pre-calculating a given information, storing it and restituting it, faster, as a later rendering phase. Two types of data will be baked in our example:

The global illumination information of our scene.

The direct contribution of physical lights in the scene.

The baking process for this model takes between a few seconds up to a few minutes, depending on the quality of the baked signal. Then, the replay of the baked cache is either performed in real-time or in software.

Enable - Turns on/off GI cache. By default - **Off**.

Save As - Saves the GI cache to a RED file. Enabled - if GI cache was created or loaded.

Delete - Clears the GI cache. Enabled - if GI cache was created or loaded.

From file - Turn on/off GI cache loading from file. By default - **Off** Enabled - if GI cache is **On**

File name - RED file name. By default - **Empty** Enabled - if GI cache is **On** and From file is **On**

... (Browsing RED file) - Browsing RED file. Enabled - if GI cache is **On** and From file is **On**

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Active View or Render Style name - Source of parameters for GI cache creating. By default - **Active View**. Enabled - if GI Cache is **On** and From file is **Off**

... (Browsing render style) Browsing renders style. Enabled - if GI cache is **On** and From file is **Off**

GI cache computation mode - Choosing GI cache computation mode. By default - **Active Camera View**. Enabled - if GI cache is **On** and From file is **Off**



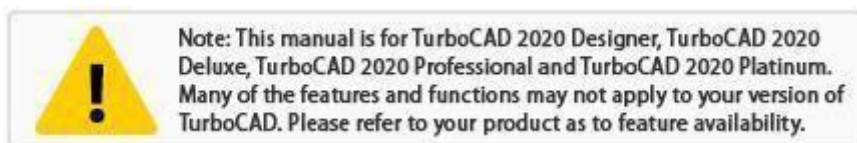
Active Camera view	The GI cache is computed for the given camera only.
Around active camera position	the camera position. The generated GI cache is then suitable to render panoramas.
Around the Named View camera positions	The GI cache is computed for the whole scene, without taking the camera into account.

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Miscellaneous

Export to RED f ile: This option allows you to save the current RedSDK settings.

Render Scene Environment



Default UI Menu: Options/Drawing Setup/Render Scene Environment

Ribbon UI Menu:

The options on this page are relevant while rendering. You can specify a background and/or foreground for the model.

See [Render Styles](#) and [Environments](#)

Render Scene Luminance

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Drawing Setup/Render Scene Luminance

Ribbon UI Menu:

The options on this page are relevant while rendering. You can specify one or more lights to the overall drawing render.

Luminance uses the RedSDK and Lightworks rendering engine and provides a wider range of effects than standard lights. You can also specify luminance for individual objects (as opposed to the overall drawing) by opening the **Luminance** page of the object's **Properties**.

See [Luminance](#)

Drawing luminance can be applied to the WCS (World Coordinate System) or UCS (User Coordinate System). See [WCS and UCS](#)

Space Units

Default UI Menu: Options/Drawing Setup/Space Units

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Ribbon UI Menu:

Controls for setting units in Model and Paper Space.

Note: Templates generally have units built in, but you can change units, or create custom units.

Workspace name: Available in Paper Space only.

System: Selecting English or Metric will affect controls on the rest of the page.

Scale: The ratio of measurements on the page (Paper units) to measurements in the object being modeled (World units).

Architectural: $3/16" = 1'-0"$

Engineering: $1" = 200'$

Absolute: 1:500

Units: The unit of measure for linear measurements.

Show all units: Displays all available units, regardless of whether English or Metric is used.

Line width units: The unit for setting and displaying the line width in the **Pen** page of the **Properties** window.

Precision: The number of decimal digits. For **Fractional** or **Architectural** units, **Precision** determines the accuracy of the denominator, in powers of two ($1=1/2"$, $2=1/4"$, $3=1/8"$,

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etc.).

Text units: Unit for setting and displaying the size of the text.

Area Units: Unit to be used for displaying area.

As Linear Units: Sets Area units to the same unit type as Units.

Use Exponential Notation: Exponential Notation is used in Area fields



Format: Controls the display of numbers:

Decimal - 3.25

Scientific - 1E+01

Fractional - 3 3/16

Architectural 1'-5 11/16"

Engineering (scale): 1"= 200'

Text height specified by: Two methods to adjust text height when the current font is changed.

Text box height (default): In specifying the height value for the text, define the height of the text bounding box. In this case, changing the font does not affect the height of the bounding box.

Character height above the font base line: The text height value will be used as the actual height of the text. This method preserves the text height (but not the height of the text box) when you change the font.

Note: The value of text height is determined by the vertical size of the uppercase letter "A" in the currently selected font. This value is the sum of four components: **External Leading** is the amount of space that the designer of the font suggests should be added between character rows. **Internal Leading** is the amount of space allowed for an accent mark above a character. **Ascent** is the amount of space allowed for the characters that have neither an accent mark above the character nor a character part below the font baseline. **Descent** is the amount of space allowed for the character part below baseline. (Lowercase letters such as "g", "p", and "q" have a character part occupying some space below the font baseline.)

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Convert units after alteration: Forces unit recalculation after changing the units.

Example 1 - Converting a drawing from feet to inches: if unchecked, 1 foot will become 1 inch. If checked, 1 foot becomes 12 inches.

Example 2 - Converting a drawing from inches to mm: if unchecked, 1 inch will become 1 mm. If checked, 1 inch will become 25.4mm.

Convert material after alteration: Forces conversion of material sizes after changing units.

Convert styles properties on switching between spaces: Activates the conversion mechanism when switching between Model Space and Paper Space. Units are typically viewed in a variety of places including Selection Info;

Visualize

(Available in Platinum, Professional, and Deluxe)



Note: This manual is for TurboCAD 2021 Designer, TurboCAD 2021 Deluxe, TurboCAD 2021 Professional and TurboCAD 2021 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Drawing Setup/Visualize

Ribbon UI Menu:

This option lets user set various options for Visualize.

Lighting:

Lighting parameters section allows you to setup ODA Visualize SDK default lighting options.

The section gets enabled if Device Type is OpenGL ES2.

Parameters:

Default presets

Description: Represents default lighting types:

- One light
- Two lights
- Back light

By default: Two lights

Intensity

Description: A double value that represents the new intensity of the default lights.

By default: **0.7**

Ambient Occlusion:

Ambient Occlusion parameters section allows you to setup Ambient Occlusion options.

The section enabled if Device Type is OpenGL ES2.

Parameters:

Enable

Enables Ambient Occlusion effect.

By default: **Off**

Loops

Specifies number of samples which are computed for each Screen Space Ambient Occlusion fragment. This property directly influences the quality and smoothness of Screen Space Ambient Occlusion effect. It is not recommended to set very large values for this property, because number of sampling operations for each image fragment can seriously downgrade processing performance.

By default: **32**

Focus

Specifies radius of Screen Space Ambient Occlusion effect sampling area which is represented as hemisphere in World Coordinates Space. Increasing of radius increases the area of Screen Space Ambient Occlusion effect application. Since different drawings may be drawn in different coordinate spaces, units and dimensions, this property is one of the most important for Screen Space Ambient Occlusion effect configuration.

By default: **1.5**

Power

Specifies additional multiplier for resulting colors of the Screen Space Ambient Occlusion effect. The 1.0 value disables additional multiplication, however results produced by the

Screen Space Ambient Occlusion effect may be too pale and inexpressive. Larger multiplier values produce more expressive and perceptible effect shades

By default: **8**

Blur Radius

Sets Screen Space Ambient Occlusion effect blur. Zero value for this property disables blur effect and the resulting image is crisp. Increasing this value increases amount of blur effect and smoothness of final results:

- 1 — invoke 3x3 Gaussian blur kernel
- 2 — invoke 5x5 Gaussian blur kernel
- 3 — invoke 7x7 Gaussian blur kernel and etc

Range: [0...1000]

By default: **2**

Blur Power

This property can be used to add more expression for Screen Space Ambient Occlusion blur effect results. This multiplier is applied to shading fragments during blur effect computation. You can get more clear blur effect results by increasing power of blureffect.

Range: [0...6]

By default: **2**

Import/Export:

Import from File: This button is enabled if the ODA Visualize SDK mode is set in the active window. Formats OBJ and DWG are available for import now. (Set of formats can be significantly expanded if necessary, all formats that are supported by ODA can be added). The imported objects are not added to the drawing.

Export to file: The button is enabled if the ODA Visualize SDK mode is set in the active window. Format VSF (Open Design Visualize Stream) is available for export now. (Set of formats can be significantly expanded if necessary, all formats that are supported by ODA can be added).

2.11. Object Properties

Default UI Menu: Modify/Properties

Ribbon UI Menu:

You can set an object's properties before or after it is created.

Setting properties for a group of tools (setting properties before creating an object)

Right-click on the tool icon to open the **Properties** window. Or activate a tool, then select **Modify / Properties**. Whatever properties you set here will be applied not only to the selected tool, but to all tools on the same toolbar. For example, if you set **Line** to be drawn in red, the **Polygon** and **Rectangle** tools will also share this color. If you change properties of a tool, the new properties will be assigned to all future objects created with the entire set of tools. However, objects already created will not be affected.

Note: You can save tool properties in template files, so that you don't have to create styles from scratch each time. To do this, set up the properties you want for the tools you commonly use. Then use File / Save As to save the file as a .tct file (TurboCAD Template). Place the template file in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use *File / New, and select New from Template.

Setting properties after creating an object

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You can set certain basic properties, such as color and layer, directly on the **Property** toolbar. For more options, there are several ways to set the remaining parameters: With the **Select** tool active, double-click the object. Right-click anywhere in the drawing and select **Properties** from the local menu.

Click the **Properties** icon on the Inspector Bar.



Open the Selection Info Palette. You can set properties within the table that appears in the palette, or access the object's **Properties** window from within the palette. See [Selection Info Palette](#)

Tip: You can take properties from one object and assign them to other objects.

See [Format Painter](#)

Property Toolbar

Default UI Menu: View / Toolbars

Ribbon UI Menu:

Provides a quick and convenient way of setting some basic properties of a selected object or group of objects, or for setting the default properties of a group of drawing tools. If the **Property** toolbar is not visible, you can display it using the **View / Toolbars** window.



If no objects are currently selected, the settings on the **Property** toolbar apply to the active drawing tool. For example, if you activate a **Line** tool and then change the

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settings on the **Property** toolbar, the settings will apply to all objects drawn with any **Line** tool. If there are objects selected, settings in the **Property** toolbar apply to the selected objects.

Tip: For properties to appear, the number of selected objects must be less than or equal to **Maximum Multiple Entity Property Count**, which is set in the **Program Defaults** page of the TC Explorer Palette.

Entity Style: Applies predefined property settings to a drawing tool. You can save styles in the **Properties** window. This control can be used only to set the properties of a drawing tool; it is unavailable if you are setting the properties of selected objects.

Layer: Set the layer on which objects are drawn, or move selected objects to a layer. See [Snap Modes](#)

Pen Color: Sets the pen color. There are two types of Pen color available. True Color/RGB and Index color. For True Colors You can select a predefined color, or define your own using RGB or HSL values. The last three icons in the color thumbnail rows enable you to set the color by layer, by block or open a menu in which you can change how the colors are sorted.

For Indexed colors you can only select the predefined colors.

For more on colors, see [Color Palette](#)

Pen Style: Sets the line style of the line. See [Line Styles](#)

Brush Pattern: Sets the fill pattern for closed objects. See [Properties Window](#)

Pen Width: Sets the line width.

Font: Available for text objects.

Text Height: Available for text objects.

Property Value Presets

Presets bundle the properties defined for a group of tools for reuse. For example, you can have one preset to create blue, dashed lines on Layer 5, and use another preset to create red thick lines on Layer 2

Presets provide a convenient way to quickly assign a set of properties to objects.

Presets can be accessed on the **General** page of an active tool's **Properties** window.

Presets are only applicable to tools, not objects. You cannot apply a preset to selected objects. You can define a style based on the properties of selected objects by using the Format Painter.

To create a new preset:

Right-click on an icon for a group of tools to open the tool's **Properties** window.

The tool must be active for the **Property Values Presets** field to be editable.

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Note: Properties set for a tool affect all other tools in its group. For instance, setting **Line** properties affects **Rectangle**, **Polygon**, etc. Presets are separate for different sets of tools.

On the **General** page, enter a new preset name and click **New**.

Make any changes to the other properties (**Pen**, **Brush**, etc.). The available properties depend on the type of object (for example, the **Properties** window for **Text** tools contains a **Text** page). These properties will be saved to the new preset.

To edit an existing preset, select it from the **Property Values Presets** menu and make any property changes.

Properties Window

Default UI Menu: Modify/Properties

Ribbon UI Menu:

An object's **Properties** window consists of a series of categories, each on its own page. There are categories that are common to all objects: **General**, **Pen**, **3D**, **Luminance**, **Brush** (for 2D objects) and **Custom** - these are all described in this section.



3D Properties

Properties relating to 3D, either for a standard 3D object such as a sphere or box, or for a 2D object made into 3D by assigning it a thickness.

See [3D Properties](#)

Luminance Properties

Light properties of objects, which is different than the lights contained in the overall model.

See [Luminance](#)

Custom Properties

Attributes you can add to objects via the database.

See [Custom Properties, Database, and Reports](#)

Material Properties

Properties related to material for the 3D objects. They are not enabled for 2D objects.

See [Materials](#)

Brush Properties

Default UI Menu: Modify/Properties/Brush

Ribbon UI Menu:

Brush properties specify the filling properties for closed objects such as circles and polygons, and for flexible text.

Setting the **Brush** pattern for the group of **Hatch** tools establishes the hatch pattern that

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will be applied. See [Hatching](#)

These settings are relevant for closed objects such as circles and polygons, and for flexible text. **Pattern:** Select solid fill, or one of the predefined hatch patterns. To define your own hatch, bitmap, or gradient pattern, see [Brush Styles](#). In the Preview window, the red diagonal line is to help visualize transparency in the case of a solid or gradient fill, or when a **Background Color** is used.

Color: Set the color of the brush.

Convert Color in Accordance with a Current Background Color: This feature is used to override the setting in the Background Color dialog for the selected object.

Scale: Sets the scale of the hatch pattern. A scale of 2 doubles the size.

Angle: Sets the angle of the pattern.

Crossed: The pattern will be drawn a second time, perpendicular to the first pattern.

Background Color: The color used for empty spaces in the pattern.

Origin: By default, hatch patterns use the origin as a point of reference. If you want to modify the placement of hatches without changing their angle, you can modify the point of origin used to place the hatch. **World** uses the WCS, and **Entity** uses the lower left corner of the object being hatched. Use the **Offset** fields to change the reference origin.

Draw Mode: Options for how hatch patterns are drawing over overlapping objects.

Winding: The same hatch pattern will cover areas where filled objects overlap

Alternate: The hatch pattern to be drawn on every other overlapping layer, creating an alternating pattern.

Transparency (%): For solid or gradient fills. A percentage of 0 means the fill is opaque, and 100 means the fill is invisible.

Gradient Fill: Relevant for gradient brush styles, which must be defined using **Brush Styles** (see Gradient patterns in [Brush Styles](#)). The parameters in this section are the same as the ones already set for the gradient pattern. For **Fit Mode** and **Center Point mode**, use **By Brush Style** to keep the parameters as defined for the pattern. Changing either setting overrides the pattern's settings. For **Focus Scale**, a

value of -1 keeps the focus scale the same as the pattern definition. Use a value between 0 and 1 to override it.

General Properties

Default UI Menu: Modify/Properties/General

Ribbon UI Menu:

These are properties generally not related to geometry or other physical characteristics.

Attribute: For text and dimension objects, contains the text string. For groups, contains the group name.

Hyperlink: Specifies the location of a desired file on your hard disk or on a company's network, or specifies a URL. You can type in the field or browse your system or the web to find the desired path.

Layer: Set the object's layer by selecting from the drop-down box. See [Layer Management](#)

Print Styles: Select the print style you want to use when printing. See [Print Style Options](#)

Property Value Preset: A group of properties defined for a group of tools. See [Property Value Presets](#)

Pen Properties

Default UI Menu: Modify/Properties/Pen

Ribbon UI Menu:

Properties of the pen and line style.

Tip: You can also use the Design Director to quickly set the color of an object or group of objects. For single objects, see [Design Director-Graphics](#). For groups of objects, see [Design Director-Categories](#).

Pattern

Select a continuous line or one of many dot-and-dash patterns. If **By Layer** or **By Block** is used, the pattern will depend on the layer or block. There are several predefined line styles to choose from, or you can create your own. See [Line Styles](#).

Dash Scale: Scale of the dot-and-dash pattern.

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Alignment: Align the pattern so that the corners of rectangles and polygons are always solid.

Color

Color: Pen Color with which the object will be drawn.

Convert Color in accordance with Current Background Color: This option is designed to prevent objects from disappearing into a background of the same color (e.g. White on White) This is accomplished by inverting the display color of the object, but not the actual color setting. There are three options:

By Drawing - color conversion will match the global setting in the Drawing Setup under [Background Color](#)

Yes - Color conversion will occur

No - Color conversion will not occur

Width

Scaling: Determines whether the width of the line will be scaled, or remain the same size, when you zoom in and out of the drawing.

Device: The pen's width and pattern sizes are defined in device units (the monitor or printer). If you zoom the line width and pattern size will not change on the screen.

World: The pen's width and pattern sizes are defined in by the drawing spaces units. If you zoom the line width and pattern size will change in accordance with the zoom factor.

Device Width: The pen's pattern size is defined by the drawing spaces units, and pen's width is defined in device units (the monitor or printer). If you zoom the line patterns size will change, but the pen width size will not change. Value: Pen width. A zero width uses one screen pixel, and will print at one unit of the printer's available resolution (a 300 dots-per-inch printer will print a zero width line at 1/300".)

Geometry: Used only when **World** is selected for the Scaling. If checked, TurboCAD creates external and internal "walls" for the object obtained by applying a thickness method to a 2D object. You can render your drawing to see the effects of this option.

Brush Styles

Default UI Menu: Tools/Palettes/Colors and Brushes

Ribbon UI Menu:

The **Brush Styles** palette appears by default in the palette area, on the right side of the screen. This tool enables you to modify existing hatch patterns and create new ones. You can also define and edit bitmap and gradient patterns. Brush styles can be used to fill a closed 2D object, either by using an object's **Brush Properties** (see [Properties Window](#)) or by **Hatching** (see [Hatching](#)).

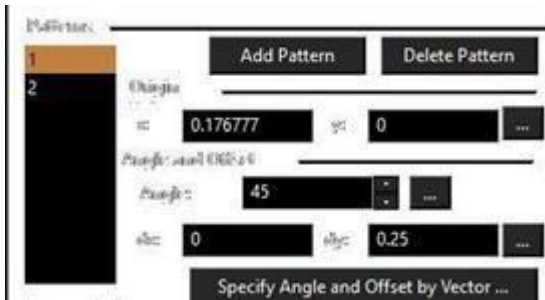
Note: You can use this tool to update existing patterns. Select the hatch, bitmap, or gradient pattern, make the changes, and click Update. Return to the drawing, and the fill will update in each object where it appears. If the update is not immediate, click the object in Select mode.

Hatch Patterns

You can use **Brush Styles** to edit existing hatch patterns and create new ones. As an example, select **Hatch Patterns** for **Brush Style Mode** and set the **Brush Style** to ANSI35.



This style consists of two Patterns - each pattern is a line that repeats at constant offsets. Highlight each pattern number to see the line highlight in red below, in the Preview area.



In this case, Pattern 1 is the dashed line and Pattern 2 is the unbroken line.

To edit a Pattern line, you can change its **Origin Point** manually, or click the 3-dot icon to select an origin on-screen. You can set the **Angle** and **Offset** values the same way. If you click **Specify Angles and Offset by Vector**, you can define two vectors on-screen: the first defines the angle of the line, and the second defines the spacing between repeating lines. Highlight Pattern 1 (the dashed line), and you can see that it consists of

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four patterns.



Dashes are always defined in pairs; odd numbers represent the line segments and the even numbers represent the dashes. The length value for dashes is always negative.

Specify Dashes is used to create a dashed pattern from an unbroken line. Use **Add Dashes** and **Delete Dashes** to modify the dash pattern. You can update pattern lengths manually or use the 3-dot icon to define length on-screen.

The following example shows how to define a new hatch pattern.

1. At the top of the palette, click **New** and enter a new name for the hatch pattern.



The new pattern will be based on whatever pattern was active when the new pattern was created (ANSI35 in this case). Click **Delete Pattern** so that only one pattern line remains.

Click **Specify Angle and Offset by Vector**.

Draw a line on-screen that has the angle you want in the pattern. You can snap to existing points or draw a free-form line.

5. Next, draw a line on-screen that defines the offset between repeating lines.

To make this a dashes line, click **Specify Dashes**.



The first line on-screen defines the length of the line segment of the dashed pattern. Be sure to define this length in the same direction you used to create the original pattern line.

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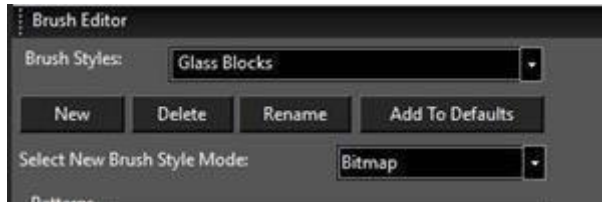
The next vector defines the length of the dash, resulting in two dash patterns. If you want more dash patterns, always in pairs, continue defining vectors in the same direction.

Create more patterns and dashes as needed to get the entire pattern.

When the style is defined, click **Update Style** to use it in the current drawing, or **Add to Defaults** to be able to use it in future drawings.

Bitmap Patterns

Enter the name of the style at the top of the palette.



In the middle section, use the browser to locate the image file you want to use. A preview of the image appears in the lower section. The size of the repeating image is controlled by the **Width** and **Height** values. You can change these values manually, or click **Advanced** for more control.

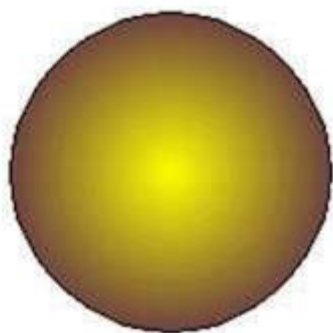
Gradient Patterns

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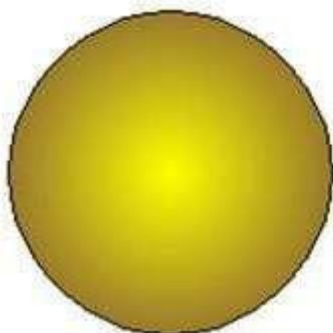
Enter the name of the style at the top of the palette.

There are four types of gradient fills: **Linear**, **Radial**, **Reflected**, **Diamond**, and **Custom**. Each type is explained later in this section. **Mode** (called **Fit Mode** in the **Brush Properties**):

Exact Fit: The gradient completely fills the object itself. In this example, the fill completely reaches the second gradient color at the boundary of the circle.



Fit to Extents: The gradient completely fills the extents rectangle that encloses the object. In this example, the second gradient color is only reached at the boundary of the extents rectangle, which is larger than the circle itself. This is why the color at the circle boundary is lighter than in the example above.



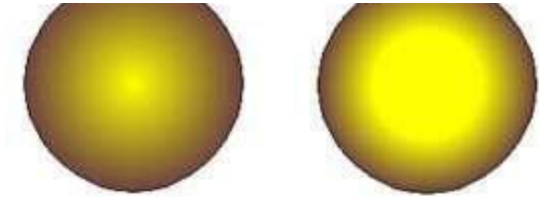
Center Point:

Modes: Select **Extents center** to center the fill at the center of the object's extents rectangle. **Reference point** centers the fill at the object's reference point (see

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[Components of Select Edit Mode](#)).

Focus Scale: This must be a value between 0 and 1. Zero means the color interpolation start from the center (left image below). A value of 0.5 will start the interpolation halfway between the center and the edge (right image below).



Note: Settings for **Fit Mode**, **Center Point**, and **Focus Scale** are set here as the fill defaults. These values can be overridden for filled objects by modifying the object's **Brush Properties** (see [Properties Window](#)).

Linear Gradient

Linear fill moves from one color to the other along a straight vector. Under **Control Points**, highlight **First** and set the **Color**. Do the same for **Second**.

By default, the **First** color runs vertically along the left side and changes to the **Second** color along a left-to-right vector. To change this vector, click the icon with the three dots. Define the new vector on the screen. The first point of the vector controls where the **First** color appears.

Radial Gradient

Radial fill moves from one color to the other along a radial path. Under **Control Points**,

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highlight **Center** and set the center color. Highlight **Radius** and set the outer color. Set the radius manually, or click the 3-dots icon and set the radius on-screen.



Reflected Gradient

Reflected fill consists of one line of reflected color, fading to another color on either side. moves from one color to the other along a radial path. Under **Control Points**, highlight **First** and set the reflective color. Highlight **Second** and set the fade color. Set the angle of the reflection line manually, or click the 3-dots icon and set the direction on-screen.



Diamond Gradient

Diamond fill moves from from the center outward in four directions, each separated by 90 degrees. Under **Control Points**, highlight **Center** and set the color of the center and linear patterns. Highlight **Radius-Vector** and set that appears in between the **Center** color lines.

By default, the **Center** color runs vertically and horizontally to either side. To change this vector, click the icon with the three dots.

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Define the new vector on the screen. The first point of the vector controls where the **Center** color appears. The second point of the vector controls the angle of the four lines.

Custom Gradient

Use this type to create a custom polyline and apply colors to each polyline point. The results will resemble a **Diamond Gradient**, but can have numerous points, each with its own color. **Color 1** runs from the center outward to each point. By default there are three other points (triangular polyline). To change the polyline, click the icon with the three dots.

Define the polyline on the screen, proceeding in the order of **Color 2**, **Color 3**, and so on. The polyline automatically closes, and cannot intersect itself. Press space bar to finish. You can also select an existing polyline from the drawing; click the arrow icon in the Inspector Bar to do this.

If the polyline has more than three points, the number of **Control Points** increases, and you can set a color for each point.

When the style is defined, click **Update Style** to use it in the current drawing, or **Add to Defaults** to be able to use it in future drawings.

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2.12. Palettes

Default UI Menu: Tools/Palettes

Ribbon UI Menu:



TurboCAD palettes are convenient screen areas used for performing common tasks and for obtaining information. By default, the palettes are displayed on the right side of the screen, and contain tabs for easy display and switching. The palettes commands are available on the **Tools** menu.

You can also display the **Palettes** toolbar by right-clicking on any toolbar area and selecting **Palettes**.



To customize how palettes are displayed, see [Customize Palettes](#)

See Also:

[TC Coordinates Export](#)

[Database Connection Palette](#)

Type a word to search to browse through the commands and materials. The first suitable item is highlighted. Buttons next to the input field highlight the previous or next suitable item. Successful search options are added to the list and saved between sessions.

The Find box is used in Materials, Luminance, Environment and Advance Rendering style palettes:

Find can also be used to search through the Toolbars and Menus under the Options menu.

Blocks Palette



Default UI Menu: Tools/Palettes/Block

Ribbon UI Menu:

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A block is a collection of objects combined into a single object. Blocks are useful for storing complex, common drawing objects that you will use more than once. Blocks are stored in the drawing's internal block library and a block reference, and not the actual object, is inserted into the drawing. When a block is edited, all instances of that block in the drawing are updated. See [Blocks](#)

User can scroll through the Blocks Palette via mouse wheel, find block within the Blocks Palette, or resize blocks previews as well.

Smallest Preview:

Largest Preview:

BIM Palette

Default UI Menu: Tools/Palettes/BIM Palette

Ribbon UI Menu:



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Now you can use BIM Tool and IFC BIM Data via BIM Palette.

Enable BIM Data

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To let the BIM Palette work, you need to enable it by clicking over Enable BIM Data.

Upon selecting Yes, you are displayed with the BIM Data in the Palette.

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IFC BIM Data

IFC stands for **I**ndustry **F**oundation **C**lass, and these are the international standards for types of objects used in CAD software, design, and BIM (Building Information Management) software. By enabling IFC BIM data from an imported drawing, you can to use the property sets that are part of IFC standard.

You can choose property set from the Selection Info palette, under the General option.

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You can also set or retrieve data about the IFC file itself, including the creator and the owner of the file, and the organization associated with the creator.

Edit BIM Property

You can Edit the BIM properties by clicking over Edit BIM Property. To enable the option, you need to click/select the BIM object from the BIM palette. When clicking Edit BIM Property, the following window opens to let you edit the BIM Property.

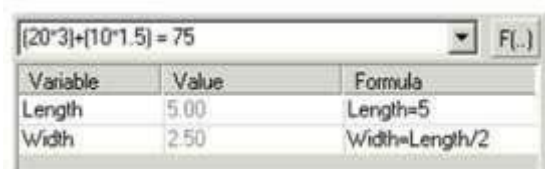
Calculator Palette - Variables Palette

Hotkey: F2 (Ctrl+F2 to close)

Default UI Menu: Tools/Palettes/Variables

Ribbon UI Menu:

Enables you to make numerical calculations within TurboCAD, and to define variables.



You can also use this palette to assign constraints to dimensions. See [Constraining Dimensions](#)

Mathematical Calculations

To perform a calculation, enter the values in the top field of the palette, using parentheses as needed to create inner expressions. No spaces are allowed. The four

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mathematical operators should be entered as +, -, * (multiply), and / (divide).



To calculate the result, press Enter.



You can also use these commonly-used values in place of numbers:

pi =3.1415926 e =2.71828

The function (F) button enables you to perform a function on the value in the parentheses. For example, select **sin**  to get the sine of an angle.

sin  appears in the calculator field (you also could have entered this expression yourself manually).



Sine is calculated for radians, so if you have a value in degrees it must be converted. Look in the functions menu under **Units Conversion** and select rad(a), which converts degrees to radians.



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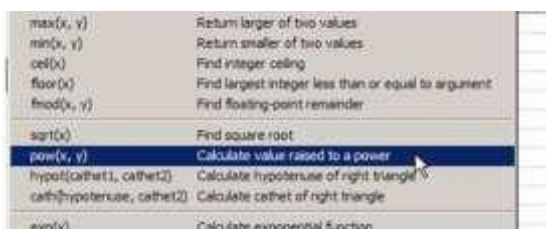
Arrange the expression so that the parentheses are in the correct places, and include the angle in degrees (45 in this case):



Press Enter to obtain the result.



Some functions require two values, such as exponentials. **pow(x, y)** raises x to the y power.

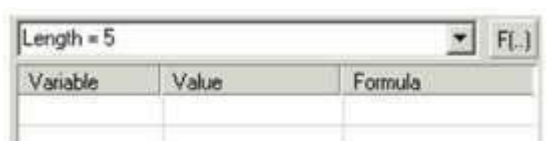


Include the values and press Enter to get the result.



Defining and Using Variables

In addition to calculating numerical equations, you can use the Calculator Palette to define variables. Subsequent values and variables can then be based on variables you've already created. In the calculator field, enter the variable definition in the format "variable name=value" (in this example, Length=5). The value can be a number, function, or another variable. Variables are case sensitive and cannot have spaces.



Press Enter, and the variable, value, and formula are listed in the list below.

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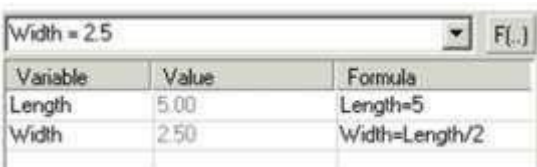
Length = 5		
Variable	Value	Formula
Length	5.00	Length=5

You can also enter a variable and variable or formula directly in the fields of the list. The next variable can be based on an existing variable. In the calculator field, enter an equation that uses a variable name, such as "Width=Length/2."



Width=Length/2		
Variable	Value	Formula
Length	5.00	Length=5
Width	2.50	Width=Length/2

Press Enter, and the new variable appears, with its calculated value.



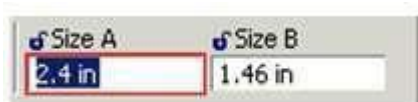
Width = 2.5		
Variable	Value	Formula
Length	5.00	Length=5
Width	2.50	Width=Length/2

Using Expressions in Data Field

Numbers or variables from the Calculator Palette can be used as values in the Inspector Bar or Coordinate fields. As an example, define **Length** and **Width** variables, and draw a rectangle (see [Rectangle](#)). Select the first corner point, but not the second point.



Press Tab to access the first field in the Inspector Bar. The value is highlighted.



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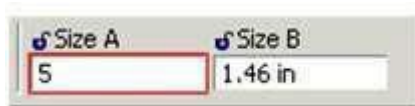
Press F2 to access the Calculator Palette. In the calculator field, type Length and press Enter.



Variable	Value	Formula
Length	5.00	Length=5
Width	2.50	Width=Length/2

Note: You could also enter a mathematical expression based on numbers, or an expression that includes one or more variables.

Back in the Inspector Bar, the first field contains the new value.



For subsequent fields of the Inspector Bar or Coordinate Fields, do not press Tab to scroll. Instead, place the cursor in the desired field, then press F2 to access the Calculator Palette.

Colors and Brushes

Default UI Menu: Tools/Palettes/Colors and Brushes

Ribbon UI Menu:

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Displays all colors and brushes currently available. See [Color Palette](#) for details on adding or modifying colors. By default, the Colors and Brushes palette shows colors on the top and brushes (hatch patterns) on the bottom.

The color selected at the top and the brush pattern selected at the bottom appear in the **Property** toolbar. These properties become the default for the current tool and other

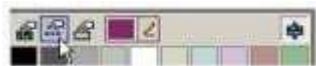
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tools in the tool group.  For example, if **Line** is active and you set a color, this color will become the default for **Rectangle**, **Polyline**, etc. The icons at the top of the palette are as follows: **ColorPalette**:



Opens the palette in which you can create new colors or modify existing ones. See [Color Palette](#).

Line Style Editor:



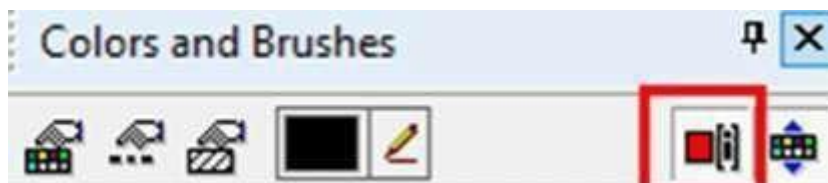
Opens the **Line Styles** page of the **Drawing Setup**. See [Line Styles](#). **Brush Style Editor:**



Opens the **Brush Editor**, in which you can create new brushes or modify existing ones. See [Brush Styles](#). **Pen Color:**



Click the color swatch to replace the color with one from the palette. If you click the pen icon, you can then click any TurboCAD object to select its color. To pick the color of any other object, i.e. objects in other applications, keep the left mouse button pressed and release it when hovering over the object whose color you want. **Index Colors:** Toggles on the display of indexed colors instead of RGB colors. Indexed colors correspond to the indexing of colors in AutoCAD.



Expand Colors:

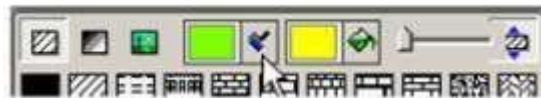
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Toggle this icon off if you don't want the colors displayed. The icons at the bottom of the palette are as follows: **Show hatches, gradients, or bitmaps:**



Filters what types of brushes are displayed. Hatches are simple repeated line patterns. For details on gradient fills, see [Brush Styles](#), and for bitmaps, see [Brush Styles](#). **Brush**



Color and Fill Color:



Sets the colors of the hatch pattern lines and the empty space between lines. If you click the brush or fill-can icons, you can select color on screen as described for **Pen**

color above. **Brush Transparency:**



Controls the transparency of the brush pattern and fill. **Expand brushes:**



Toggle this icon off if you don't want the colors displayed.

To see how Colors and Brushes works, set a color, line weight and brush for the **Circle** tool group, and create a circle.



Select the circle, and click another color in the upper section of the palette.

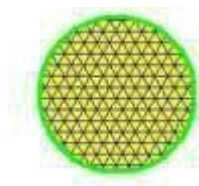
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The color of the circle updates.



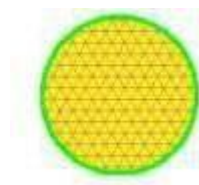
Change the brush pattern the same way - first select the circle, then click a different pattern from the lower section of the palette.



To easily change the brush color, select the circle and right-click on the color you want for the brush.



The brush pattern (not fill color) updates to the new color.



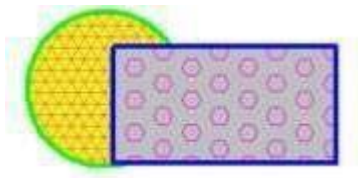
Note: The above changes only affect the selected circle. If you create a new circle, or use any other tool in the **Circle/Ellipse** group, the properties you originally set will be

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used. You can change the default properties simply by activating a tool and setting new properties.

Activate **Rectangle** (or any other tool in the **Line** group), and set a new pen color, brush, and brush and fill colors.

6. Create a rectangle that overlaps the circle.

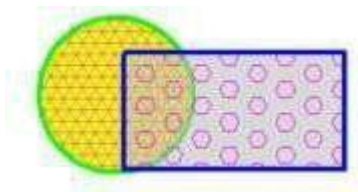


Any tool you use in this group will have these properties.

Select the rectangle and lower the **Brush Transparency**.

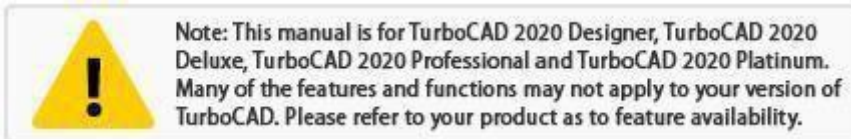


You can now see through the rectangle.



Note: The rectangle is in front of the circle because it was created after the circle. For details on stacking, see [Stacking Objects](#)

Design Director



Default UI Menu: Tools/Palettes/Design Director

Ribbon UI Menu:

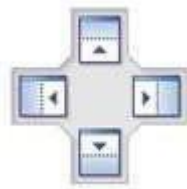
A convenient place to perform commands related to layers, workplanes, views, cameras, and objects or object groups. See [The Design Director](#)

Docking Palettes

Palettes can be docked just like toolbars (see [Toolbars](#)). In addition, palettes can be docked onto other palettes to create palette groups. Drag a palette onto the desktop by its named tab (the tab that contains the name of the palette. The palette must actually be displayed before it can be dragged onto another palette group. To dock a palette to any of the four edges of the TurboCAD window, drag the palette toolbar to the relevant

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arrow, either displayed at the center of the drawing area, or along the desired edge.



Note: These arrows are displayed only if **Docking Stickers** is checked on the **Palettes** page of the **Customize** window (**Tools / Customize**).

To combine palettes into a group, drag more palettes onto a docked palette.

To replace palettes, simply drag their tabs back to the palette area.

Drafting Palette



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/Palettes/Drafting

Ribbon UI Menu:

Enables you to place standard views of a model into Paper Space. You can also create sectional views. See [Drafting Palette-Creating Standard Views](#)

Environments Palette



Default UI Menu: Tools/Palettes/Environments

Ribbon UI Menu:

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Render scene environments (see [Environments](#)) can be assigned to a drawing to enhance its rendering. Options are available for the model background and foreground. For details on using the palette, see [Using the Environments Palette](#)

Library Palette

Default UI Menu: Tools/Palettes/Library

Ribbon UI Menu:

Symbols and parametric parts are groups of objects available for repeated use. While groups and blocks are internal to a drawing, symbols and parts are external files. The **Symbols** palette is used to display and insert symbols found in the various symbol libraries - both those included with TurboCAD installation and those you create yourself. You can also use the palette to save symbols. See [Library](#)

Luminances Palette



Default UI Menu: Tools/Palettes/Luminances

Ribbon UI Menu:

Luminances (see [Luminance](#)) are light types that are added to an object, as opposed to lights added to the overall model. See [Using the Luminances Palette](#)

MacroRecorder Palette



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/Palettes/MacroRecorder

Ribbon UI Menu:

Creates and plays scripts - scenarios of creating and editing objects and manipulating their properties. You can also record object transformation - moving, copying, scaling, and rotating. Scripts can be saved for future play.

Materials Palette



Default UI Menu: Tools/Palettes/Materials

Ribbon UI Menu:

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Materials (see [Materials](#)) can be assigned to 3D objects in order to create more realistic rendering. See [Using the Materials Palette](#)

Measurement Info



Default UI Menu: Tools/Palettes/Measurment Info

Ribbon UI Menu:

Displays measurements calculated by the **Measure** tools. You can measure point coordinates, distance and perimeter, angle, and area. See [Measuring](#)

Parametric Part Script Editor Palette



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Default UI Menu: Tools/Palettes/Parametric Part Script Editor

Ribbon UI Menu:

Enables you to write or edit scripts for parametric parts. See [Parametric Part Script Editor](#)

Selection Info



Default UI Menu: Tools/Palettes/Selection Info

Ribbon UI Menu:

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Displays information about the currently selected object or objects, such as entity type, measurements, location in the drawing, and physical and engineering properties.

User can now select Object Name in the Selection Info Palette Options dialog. The name displays the value set in "Info" attribute for that particular object.

Selection info palette shows block insertion structure as well.

The Selection Info palette shows the Block Insertion structure. This feature does not allow the editing, but only visualizing the structure.

You can also use this palette to edit 3D objects. See [Editing 3D Objects using Selection Info](#)

Style Manager



Default UI Menu: Tools/Palettes/Style Manager

Ribbon UI Menu:

Enables you to define styles for commonly-used architectural items, such as windows, doors, and tables.

See [The Style Manager Palette](#)

TC Explorer Palette

Default UI Menu: Tools/Palettes/TurboCAD Explorer

Ribbon UI Menu:

A multi-purpose palette, enabling you to browse files, view components (layers, blocks, etc.) of all open files, and copy components from one file to another. See [The TC Explorer Palette](#)

Tools Palette

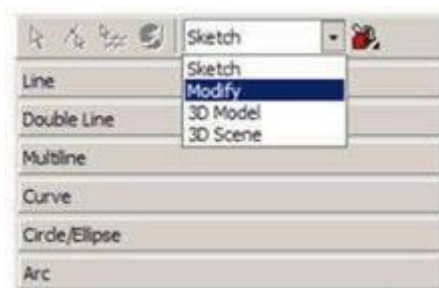


Default UI Menu: Tools/Palettes/Tools

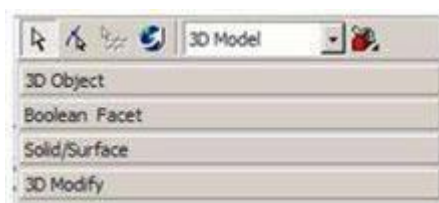
Ribbon UI Menu:

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A palette that enables you to view commonly-used sets of tools. The Tools Palette contains, by default, four templates of tool groups. The default template is Sketch, which contains toolbars for **Line**, **Double Line**, **Curve**, etc.



If you switch to another template, such as **3D Model**, you will see tools relating to 3D modeling: **3D Object**, **Boolean Facet**, etc.



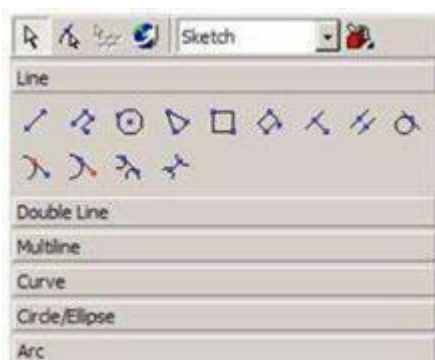
Open a set of tools to see what tools are listed within. These are the same tools you would see in the toolbar of the same name. For example, the **Line** toolbar is available as a separate toolbar, or as a fly-out toolbar along the left vertical toolbar. But if you use **Line** tools often, it is handy to keep them open in this palette for easy access.

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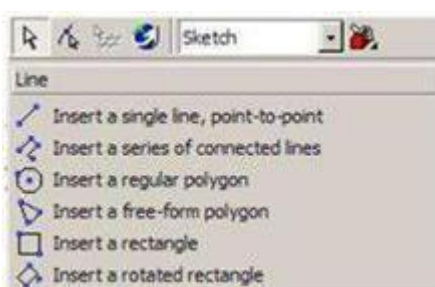
You can also control how these tools are listed in the palette. Click the toolbox drop-down icon on the Tools Palette toolbar. **Icons and Tips** is shown below, each tool has an icon and a tool name listed.



Icons Only displays only the icons. This is handy if you are already familiar with the tools and want to save space.

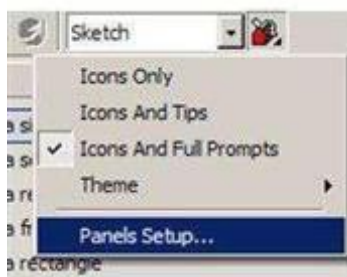


If you are not familiar with each tool, **Icons and Full Prompts** shows the icon along with a complete tool description.



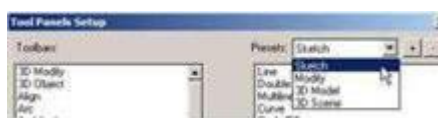
You can also create new templates. Click the toolbox icon and select **Panels Setup**.

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Note: If you select **Theme**, you can control the look and colors (skins) of the Tools Palette.

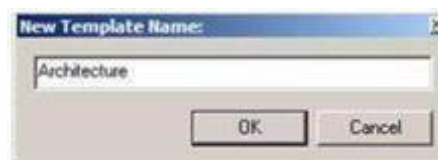
The default templates, or panels, are listed here:



To create a new template, click the + icon.



You are then asked for a template name. For example, if you create architectural drawings, name this template "Architecture."



On the left side of the window, find the toolbars you want to add to this template. Click the right-facing arrow to add it to the list. (Or double-click on the toolbar name.)



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Continue adding toolbars. You can use the **Move Up** and **Move Down** icons to arrange the order of the template.



Click OK, and the "Architecture" tool group appears in the palette.



Event Viewer

(Available in Platinum, Professional)

Default UI Menu: Tools/Palettes/Event Viewer

Ribbon UI Menu:

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The Event Viewer palette creates a log detailing Error and Warning events effectively improving access to the TurboCAD diagnostic information.

Local menu

Clear - clears all events

Write to Log File - turns on/off the recording to the log file.

Open Log File - opens the log file in a text editor.

Level

The icon shown under the 'Level' field can have 3 colors with the values: YELLOW = Warning, RED = Critical, and GREEN = Info

If the 'Show' parameter of a warning dialog is set to 'Off', the icon in the Event Viewer will be displayed with a strikethrough X.

Image Manager Palette

(Available in Platinum, Professional)

Default UI Menu: Tools/Palettes/Image Manager Palette

Ribbon UI Menu:

Enables you to manipulate all the raster images in the current drawing. You can insert files directly from this window using **From Image List** . You can also select Multiple files by using "New" button and importing multiple images in the Image Manager palette.

Insert

Insert a new image by drag and drop.



Use the PPI (pixel per inch) option or the Width/Height options from the Inspector bar to set the size of the image.

The PPI image property is also available from the Selection Info Palette



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The toolbar at the top of the window can be used to change the image list as thumbnails, list, or detailed list.

Image List: Lists all images inserted into the drawing. The list can be sorted by name, status, or path. To edit a name, select it and then click it again. You can then enter the new name.

New: Adds a new image file to the list.

Delete: Deletes an image file from the list.

Browse: Browse to replace the source of a drawing.

Warning!: the same name from the original source will continue to be used.

Ext ract: Option to dis-embed and save and embedded image

Embed: Saves the image file in the drawing as an embedded object, and not as a reference to an external file.

Resize: decrease the resolution of the embedded image:

open Image Manager Dialog or Image Manager Palette;

press "Resize" button for embedded image or "Embed" for not embedded one;

set by slider desired image resolution and press OK;

press "Apply" button. **Warning!:** The decreasing resolution of image is irreversible.

Unload: Unloads image data from work memory.

Reload: Reloads image into work memory.

Properties: Opens the Image Visualization Properties* window, which is useful for

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setting the image quality for large images.

OpenGL: Limits image output to the maximum OpenGL texture size (1024x1024 is supported by most hardware), and resizes image to a square view. For example, a 400 x 300 image will be resized to 512 x 512, thereby losing accuracy. As you increase the zoom, the image may become blurry.

GDI+: Supports large images without resizing, maintaining high accuracy. Not all hardware can render this way efficiently, so this method may be slower than OpenGL.

Filtering quality: Controls scaled or rotated images.

Nearest point: The pixel with coordinates nearest to the desired pixel value is used, resulting in a set of large squares when viewed closely.

Bilinear: A weighted average of a 2 × 2 area of pixels surrounding the desired pixel is used. This is the most common filtering algorithm, adding smoothness in large scales.

Trilinear: A more exact extension of **Bilinear** and is practically indiscernible. But it is unsupported with OpenGL.

Note: These parameters are not file-specific; they affect visualization of all images in all TurboCAD files.

Choosing Multiple Images:

(Available in all Variants)

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You can now select Multiple files from Open Dialog box when importing pictures in the Image Manager palette.

Demo Video:

https://youtu.be/_9_QIJn5zmQ

GeoMarks and TAP files

(Available in Platinum, Professional)

Default UI Menu: Tools/Palettes/Geo Marks Palette

Ribbon UI Menu:

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The Geomarks palette allow you to open TAP file created in IMSI/Design TurboApps products.

To open a TAP file:

Open the GeoMarks palette.

Click the Browse button.

Browse to the file you wish to open.

4. Select the file and click **Open**.

The contents of the TAP must be unpacked. Select a folder to unpack the TAP content, or click Make New Folder to create a new folder, or press Cancel to use a temporary folder.

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You can click on any link to view its content.

The controls are explained below:

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If a TAP file has been created in TurboApps with markups, then TurboCAD now has the ability to display these mark-ups when a TAP file is imported and opened into the program.

2.13. TurboCAD Content Server Palette

Default UI Menu: Tools/Customize/Palettes/Content Server

Ribbon UI Menu:

This Palette shows directory structure of the server.

Content/Folder Structure 58 Folders —Each folder contains SAT and thumbnails 4 Example Folders of the 58 —ANSI, BSI, DIN, ISO Example contents of ANSI Folder (39 folders) —BEAM, HEXB, EYEB, CRSS, WASH Example contents of HEXB folder (8 folders) —Hex Bolt, Hex CAP, Hex HD Example contents of Hex Bolt folder (36 folders) --250-20 UNC, 250-28 UNF Example content of 250-20 UNC (2 folders) —NONE, SIMP Example content of SIMP (13 folders) —HEX BOLT 250-20 UNC 0.5.sat

Folder Names ALPLST GLOBE KHK OILES SPC ALSKCB GRLK KS ORMOTR THK ANSI HHS MINR PACSCI TOCH AS HMND MISUFA PEM TOK BERG IGUS MISUPD PIC TOM BSI IKO MISUPM PUNCH TORR CARR INA NAT REID TORRFAF COOPER ISO NMB REYNOLDS TORRKLN DECCO JERG NOLU RUL VNLC DIN JIS NOR SDP WINCO DME KEY NSK SKF FUTA KGMCO NTN SOI

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Usage

1. User will activate the Application on each run, this will invoke an XML file download to load server directory structure.

1. First time activation will invoke iDRM subscription from the Palette.

Subsequent runs will check subscription in the background.

User can un-register iDRM subscription using the button in the palette.

1. Folder structure will appear in the top window, while preview of files in selected folder will be shown in the below window. Preview is only visible when leaf folder is selected that contains the files.

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1. Downloaded file are placed in ""C:\Users\\Documents\TurboCAD 2023 x64\ContentServer\ContentPltLog.log" folder
2. A log file is also made in "C:\Users\\Documents\TurboCAD 2023 x64\ContentServer\ContentPltLog.log". Sample log file is attached. Any crashes on users' PC can be traced using this.
3. If thumbnails of preview are not downloaded already then they will be downloaded each time a leaf folder with files is clicked on.

To Download

1. Check the item(s) (including folders) and press download button.
2. Double Click on the preview item and the part will be downloaded
3. User can drag and drop the items in the drawing. If the item is not downloaded, the item will be downloaded during the drag-n-drop.

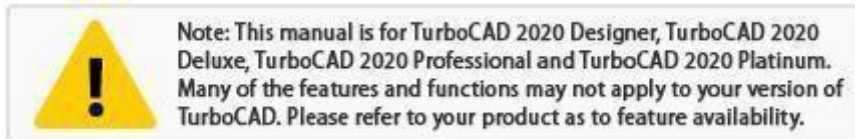
Some things to keep an eye-out:

1. Usability of the App (UI and UX)
2. Multiple PC downloading simultaneously
3. This is a FTP server so any or all things you test for the server

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4. Any crashers/failed downloads
5. Thumbnail downloads for preview
6. iDRM integration

2.14. The TC Explorer Palette



Default UI Menu: Tools/Palettes/TurboCAD Explorer

Ribbon UI Menu:

This palette enables you to perform many operations in one place. It serves as a file browser with previews, and displays components and editable parameters for all open drawings.

Drawing Settings

The **Drawings** branch of the tree enables you to view components of all open drawings.

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Click the **Drawings** header to see a list of all open drawings.

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For each drawing there are six components you can view, edit and/or share between drawings.

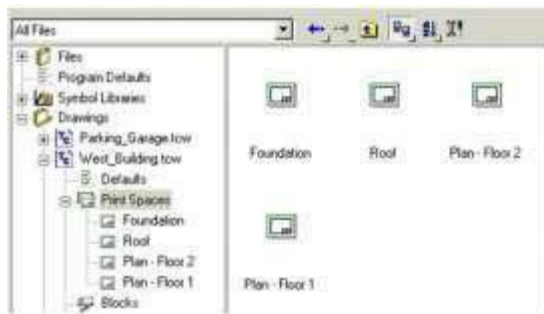
Drawing Defaults

Here you can control default drawing settings - parameters that apply onto to the relevant drawing. For example, you can change the colors of the major and minor grid lines. Nearly all of these parameters can also be set in **Options / Drawing Setup**.

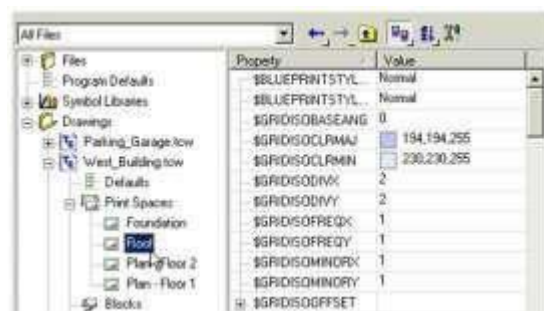
Print Spaces

Lists all paper spaces in the drawing. (See [Paper Space](#))

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If you highlight a Paper Space, you can edit its properties.



Blocks

Displays all blocks defined in the drawing (see [Blocks](#)). Highlight a block's icon or thumbnail to see a preview of the block's contents.

You can use the palette to insert blocks from one open drawing into another open drawing. For example, the West_Building.tcw file contains two blocks. If you open East_Building.tcw, drag the Desk_Large block's icon into the drawing. The block will then be listed under East_Building's blocks. It will also appear in that drawing's Blocks palette.

Layers

Displays all layers defined in the drawing (see [Snap Modes](#)).

You can use the palette to import layers from one open drawing into another open drawing. This imports a layer's name, color, style, width, etc., but does **not** import a layer's objects. For example, the West_Building.tcw file contains four layers. If you open East_Building.tcw, drag the Electrical layer's icon into the drawing. The layer will then be listed under East_Building's layers, and will appear in **Options / Layers** as well.

To change an object's layer, select the object and then double-click the desired layer's icon. If you double-click a layer belonging to another drawing, that layer will automatically be imported into the current drawing. The active drawing tool will automatically be placed on the double-clicked layer.

Line Styles

Displays all line styles defined in the drawing (see [Line Styles](#)). Highlight a line styles' icon or thumbnail to view it.

To import a line style from one open drawing into the current drawing, drag and drop or double-click its icon. To change an object's line style, select the object and then double-click the desired layer's icon. If you double-click a line style belonging to another drawing, that line style will automatically be imported into the current drawing. You can

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also make any line style active by double-clicking its icon. All new objects will be created in the current line style. You can also change line styles while creating objects; for example, you can select one corner point of a rectangle, change the line style, and complete the rectangle.

Brush

Displays all brush (hatch) styles defined in the drawing (see [Properties Window](#)). Highlight a line styles' icon or thumbnail to see how the line style appears.

To change an object's brush, select the object and then double-click the desired brush's icon. You can also make any brush active by double-clicking its icon. All new objects will be created in the current brush. You can also change brushes while creating objects; for example, you can select one corner point of a rectangle, change the brush, and complete the rectangle.

Settings

Lists parameters such as units, fonts, and tables currently defined in the drawing.

One of the more important of the drawing defaults is the \$MIRRTEXT under Drawing Defaults. This setting controls. When the mirror tool is used block attributes are not flipped if the drawing variable \$MIRRTEXT is set to 0, and are flipped if it is set to 1. \$MIRRTEXT is set to 0 by default.

General Settings

All branches of the explorer tree except the last one (Drawings) enable you to set parameters for the program as a whole.

Files

This branch acts just like Windows Explorer. The files that are listed for a highlighted folder depend on what **Filters** are set. You can list all files, only TurboCAD files, only image files, or you can define your own filter for files of specific extensions.



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You can expand a folder by clicking the "+" icon, or by pressing the * on the numeric keypad. If you highlight a TurboCAD file (*.tcw) or an image file, a preview of the file will appear.

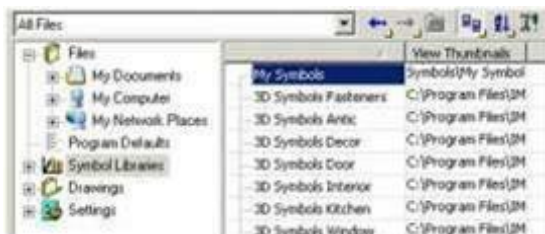
Program Defaults

Here you can control default program settings - parameters that are active for all files. For example, you can change the aperture size and toggle **Auto Add Constraints** on and off. Most of these parameters can also be set in **Options / Program Setup**.

Symbol Libraries

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Enables you to change the path for user-defined symbols. You can also drag symbols as you would from the Library palette.



See [Library Folders](#)

TC Explorer Toolbar

File filter: Enables you to select which files will be listed for a selected folder of the explorer tree. You can define your own file filters in the **Palette Options** (last icon).



Back and Forward: Scrolls backward or forward in the explorer tree.



Level Up: Moves up to the higher folder in the explorer tree.



View: Determines how the contents of the folder are displayed: icons, thumbnails, detailed list, etc.



Sort: Arranges the contents of the folder by date, size, etc.



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Palette Options: Sets the palette display and file filters.



The **Panel** tab contains choices for panel appearance, and enables you to select what components of the palette are visible.

The **Thumbnails** tab enables you to control the size, shadows, and borders of thumbnails.

Image Size and **Space** are both measured in pixels. Space refers to pixels around the icon plus its name. The **Filters** tab enables you to add, delete, or edit file filters. Filters control which files appear in a folder's contents, and are set according to file extensions.

2.15. Working with Multiple Windows

Default UI Menu: Window

Ribbon UI Menu:

TurboCAD uses a multiple document interface (MDI). This means that the TurboCAD application window can contain multiple windows, each of which contains a TurboCAD drawing. Whenever you open a drawing or start a new drawing, the drawing will appear in its own window within the TurboCAD application window. You can also use multiple windows to display varying views of the same drawing.

One advantage of the multiple document interface is that it allows you to intuitively move and copy objects from one drawing to another.

The window commands can be accessed from **Window** menu or from the **Window** toolbar, opened by right-clicking on any toolbar and selecting **Window**.

Creating and Displaying Windows

These commands apply to all open windows, whether or not they belong to one drawing or multiple drawings.

New Window

Default UI Menu: Window/New Window

Ribbon UI Menu:

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Opens another window for the currently active drawing. The contents of this window are identical to those of the active window, but are displayed in **Zoom All** (zoomed so that all objects are visible in the window). If the current window is maximized, the new window will also be maximized.

Cascade

Default UI Menu: Window/Cascade

Ribbon UI Menu:

Resizes and arranges all windows so they overlap, with their title bars showing.

Tile

Default UI Menu: Window/Tile

Ribbon UI Menu:

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Resizes and arranges all windows so that they are all visible, do not overlap, and are laid out to fill the screen area.

Arrange Icons

Default UI Menu: Window/Arrange Icons

Ribbon UI Menu:

When you minimize a drawing window, Windows shrinks it to an icon. Use **Arrange Icons** to line up these minimized drawing windows along the bottom of the TurboCAD window. (The command does not affect windows that have not been minimized.)

Close All

Default UI Menu: Window/Close All

Ribbon UI Menu:

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Closes all open windows. You will receive a warning if any file has unsaved changes.

Hide Caption

Default UI Menu: Window/Hide Caption

Ribbon UI Menu:

Removes the title bars from open windows.

Auto Resize

Default UI Menu: Window/Auto Resize

Ribbon UI Menu:

Preserves the relative proportion of windows when the TurboCAD window is resized.

Auto Align

Default UI Menu: Window/Auto Align

Ribbon UI Menu:

You can resize a group of windows if the following is valid for every window: one side of the window lies on a line common to the entire group. This is the line of alignment. The aligned windows can be on the same side of the line of alignment or occupy both sides of it. When placed on opposite sides, the aligned windows may touch each other or not. Occupying the same side of the line of alignment, windows may overlap each other or be separated by space. In all these cases, whenever you catch the side of a window "belonging" to the line of alignment and then drag the window to change its size in the direction perpendicular to the line of alignment, all the windows "belonging" to the same line of alignment will change their size accordingly.

Moving and Copying Between Different Files

There are two basic methods for copying and moving objects between drawings: copy and paste (or cut and paste), and OLE drag and drop.

Cut and Paste

You can use the Windows clipboard to transfer objects between drawings.

1. Select the objects in the source document that you want to copy or move.

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Select **Edit / Copy** (Ctrl+C), or **Edit / Cut** (Ctrl+X). This places a copy of the objects on the Windows clipboard. **Copy** leaves the source objects in place, while **Cut** removes them.

Activate the target document by clicking in its window or on its title bar, or by choosing it from the **Window** menu.

Select **Edit / Paste** (Ctrl+V). The objects will be pasted from the Windows clipboard into the target document. The pasted objects will appear at the center of the target document window.

Drag and Drop

OLE drag and drop provides a simple and intuitive way to copy objects between drawings. You can drag objects from one drawing window into another, and you can drag objects stored in block and symbol libraries from their palette into another drawing.

Select the objects that you want to drag into another drawing.

Place the cursor over the reference point of the selection, and it becomes a double arrow.

Drag the selected objects into the window of the target document. An outline of the selection will follow the mouse cursor as you drag. Release the mouse button to place the objects.

Warning: The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. It is therefore not a recommended method for moving objects that require precise placement.

To drag blocks and symbols, simply drag them from their palette into the target drawing window.

Multiple Windows of the Same File

When working on a large and complicated drawing, it is often useful to have simultaneous close-up views of more than one part of the drawing. Not only can you have multiple windows open on the same drawing, you can also start an operation in one window and finish in another.

For example: You may need to draw a line connecting two widely separated objects. Open a second window and tile the windows (**Window / Tile**). Adjust the zoom value in each window to get a close-up of each item. Then start the line in the first window, and finish it by clicking at the correct location in the second window.

Window Layout Templates

Default UI Menu: Window/Apply Layout Template, View/Open Multi View Layout, View/Save Multi View Layout

Ribbon UI Menu:

Enables you to customize the layout of multiple windows of the same drawing.

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Several predefined layouts can be selected from the bottom of the window. If you previously saved a template (using **Save Layout Template**), you can browse to select the layout template file.

The **Hide Caption**, **Auto Resize**, and **Auto Align** options can be used to customize some of the window layout properties. These options are also available on the **Window** menu. You can also start a new session with the last window layout from the last session. Check **Last Window Layout** in the **General** page of the **Program Setup (Options / General)**. See [General Setup](#)

Saving a Layout to a Template

Saves the current layout as a template you can use for other files. The files are saved with the *.twl extension.

Applying a Layout Template

Applies the last template used.

3. Drawing Aids

Drawing Aids

This section covers tools and features of TurboCAD that make your work easier and more organized, such as grid, snaps, layers, and construction geometry.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

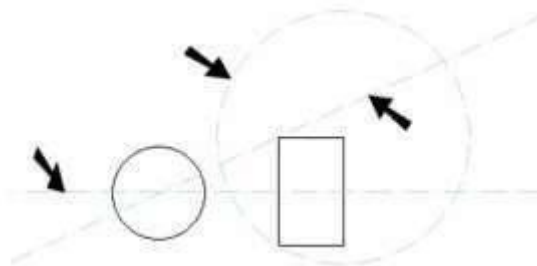
3.1. Construction Geometry

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Construction

Ribbon UI Menu:

Construction geometry tools enable you to place temporary lines and circles in your model. Construction geometry are not drawing objects; they are used as references.



Snap can be used on construction geometry for creating model geometry, as well as other construction geometry. However, because construction lines are infinite, the **Vertex** and **Middle Point** snaps are not available. You can display the **Construction** toolbar by right-clicking in any toolbar area and selecting **Construction**.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.

Note: You can use construction entities with the Trim tool to trim other drawing entities, however you cannot trim construction entities themselves.



Clearing and Hiding Constructions

Default UI Menu: Edit / Clear / Construction or Edit / Clear / All Constructions

Ribbon UI Menu:

You can clear construction objects via the **Edit** menu.

Edit / Clear / Construction enables you to delete a single construction object.

Edit / Clear / All Constructions deletes all construction geometry.

To hide constructions, you can hide the \$CONSTRUCTION layer. Open the **Layers** window and uncheck the visibility box for this layer.

Construction Geometry Properties

Construction objects are placed on layer "\$CONSTRUCTION". By default, the layer color is light blue, and the line style is dash-dot. You can change construction geometry color and line styles via the layer manager.

Note: It is not recommended to place model geometry on the \$CONSTRUCTION layer. If the color and line style of objects are set to **By Layer**, the objects will appear as construction geometry. It is also not recommended to change the line style to **Continuous**.

Creating Construction Geometry

Default UI Menu: Draw/Construction

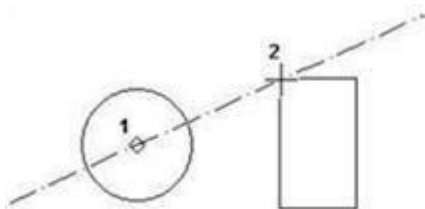
Ribbon UI Menu:

Several types of construction lines and circles are available.

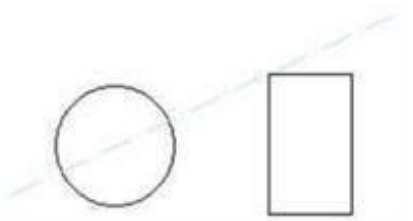
Construction-Angular Line



Select a point through which the construction line will pass. Select a second point, or enter the slant in the Inspector Bar.



The construction line is created, with the properties assigned to the construction layer.

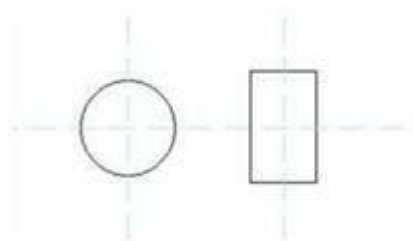


Construction-Horizontal Line, Vertical Line



Select the point through which the construction line will pass.

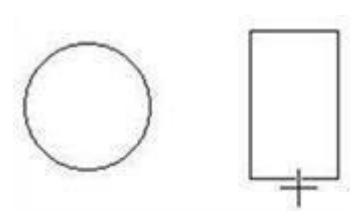
Continue creating construction lines of the same type, and select **Finish** when done.



Construction-Parallel

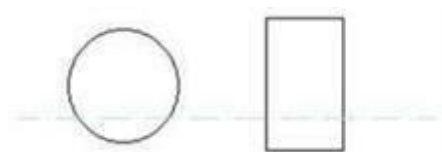


Select the line to which the construction line will be parallel. You can also select a construction line.



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Select a point through which the construction line will pass, or enter the offset in the Inspector Bar.

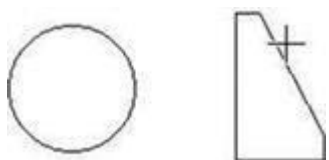


Tip: To make multiple construction lines separated by the same distance, lock the **Offset** field in the Inspector Bar, and create parallel lines from each successive line.

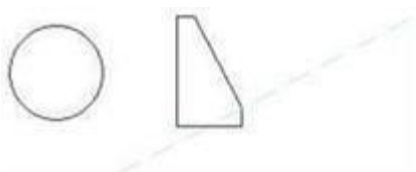
Construction-Perpendicular



Select the line to which the construction line will be perpendicular. You can also select a construction line.



2. Select a point through which the construction line will pass.

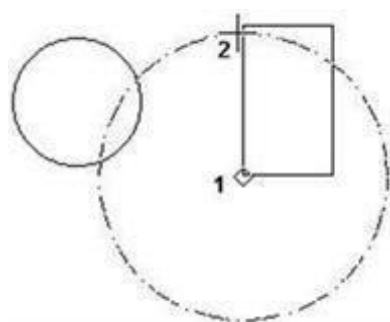


Construction-Center and Point Circle

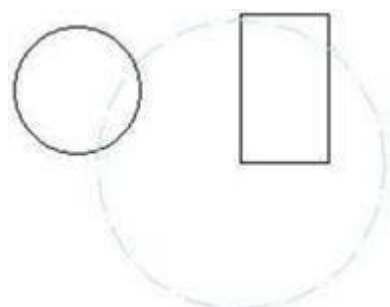


Creates a construction circle by defining its center and a point on its circumference.

1. Select the circle centerpoint. Move the cursor to begin to size the circle.



Click to create the circle, or enter the radius, diameter, or circumference in the Inspector Bar.

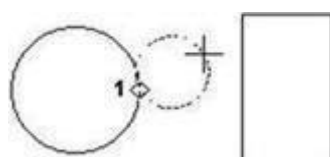


Construction-Double Point Circle

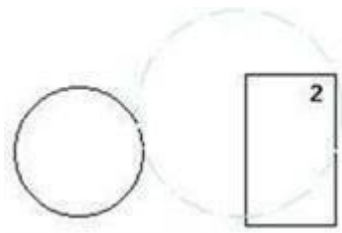


Creates a construction circle by defining two opposite points on its circumference.

1. Select a point on the circle circumference.



Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.

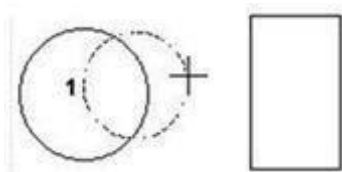


Construction-Triple Point Circle

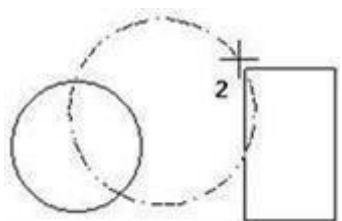


Creates a construction circle that passes through three points.

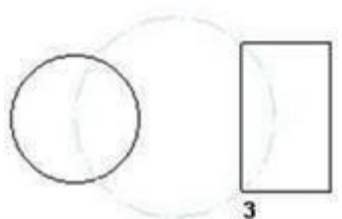
1. Select the first point on the circumference.



2. Select the second point.



3. Select the third point.



Construction-Perpendicular by Line



Creates a construction line perpendicular to a line defined by two points. Start with two circles, We will create a construction line halfway between the centers of the two circles perpendicular to the axis between the two centers.

1. Select the tool, then snap to the center of the first circle.

Move the cursor. Notice that the preview of the construction line is parallel to the line you are dragging out.

3. Snap to the center of the other circle to set the construction line.

Construction-Bisection



Creates a construction line which bisects an angle

Click on the first point. this will be the vertex of the angle to be bisected.

Click on the second point. this will be the first edge of the angle to be bisected.

Click on the third point. this will be the second edge of the angle to be bisected.

Move the cursor. Notice that the preview of the construction line is parallel to the line you are dragging out.

2. Snap to the center of the other circle to set the construction line.

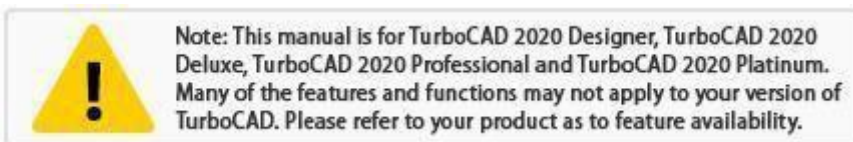
Editing Construction Lines

You can use the Edit tool to adjust the nodes of construction lines and rays. You can split a construction line into two rays by using the Split tool.

Printing Construction Geometry

While printing, you can choose whether or not construction geometry will be printed as part of your drawing. This option is set in the **Page Setup**. See [Page Setup](#)

Protractor



Default UI Menu: Draw/Construction/Protractor

Ribbon UI Menu:

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The protractor creates a construction line in any desired orientation in 3 dimensions. The protractor tool provides a visible protractor wheel to ease visual placement of the resulting construction line. To use the protractor:

First, select the tool located with the construction tools.

Second, position the protractor in the desired location then click in the drawing to place the center of the protractor, or specify a location via the Command line or Inspector bar.



Specify a base orientation for the protractor by:

- Clicking in the drawing to designate the base angle. or

- Specify a point via the Command line or Inspector bar to designate the base angle. or

- Specify an angle via the Command line or Inspector bar.

Specify the final angle for the construction line by:

- Clicking in the drawing to designate the base angle. or

- Specify a point via the Command line or Inspector bar to designate the base angle. or

- Specify an angle via the Command line or Inspector bar.

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Orienting the Protractor The protractor will automatically align to any 3D surface you hover over. Surface will highlight as you hover over them. Once you place the cursor location the resulting construction line will be oriented to the surface you have specified.

If the surface you wish to align to is located behind another surface you can use the Page Down and Page Up button on the keyboard to cycle through the available surfaces until the desired surface is highlighted.

Rays

Rays are a special type of construction line with only one end point extending infinitely in one direction.

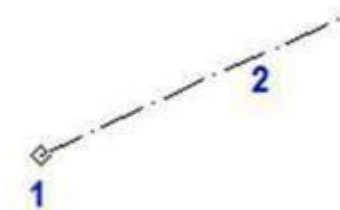
Angular Ray



Default UI Menu: Draw/Angular Ray

Ribbon UI Menu:

Select a point for the end of the ray will pass. Select a second point, or enter the



slant in the Inspector Bar.

The ray is created, with the properties assigned to the construction layer.

Horizontal Line, Vertical Line



Default UI Menu: Draw/Construction/Horizontal Line, Draw/Construction/Vertical Line

Ribbon UI Menu:

Select the point through for the end of the ray.

Continue creating ray lines of the same type, and select **Finish** when done.

Local menu options:

Flip: the Flip option changes the direction of the ray 180 degrees.

3.2. Coordinate Systems

Coordinate Systems

(Available in all TurboCAD Variants)

Default UI Menu: Modes/Coord System

Ribbon UI Menu:



There are several types of coordinate systems you can use, and you can switch between them at any time.

For example, when drawing the outer wall of a house, you may want to start the first wall at an absolute location. Each successive wall, however, will be defined by its length and angle relative to the first wall, so you would use polar coordinates for these points. To place walls at an X, Y distance from any other point, you could use relative coordinates.

You can display the **Coord System** toolbar by right-clicking in any toolbar area and selecting **Coord System**.



Coordinates, when entered manually, are entered in the Coordinate Fields, at the lower right corner of the screen. See [Coordinate Fields](#). You can press Shift+Tab to jump to the first field in the Coordinate Fields, then press Tab to scroll through the remaining fields.

Tip: If you precede a coordinate with a \$ sign, it will be interpreted as an absolute coordinate; if you precede it with an @ sign it will be interpreted as a relative coordinate; if you precede it with a > sign it will be interpreted as a polar coordinate.

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Coordinate systems behave the same way in 2D and in 3D, but in 3D you need to be familiar with the concept of workplanes as well. See [3D Coordinate Systems](#) and [Workplanes](#).

Absolute Coordinates

Default UI Menu: Modes/Coord System/Absolute Coords

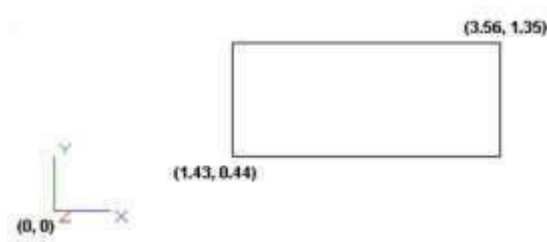
Ribbon UI Menu:



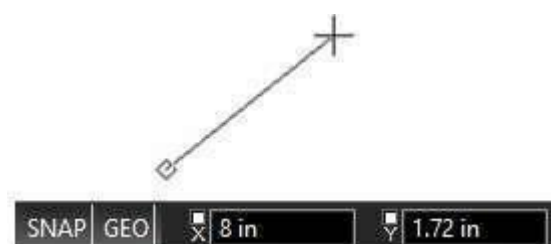
Hotkey: A, Shift+ A



Cartesian (X, Y) coordinates relative to the fixed, absolute (WCS) origin.



You can enter absolute X and Y values in the Coordinate Fields.



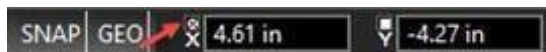
Locking Coordinates

By default, when you move the mouse around the screen, the values in the Coordinate Fields update dynamically. However, you can enter a value and lock it so that it will not change.

Locking coordinates comes in handy when you need to only lock one field. For example, you can lock an X value and constrain all future points to be placed on the vertical line that passes through the X point.

Warning: Be sure to unlock coordinates when you are finished. As long as a coordinate is locked, you are not free to select the usual range of points.

To lock a value, check the lock box for the relevant field.



You can also use the **Lock** options in the **Modes** menu, or use the X, Y, or Z hotkeys.

Polar Coordinates

Hotkey: P, Shift+ P



Default UI Menu: Modes/Coord System/Polar Coords

Ribbon UI Menu:

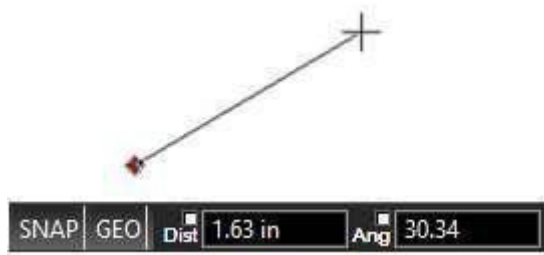


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Polar (length and angle) coordinates relative to a specified origin. The relative origin (moving or fixed) behaves the same way as for **Relative** coordinates.

Tip: If you precede a coordinate with a > sign it will be interpreted as a polar coordinate.

You can enter the distance and angle to the selected point in the Coordinate Fields.



Relative Coordinates

Hotkey: R, Shift+ R



Default UI Menu: Modes/Coord System/Relative Coords

Ribbon UI Menu:



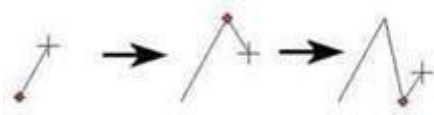
Cartesian (X, Y) coordinates relative to a specified origin.

Tip: If you precede a coordinate with an @ sign it will be interpreted as a relative coordinate.

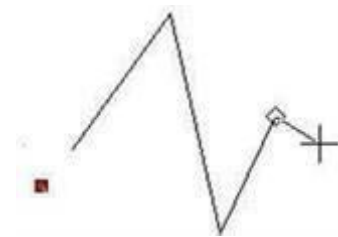
The relative origin can move as you work, relocating to each selected point, or it can remain fixed. The relative origin is controlled by the **Fixed relative origin** box on the

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Advanced Preferences page of the **Program Setup**. The relative origin is marked by a red square. You can use **Relocate Origin** (Shift+L) at any time to move it.

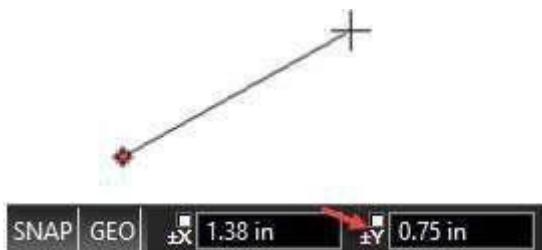


Relative Origin not f ixed



Relative Origin f ixed

You can enter relative X and Y values in the Coordinate Fields. Note the +/- symbols, indicating that the distance is not absolute.



Relocate Origin

Hotkey:L, Shift+ L



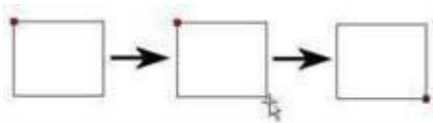
Default UI Menu: Modes/Coord System/Relocate Origin

Ribbon UI Menu:

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For Relative and Polar coordinates, moves the relative origin.

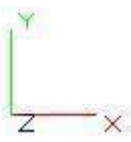


If you want to fix the relative origin in place, open the **Advanced Preferences** page and check **Fixed relative origin**.

WCS and UCS

The UCS (User Coordinate System) is the system in which you are working. By default, it is located to coincide with the WCS, but it can be moved. UCS axes can be displayed from **Program Setup**, and they appear wherever the origin is located.

The different coordinate systems in TurboCAD makes it easy to adjust to different drawing environments, for example if you draw round objects it may be easier to use polar coordinates.

The World Coordinate System (WCS) is the internal, high accuracy (10 digits) and absolute coordinate system. To display the WCS axes, select View/Display/UCS Icon/ World Coordinate System, and they will appear at the lower left corner of the screen. The WCS indicator does not represent the origin; it indicates orientation.	 WCS Indicator
The User Coordinate System (UCS) is the coordinate system in which you are working. By default, it is located to coincide with the WCS, but it can be moved and relocated by the user. UCS axes can be displayed by selecting View/Display/UCS Icon/ User Coordinate System, and they appear wherever the origin is located. Selecting an object using 2D Selector mode changes the UCS (and therefore the workplane) to the one attached to the selected object.	 UCS Indicator

In each coordinate system you also have an option to use either **relative** or **absolute** coordinates. The UCS indicator is helpful if you want to see how the UCS

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moves. You can display markers for the UCS and WCS via the Workspace menu, and customize the markers' appearance via the Preference page of the Program Setup (Options / Preference). The WCS indicator is always shown at the lower left corner of the screen; the UCS indicator is shown at the UCS origin.

When using the 3D Selector, the selection shell local menu and Inspector Bar provide two options that relate selected objects to the UCS (see [2D-3D Selector](#)). **Set UCS by Selector:** Moves the UCS origin to the selection reference point.



Set Selector by UCS: Moves the selection to the UCS origin (similar to the **Place on WorkPlane** option for 2D objects).



Note: The 2D Selector always moves the UCS (Workplane) to the selection.

3.3. Drawing Compare

(Available only in Platinum)

Default UI Menu: Tools/Drawing Compare

Ribbon UI Menu:

Drawing Compare is an easy to use utility which allows you to compare two drawings and find the differences between them. You can use 2CD, DWG, TCW, and BMP files. The process is simple, just load two drawings and press the Compare button. The Vector drawing files are converted to a raster image map and pixel testing detects changes between the two images.

Comparing Drawings

Comparing drawings is a simple process:

Use the **Select File 1** button or path to specify the first file for comparison.

Use the **Select File 2** button or path to specify the second file for comparison.

Press the **Compare** button.

If there are differences between the two drawings you will see the following message:

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Press the **OK** button. Your Drawings will now be loaded for comparison

Use the Zoom and Show buttons to examine your files and the differences.

Drawing Compare File Types

TCW: (TurboCAD Native Files): You can use any TCW file from v4 through v24.

DWG: (AutoCAD Native Files): You can use any DWG file from version R13 to version 2018.

BMP: You can use any standard Windows BMP file. However both BMP files must be the same size. If the images are of different sizes you will get an error message. This limitation applies only to raster images.

PNG: You can use any standard Windows PNG file. However both PNG files must be the same size. If the images are of different sizes you will get an error message. This limitation applies only to raster images.

JPG: You can use any standard Windows JPG file. However both JPG files must be the same size. If the images are of different sizes you will get an error message. This limitation applies only to raster images.

PDF: Adobe Portable Document Format

SKP: SketchUp file format.

Mouse Controls

You can use the mouse to control the display of the images in the following ways:

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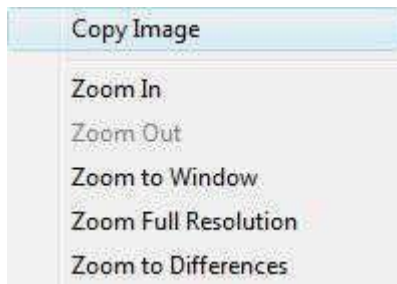
You can zoom in by rolling the mouse wheel forward.

You can zoom out by rolling the mouse wheel backward.

You can pan by holding the mouse wheel down and dragging.

You can pan by holding the left mouse button down and dragging.

Clicking the right mouse button will open a local menu which provides zoom commands and a Copy image command.



Copy Image

The Copy image command from the local menu allows you to copy the current image on the screen to the Windows clipboard. You can then paste that image into documents and other applications as a bitmap. This allows you to use the image in emails; Word files, CAD files etc. for use as notation, comment, illustration or documentation. The image will be captured at the full size of the image map specified by the Image Size setting in the Settings dialog. If the Image Size is set to 1000, the resulting image will be 1000x1000 pixels.

Other Controls



Settings: Opens the settings dialog.



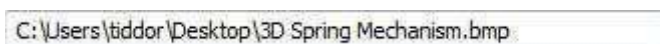
Select File 1: Starts the Open file dialog to select the first file for comparison.

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Select File 2: Starts the Open file dialog to select the second file for comparison.

You can also use the path fields to specify the location of your drawings:



Settings

The Settings dialog allows you to control how the bitmap image maps are generated for vector files.

Image Size: Allows you to specify the size in pixels that will be used for the generation of the image maps.

Change Background Color: Allows you to control the background color of the image maps. By default a white background is used when generating the image maps. If this option is checked the background color will be the color specified using **Select Color**.

Select Color: Open a standard windows color dialog that allows you to select the

desired color for the background of the image maps.

Show & Compare buttons



Show First File: Shows only the first file.



Show Second File: Shows only the second file.



Show the Differences: Shows only the differences between the two files.



Blink Differences: Blinks between the two files to illustrate the changes.



Compare: Compares the two files and generates the difference images.

The Interface

Zoom Buttons



Zoom to Window: Zooms out to show the full extent of the image.



Zoom to Full Resolution: Zooms in to show the image at 1:1 pixel resolution.



Zoom In: Zooms in one step. Maximum in is at 1:1 pixel resolution.



Zoom Out: Zooms out one step. Maximum out is at full extent of image.



Zoom to Differences: Zooms to show only the area where differences occur.

3.4. Grid

(Available in all TurboCAD Variants)

Default UI Menu: View/Display/Grid

Ribbon UI Menu:

The grid is a visual aid you can use when locating, sizing, and moving objects. The GSEKE (see [Running and Local Snaps](#)) and **Grid Snap** (see [Snap Modes](#)) enable you to use grid points to precisely locate objects.

Grid tools can be accessed from the menu, or from the flyout icon on the **Grid** toolbar:



You can display the **Grid** toolbar by right-clicking in any blank toolbar area and selecting **Grid**. The grid can be a pattern of lines, crosses, or dots that divides the drawing space into Cartesian or polar coordinates.

You can choose whether or not to display the grid, and you can customize how you want the grid to appear. If you display the rulers while working the grid can be a helpful indicator of orientation within the drawing. When working in 3D, the grid is displayed on the current workplane. For details, see [Workplanes](#).

Grid properties are set up in the **Grid** and **Advanced Grid** pages of the **Drawing Setup**. See [Grid Options](#) and [Advanced Grid Options](#)

Displaying and Manipulating the Grid

Default UI Menu: View/Display/Grid

Ribbon UI Menu:

The tools on the **Grid** toolbar and the **Grid** menu can be used to display the grid, size it, and change its origin.

Display Grid

Hotkey: Alt + G

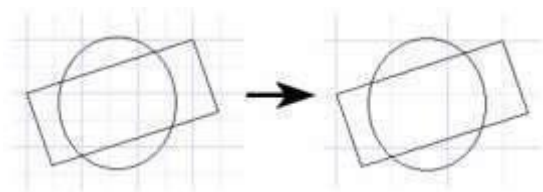


Displays or hides the grid. This command is a toggle function.

Double Grid



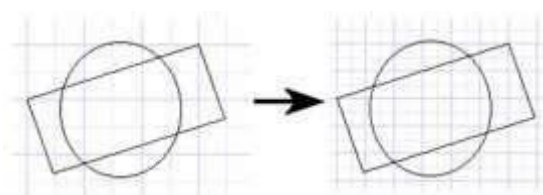
Doubles the spacing of the grid, making it less dense.



Halve Grid



Divides the spacing of the grid in two, making it more dense.



Set Grid Origin



Select the new origin for the grid, or enter coordinates in the Coordinate Fields. You can use Set Origin to align the grid to a particular point, then use the grid to set future distances.

Polar Grid



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It is a type of grid which takes a value for angle and radial which you can set in Grid in the Drawing Setup.

Iso Grid

Default UI Menu: View/Display/Grid

Ribbon UI Menu:

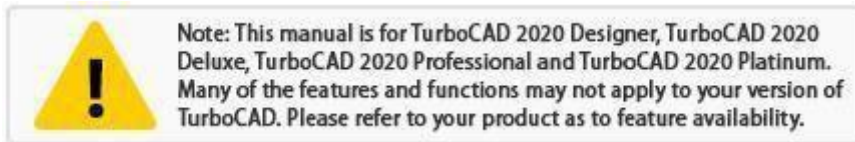
You can arrange the layout of the grid in Iso formation. There are three possible orientations. Top, Left, and Right.

Select the **Ortho grid** option or tool to reset the orientation of the grid to standard layout.

Printing the Grid

While printing, you can choose whether or not the grid will be printed as part of your drawing. This option is set in the **Page Setup**. See [Page Setup](#)

3.5. Hatch Pattern Creator



(Available in Platinum and Professional)

Default UI Menu: Tools/Create Hatch Patterns

Ribbon UI Menu:

TurboCAD's Hatch Pattern Creator (HPC) opens the door to a whole new world of customized hatch patterns. Use existing patterns, create your own patterns from scratch, or combine existing patterns with your new ideas and maximize the potential of both. Creating and editing hatch patterns has never been easier or faster.

The HPC a tool icon  will appear below TurboCAD's main toolbar and a menu item will appear in the Tools menu.

When you select either the button or the menu item the HPC will appear in the Inspector bar.



Clearing Patterns

It is possible to clear all of the item accumulated in the pattern on which you are working.

To clear accumulated patterns:

In the local menu or Inspector bar choose "Clear." All of the accumulated items will be removed.

Note: The tool has limitation on the number of created patterns (1000). You can change this value using corresponding field in the Inspector bar.

Creating Hatch Patterns

There are two primary ways of using the HPC: By Symbol, and By Sample

Creating Patterns - By Symbol

In the "By Symbol" mode you can select any drawing entities (flexible texts, polylines, arcs, curves, etc.) or their parts, using a fence, to create patterns. **To create pattern by symbol:**

Select the HPC.

Click once to set the first corner of the selection fence (rectangle).

Move the cursor to a new position, and click again to set the final point of the selection fence.

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The selection fence will appear as a red dashed line. The selected elements will appear highlighted in red, and the repeating pattern of those elements will be shown in light blue-grey.

Click Finish

Click Yes when prompted to add the pattern to the Brush Style Table

Type in a name for the pattern, and then click OK.

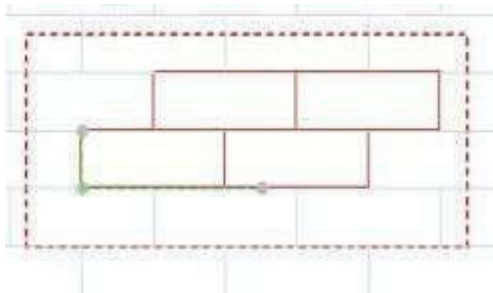
Adjusting the Pattern

Before clicking Finish you can alter the appearance of the pattern while you are creating it. By moving the blue control points (labeled 1& 2 above) you can change the frequency of reoccurrence and the direction of the pattern in those directions. Moving the other control point (labeled 3 above) allows you to change the origin of the pattern relative to the origin of the drawing.

Note: Notice that the horizontal line segment (above) was not selected as a part of the pattern. Line segments will be only be included in the pattern only if both ends of the segment are inside the selection fence.

Creating Patterns - By Sample

In the "By Sample" mode you use the selection fence to select a sample, prepared in advance, and designed to create a pattern. The Hatch Pattern Creator will automatically detect a valid repeating pattern. To be valid the pattern must repeat both vertically and horizontally within the rectangle of the selection fence.



NOTE: This mode does not provide the option to adjust the pattern with control points. To adjust the pattern you must switch to Transform mode.

To create pattern by sample:

Draw the pattern.

Select the HPC.

Click once to set the first corner of the selection fence (rectangle).

Move the cursor to a new position, and click again to set the final point of the selection fence.

The selection fence will appear as a red dashed line. The selected elements will appear highlighted in red, and the repeating pattern of those elements will be shown in light blue-grey.

Click Finish

Click Yes when prompted to add the pattern to the Brush Style Table

Type in a name for the pattern, and then click OK.

Editing Hatch Patterns

Using Existing Brush Styles

You can use existing hatch pattern type brush styles as the basis for your new pattern.

To select an existing pattern:

Select the HPC.

In the local menu or Inspector bar choose "Copy from..." to select existing brush style.

Select the Brush Style in the Select Brush Style dialog, and then click OK.

NOTE: In the "By symbol" mode the selected style will be copied as a new style. After that you can transform it. In the "By sample" mode a sample of the selected style will be used as a base for the new pattern. Later it can be used by user any number of times.

Accumulating Patterns

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When this mode is active, each newly selected symbol or sample, and each selected existing pattern will be added to the pattern you are creating. **To accumulate patterns:**

Select the HBC.

In the local menu or Inspector bar choose "Accumulate patterns" to toggle on pattern accumulation.

Select a pattern, by sample, by symbol, or an existing pattern.

Select another pattern, by sample, by symbol, or an existing pattern. Continue selecting patterns until you are done.

Click Finish

Click Yes when prompted to add the pattern to the Brush Style Table

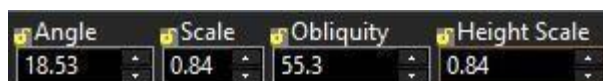
Type in a name for the pattern, and then click OK.

To turn off pattern accumulation:

In the local menu or Inspector bar choose "Accumulate patterns" to toggle off pattern accumulation.

Transforming Patterns

In transform mode you can transform the current pattern. Use the blue control points or the fields in the Inspector bar to transform the pattern.



The control point labeled "1" (above) is used to set the rotation angle and/or overall scale. The control point labeled "2" (above) is used to set the offset of the pattern's origin point from the drawing's origin.

The control point labeled "3" (above) is used to specify the scale for the pattern's height and (or) obliquity (angle) taking into account the rotation angle. At any time you can lock any of the fields in the edit bar.

NOTE: Transformations will be applied only to the last (current) accumulated item in the pattern.

Storing Created Patterns

At any time you can save created patterns as a new or existing style in the TurboCAD brush style table.

To save a pattern:

In the local menu or the Inspector bar choose "Save style...".

Type in a new name or select an existing name.

Click OK

3.6. Layer Management

(Available in all TurboCAD Variants)

Default UI Menu: Format/Layers

Ribbon UI Menu:

Layers

Layers are virtual levels in a drawing, analogous to acetate tabs (overlays) in traditional drafting. Layers enable you to sort objects in your drawing by type, by creation order, or by any criteria that suits the way you work. You can also use layers to protect certain objects from being edited or deleted.

Note: Layers are not related to how objects are stacked in relation to their order of creation. If you change an object's layer, it does not affect its position in the object stack.

Setting Up Layers

There are two primary ways of creating and editing layers. With regard to layers you can do essentially operations with either. Open the **Layer** manager by selecting **Format / Layers**, or by clicking the **Layers** icon on the toolbar.

This dialog is used to create layers, to assign properties to each layer, and to organize layers. It is divided into two frames. The first frame show the tree containing the layer

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filter, and layer templates. The second frame show the layers. controlled or derive from the item selected in the tree. The top node of the tree controls and shows all layers in the drawing.

Layer 0 is the default layer, and all objects are placed here unless otherwise specified, or unless another layer is created and made active. Layer \$CONSTRUCTION is created when construction geometry is created (see [Construction Geometry](#)). Layer 0, the default layer can not be deleted, but you can change its properties.

Layer Set: See [Layer Sets](#)

Columns in Layers window: Use the horizontal scroll bar to see all columns. You can sort the list by clicking on any column heading; the list will be sorted according to the selected parameter.

Visibility (eye icon): Check to make objects on the layer visible. **Read-only (lock icon):** Check to make the layer read-only, so that objects cannot be edited or deleted. You can add objects to a read-only layer via the **Layer** drop-down list located on the property toolbar.

Color: Click on the the color box to open the color dialog, then select a color - The default color is black. Objects will have the layer color if their color is set to **By Layer**.

Style: Select the line style from the drop-down list and double-click to set it. Objects will have the layer style if their style is set to **By Layer**.

Order: Determines the order in which layers are drawn. Order can be used to place certain categories of objects in front of others; objects drawn on higher layers will be "over" those on lower layers. The order numbers are initially set to zero. Layers that have the same order number are sorted alphanumerically. The highest layer number is 32767.

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Warning: The **Draw Order** commands will not function as you expect if objects are on different layers and the layers have different **Order** values.

Pen Width: Sets the line width. Objects will have the layer width if their width is set to **By Layer**.

Print Style: Specifies the Print Style to be used by objects on that layer [Print Style Options](#).

VP Columns in Layers window: In addition to the main columns in the Layers window there are five columns with a VP prefix. VP stands for Viewport, and these columns are used to control how objects will appear within a selected viewport.



The columns are: VP Visible, VP Color, VP Line Style, Pen Width, Print Style. The VP columns operate in the same way as the main columns, except the settings effect on selected viewports. To use the VP column settings:

- Open the Layer Manager.

- Go to Paper Space.

- Select the Viewports you wish to configure.

- Apply the settings in the VP columns of the Layer Manager.

Note: Remember that settings which control object appearance via the Layer Manager (e.g. VP Color) only affect the properties objects which are set to By Layer.

Layer Manager Toolbar

Refresh: Refresh the Layer manager display.

New Layer Filter: Creates a new Layer filter. Assign a name to the filter then setup the filter in the Layer Filters dialog.

Edit Filter Parameters: Open the layer filter currently selected in the tree in the Layer

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Filters dialog.

Edit Layer Sets: Opens the Layer Set dialog to create and edit layer sets.

New Layer Template: Creates a new layer template from the currently selected layers.

New Layer: Creates a new layer. The default name will include a prefix by default, but the name can be changed.

Delete Layer: Deletes the layer.

Activate: Sets the currently selected layer (only one) as the active layer. This will effect any currently active drawing tool or currently selected object. **Select By:** Selects all objects on the currently selected layers.

Edit Properties: Opens that layer page of the Drawing Options.

Visible/Invisible All: If any layer is invisible all layers are made visible. If all layers are visible they are all made invisible.

Invert Visibility: Turns all visible layers invisible, and all invisible layers visible.

Exclusive Visible: Makes only the currently selected layers visible, all other become visible.

Lock/Unlock All: If any layer is unlocked all layers are locked. If all layers are locked they are all unlocked.

Invert Lock: Exclusive Lock: Makes only the currently selected layers locked, all other become unlocked.

Layer Prefix: By default, the layers are named "Layer 1, Layer 2," etc. You can change or remove the prefix. The @ symbol is a placeholder for the automatic layer number.

Note: Many of the functions available from the Layer toolbar are also available by right clicking and opening the local menu.

Creating a New Layer

Open the **Layer** manager by selecting **Format / Layers**, or by clicking the **Layers** icon on the toolbar. A dialog opens. In the dialog, right click on Layer and select new layer. Then assign a name for the layer in the **Layer** column (or accept the default name).

Adjust the various layer settings, such as color and line style.

Deleting a Layer

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You may delete any layer except Layers 0. Layers can be deleted even if they contain objects. If the layer to be deleted is set as the default for a tool (in the **General** page of a tool's **Properties** window), you will receive a warning message before the layer is deleted.

Select **Layers**, and select the layer to be deleted.

Click **Delete Layer**. If the layer contains objects, the objects will be deleted. This action can be undone, in case you delete objects inadvertently.

In some cases, objects on deleted layers will be moved to Layer 0, rather than be deleted. If an object exists on Layer 1 in both Paper Space and Model Space, and Layer 1 is deleted from Model Space, in Paper Space the object will be moved to Layer 0. This is due to different **Undo** buffers for Model and Paper Spaces.

Deleting Construction and Constraints

Layer:

(Available in Platinum and Professional)

Construction objects are placed on layer "\$CONSTRUCTION". You can change construction geometry color and line styles via the layer manager. You can now also delete \$Construction layer and \$Constraints layer by selecting them in the Layer palette (see [Design Director-Layers](#)).

Demo Video:

<https://youtu.be/HHohlGQX8rw>

Layer Templates

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Layer templates allow you to create and save alternate configurations for layers. Layer templates store how layers are setup, but do not store layers themselves. Layer templates are saved in a *.lrs file which can be stored anywhere in your system directory.

Layer Sets

Default UI Menu: Format/Layers

Ribbon UI Menu:

A layer set is a group of layers which can be displayed as a group. This is useful for displaying certain aspects of a drawing without changing visibility settings of each layer individually.

The default layer set is "All Layers," which appears in the **Format** menu.

Creating and Manipulating Layer Sets

In the Layers window (Format / Layers), click Edit Layer Sets.

When the **Layer Set** dialog opens select **New**.

Assign a name to the set, or accept the default name. The name appears on the **Layer Set** list.

On the list of layers, check the visibility of each layer you want to include in the layer set.

5. To display a layer set, open the **Format** menu.

Note: While a layer set is displayed, the properties of each layer are not editable.

To delete a layer set, select it from the **Layer Set** list and click **Delete**.

To change the layers that appear in a layer set, select it from the **Layer Set** list and change the visibility settings.

Layers of Groups and Blocks

If you create a group composed of objects on the same layer of the drawing, the group will reside on that layer. For objects on different layers, the group will be on Layer 0. If you subsequently explode the group, its constituent objects will return to its original layer.

Objects should be moved to Layer 0 before being used to create a block. The block itself, when inserted in the drawing, can be reassigned to the proper layer. Any objects within the block that have a **By Layer** property will take on the assigned layers attributes. When a block is exploded, all objects originally assigned to Layer 0 will stay on the layer the block was on. Any objects with other layer assignments will retain those layers.

Layer Sorting

You can use the Layer Manager or the Design Director to sort layers into a desired order. See [Design Director-Layers](#) and [Layer Management](#)

To sort the layer click the icon at the top of any column e.g. name.

Click that Icon again will reverse the order of layer by that category.

When layers are sorted the order of layers will be the same throughout the application. this means that the order of layer shown in drop-down boxes will be the same as that in the Layer Manager or Design Director.

Manipulating Layers and Properties

By default, new objects are created on Layer 0, unless a layer is specified in the **Properties** window of a specific tool. You can change the layer of objects after they are created, or you can change the default layer of a group of tools.

Setting the Active Layer

Set the layer you want to be active by selecting it from the **Layer** list in the **TurboCAD Layers** toolbar. You must have a drawing tool selected or an object selected to change the active layer. Setting the Layer of Selected Objects or a Group of Tools To quickly change the layer of selected objects:

- Select the objects.

- Select the layer from the **Layers** list box in the **TurboCAD Layers** toolbar.

To set layers via the Properties window:

- Select the objects and open the **Properties** window. To set the layer for all objects created with a certain group of tools, right-click on one of the tool icons and open the **Properties** window (see [Object Properties](#)).

- On the **General** page, select the layer from the drop-down list.

If the layer is set for selected objects, the objects will move to that layer. Future objects, however, will still be created on the tool's default layer. If the layer is set for a group of tools, all objects created with any one of the tools will automatically be placed on the selected layer. For example, setting the layer for the **Line** tool also affects the **Rectangle** and **Polygon** tools.

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Note: You can also select the objects and change the layer on the Selection Info Palette. See [Selection Info Palette](#)

Changing the Layer Visibility via Dropdown

You can change the visibility of a layer using the Layer dropdown by clicking on the eye icon adjacent to the layer in the dropdown.

Changing the Layer Color via Dropdown

You can change the color of a layer using the Layer dropdown by clicking on the color icon adjacent to the layer in the dropdown.

Changing the Layer Lock via Dropdown

You can change the lock status of a layer using the Layer dropdown by clicking on the lock icon adjacent to the layer in the dropdown.

Assigning Layer Properties to Objects

Open the **Properties** window, either for selected objects or for a group of tools.

See [Object Properties](#)

Open the **Pen** page.

The properties that can be set to **By Layer** are color, line style, and width. Set any or all of these properties as needed.

Note: **By Layer** is also available for properties on the **TurboCAD Properties** toolbar.

3.7. Page Layout Wizard

(Available in Platinum)

Default UI Menu: Tools/Wizards/Page Layout Wizard

Ribbon UI Menu:

The Page Layout Wizard is designed to expedite the creation of Paper Space Layouts for 3D models and projects.

Options:

Auto: Everything is automatic, except the selection of the pre-existing project.

New: Create an entirely new project.

Use: Use an existing project with or without modifications. When the Use option is

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selected you will be prompted to specify the *.PLX file that contains the desired project.

Modify: Modify an existing project without using it.

Use margins information from defined printer: Use the margin settings from the selected printer to define the margins of the generated Paper Space Layouts. This option is not available in the Modify mode.

Create page layouts for all 3D entities in the drawing: This option will only be available if the Use mode is selected. With this option ALL 3D entities in the drawing will be used for creation of the View/Viewports and Drafting palette components. If you do not select this option you will be prompted to select the required 3D entities in the drawing space.

For optimal efficiency and performance two modes are provided for the creation of layout components:

Architectural: This mode is faster with less demands on system resources, but non-associative

Mechanical: This mode is associative, but is slower, and places greater demands on

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system resources. Note: You can use the Architectural mode for mechanical models and vice-versa but there is a potential for reduced

Options:

Create page layouts for all 3D entities in the drawing - With this option ALL 3D entities in the drawing will be used for creation of View/Viewports and Drafting palette components.

If you do not select this option you will be prompted to select the required 3D entities in the drawing space.

To select the items:

- Hold down the Shift key.

- Sequentially click on each of the items you require.

- Right click the mouse and choose Finish.

Layout Project Editor

The Layout Project editor is where you build your pages and layouts. Your options will vary depending upon whether you selected Mechanical or Architectural as the mode.

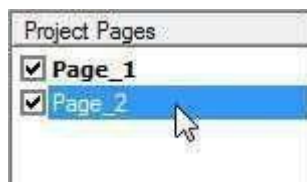
Architectural

Mechanical

Pages

The Project Pages window shows all of the pages in your current project. The currently visible page is shown in bold.

To view a particular page do a fast double click adjacent to the name of that page.



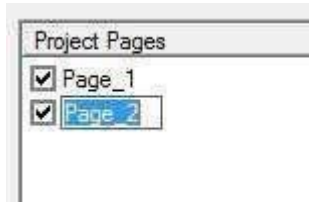
To create a new page, click the Add new page button below the Project Pages window.



To Re-name a page do a slow double click on the name of the

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page. Then type in the new name and finally press Enter.



Options

If you right-click on a page in the Project Pages window you will be presented with three options:



Delete Page: Removes the page on which you right-clicked.

Reset Page: Resets the page on which you right-clicked so that it is blank.

Clear Project: Deletes all pages except page 1 and resets page 1

Saving the Project

To save a project for future use click the Save Project As button. Projects are saved as *.PLX files.

Layout Page Area

The Layout Page area is where you compose layout for each of the pages in your project. This is accomplished by dragging & dropping views and sections from the top window into the Layout Page area. For example, if you drag a view onto the left side of the Layout Page area:

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The result would be as follows:

Another example, if you drag a view onto the center of the c:

The result is as follows:

The following diagram shows you all of the

Options

Set equal scales for all Layout areas - If the checkbox is off, scales for different size areas will vary. If the checkbox is on the same MINIMAL scale will be applied to all areas. For example if you have one large area and three identical small areas the results would be as follows: If OFF, the large area would have Scale 1 and the small areas Scale 2; If ON, all four areas will have Scale 2.

Set standard scales: When this option is turned on the scale for the views will be

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rounded off from the automatic fit scale to the closest rounded equivalent. (1:1.23234 -- > 1:1.25 etc...).

Insert View/Section name: The name of the view or section is inserted into the resulting layout as text.

Selecting a View/Section

To select a view or section in the Layout Page area, click on it.

Deleting a Layout Area

To delete a section or view, select the target view or section in the Layout Page area, then right-click over it. From the popup menu select Delete Layout Area.

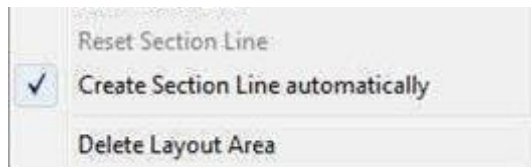


Sections – Architectural Mode

In architectural mode sections can be automatically derived or custom. By default all section are automatic.

\

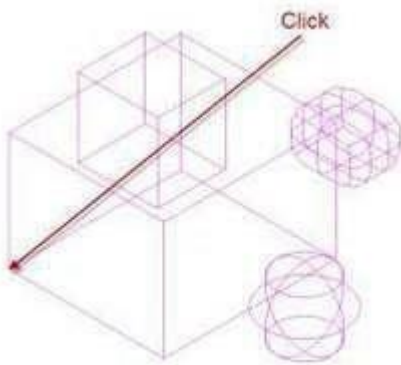
To create a custom section line, select a section in the Layout Page area, then right-click over that section. From the popup menu select Set Selection line.



The view will switch from the wizard to Model space. You need to see the space in 3 - dimensions clearly so right-click and select an isometric view. To set the selection line you will first be prompted with a dashed line.

1. Position the mouse, and then click to set the dashed line as the section line.

The next prompt is a box. Position the mouse and click again to set the depth and width of the section area.

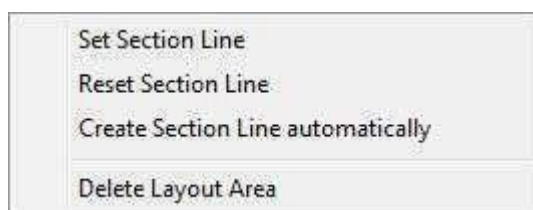


After the second click the depth and width of the section box will be set. Now position the mouse and click a third time to set the height of the section.

When you have selected the third position the section will be set and the wizard will reappear.

Resetting a Section

To reset a section, select that section in the Page Layout area, and then right -click over the section in the Layout Page area. Choose Reset Section Line from the popup menu.

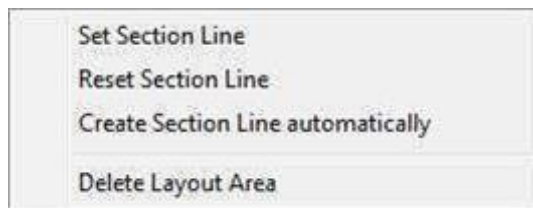


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A warning will appear on the section saying that the Section Line is not defined. Use the Set Section Line option to set the section correctly.

Automatic Section Line

To reset any custom section to be automatic, select that section in the Page Layout area, and then right-click over the section in the Layout Page area. Choose Create section Line automatically from the popup menu.



Paper Space Properties

To set the properties for any page in the project first select that page in the Project Pages window, and then click the Paper Space properties button. A paper Space Properties dialog will appear. From there you set the properties for the desired paper space in the normal fashion. Click OK when you are finished.

3.8. PDF Underlays

(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Tools/Pdf tools/Pdf underlay to Objects, Insert/Underlay

Ribbon UI Menu:

The PDF Underlay tool allows you to insert a page of a PDF file into your drawing to use as reference or tracing layer. To use properly as a tracing layer, the file must be a vector PDF, rather than a bit-mapped PDF. As a vector file, you will be able to snap to the endpoints and midpoints of lines in the PDF underlay.

Underlays consist of two parts: the underlay style and the underlay insertion. Before inserting an underlay you must first create an Underlay style, which is done through the Underlay Manager, found in the Tools menu of the Default interface, and in the Format menu of the Ribbon interface

A new tool "PDF underlay to Objects" has been implemented. It gives the ability to convert an Underlay into an editable object.

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In addition to importing a PDF Underlay as an Object, 2D Vector PDFs can now be directly imported as a vector file into TurboCAD. To learn more about this please see [Import PDF as Vector](#)

Demo Video:

<https://youtu.be/7kyLXxekCN8>

Creating an Underlay Style

Default UI Menu: Tools/Underlay Manager

Ribbon UI Menu:

Open the Underlay Manager.

It contains a list of underlay styles currently defined for the drawing.

1. Click the **New** button to create a new Underlay Style. The Underlay dialog opens.
2. Click **Browse** button to display File Open dialog and choose a pdf file.

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Now you can see the list of pages of the selected PDF document and a preview picture for the selected page. By default ,the page name is used as the Name of the Underlay style but you can change it.

Click OK to close the Underlay dialog. Now you will see the new Underlay Style is added to the Styles List.

5. Click OK to close Underlay Manager dialog

Inserting an Underlay in a Drawing

Default UI Menu: Insert/Underlay

Ribbon UI Menu:

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To insert a PDF Underlay:

Select the Underlay tool.

You will be prompted to select the desired style. Select the Underlay Style you want and then click OK.

Click to define left bottom corner of the underlay insertion.

Move the cursor to the desired location and click again to define right top corner of the underlay insertion.

Import PDF as Vector

Import PDF as Vector

(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Tools/Pdf tools/Import pdf page as vector graphics

Ribbon UI Menu:

2D Vector PDF files can now be directly imported into TurboCAD as a vector page. The file can be imported as either a TurboCAD Object or Group of Graphics. Please note, that if the PDF file contains multiple pages, only one page can be imported at a time. To import, follow these steps:

Click on the Import PDF as Vector tool and then select the PDF file and see all its pages in the preview window.

Select the desired page you wish to import.

Click in the drawing to set the lower left point then drag out to the desired size.

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Demo Video:

<https://youtu.be/MClz6VPry0E>

3.9. Snaps

(Available in all TurboCAD Variants)

Default UI Menu: Modes/Snaps

Ribbon UI Menu:

Snaps ensure precise placement of points when you use the cursor. For example, rather than enter coordinates or length to define the endpoint of a line, you can snap to the endpoint or midpoint of an existing line. This Snap toolbar displays and controls running snaps. Local snaps can be used regardless of what running snaps are active.

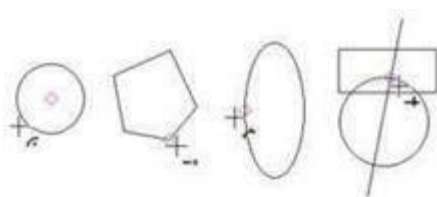
Magnetic Point



Default UI Menu: Modes/Snaps/Show Magnetic Point

Ribbon UI Menu:

The magnetic point is the indicator of the snap location. The disadvantage to showing the magnetic point is that for very large files or very slow computers, the magnetic points might take time to display.



Running and Local Snaps

Snaps can be permanent (until turned off) or can be temporary (used just once).

Running snaps are permanent, and those invoked by the local menu and SEKE's are temporary.

Running Snaps

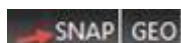
Running snaps are snaps that are constantly active, as opposed to one-time snaps invoked by SEKEs and the local menu. You can turn on running snaps by:

Selecting items from the **Snaps** menu.

Clicking the icon or combination of icons on the **Snap Modes** toolbar.



Right click on the snap button next to the Coordinate Fields, and check the desired snaps.



Use hotkeys. You can view these in the **Snaps** menu. For example, Shift+V turns on **Vertex** snap.

Note: Be careful with the snap modes you leave running. For example, if only **Vertex** is in use, you will not be able to select any point besides a vertex.

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You can temporarily turn off all running snaps by clicking the SNAP button next to the Coordinate Fields. While the button is grayed out, the running snaps are disabled. Click again to activate the snaps.

The GEO button performs the same function for geometric aids (Extended Ortho, Apparent Intersection, and Show Magnetic Point).

It is usually best to turn off both SNAP and GEO at the same time.

Local Snaps

If you want to use a certain snap once, but do not want to have the particular snap mode constantly in use, you can use local snaps. These are one-time snaps that you can select from the **Local Snap** item of the local menu. You can also use SEKE's (hotkeys) to invoke one-time snaps.

SEKE's

Single Equivalent Keyboard Entries are keyboard shortcuts, or hotkeys, for quickly performing a function or snapping. SEKE's are affected by the snap aperture settings. If the sought snap point or type is not available within the aperture and **Show snap aperture** is turned on the SEKE will not snap.

To use a SEKE for snapping, place the cursor on the desired object and press the key. For example, place the cursor anywhere on a line and press M to snap to its midpoint. The following is the list of SEKE snaps:

C - Center (snaps to the center of an arc, circle, or ellipse) G - Grid (snaps to the nearest grid point). H - Perpendicular Opposite (snaps to an endpoint so that the new line is bisected by an existing line) I - Intersection (snaps to the intersection of two objects) J - Perpendicular (snaps to the perpendicular projection to an object) M - Midpoint N - Nearest on Graphic (snaps to the nearest place touched by the cursor) Q - Quadrant Point (snaps to 0, 90, 180, 270 degree point of a circle, arc, ellipse) S - No Snap V - Vertex (snaps to endpoints of lines and arcs)

In the LTE Workspace you must use the ALT key simultaneously with SEKE. I.e ALT+N

Snap Modes

Default UI Menu: Modes/Snaps

Ribbon UI Menu:

This section lists all available snaps and geometric aids. When accessed from the **Snap Modes** toolbar or from the menu, these are running snaps (permanent until turned off). When accessed by a SEKE or from the local menu, they are one-time snaps.

LTE Snaps

In the LTE Workspace SEKEs behave with slight differences to standard TurboCAD methods. To use a SEKE you must hold down the ALT key. E.g. the Center SEKE becomes ALT+C while in the LTE Workspace.

Apparent Intersection Snap



Default UI Menu: Modes/Snaps/Apparent Intersection

Ribbon UI Menu:

Snap to points where two lines would meet. An auxiliary line is displayed from both lines, extending to this point. **Show Magnetic Point** must be turned on.

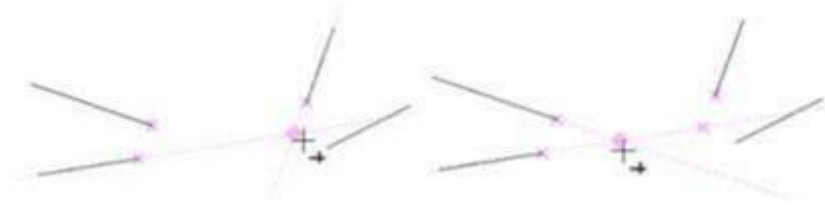
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Start with lines whose extensions will meet (non-parallel). When you pass the cursor



over each line, the line's endpoint is marked.

When the cursor passes over any apparent intersection point, auxiliary lines will be displayed from the actual lines, and their intersection point is marked.



Note: If **Always show auxiliary lines** is checked in the **Drawing Aids** window, the auxiliary lines will be displayed, even when you move the cursor away from them.

Apparent Intersection works here on drafting palette objects.

Center of Extents Snap

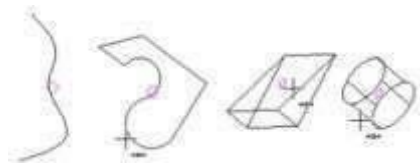
SEKE: E



Default UI Menu: Modes/Snaps/Centre of Extents

Ribbon UI Menu:

Snap to the center of extents of a 2D or 3D object. This is the center of the bounding rectangle (2D) or box (3D) that encloses the selected object.



Center Snap

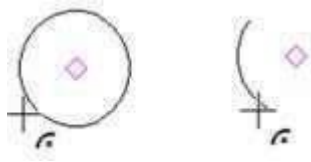
SEKE: C



Default UI Menu: Modes/Snaps/Centre

Ribbon UI Menu:

Snaps to the center of an arc, circle, or ellipse.



Divide Point Snap



Default UI Menu: Modes/Snaps/Divide Point

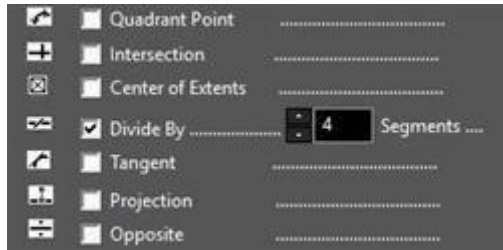
Ribbon UI Menu:

Snaps to one of the division points of an arc or line segment.



The default number of segments is 3, but you can change this in the **Segments** field of the **Drawing Aids** window. Open this window by right-clicking on the SNAP or GEO button next to the Coordinate Fields.





Tip: You can also divide arcs and line segments while using the **Edit Tool**, by selecting **Divide Segment** or **Arc Divide** from the local menu. See [Edit Tool](#)

Extended Ortho Snap

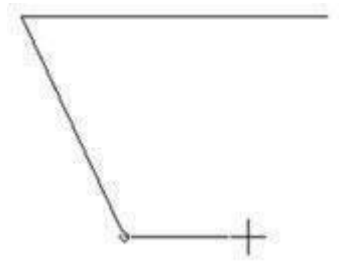


Default UI Menu: Modes/Snaps/Extended Ortho

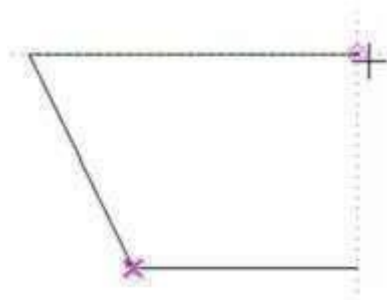
Ribbon UI Menu:

Snap to points located orthogonal from existing points. **Show Magnetic Point** must be turned on.

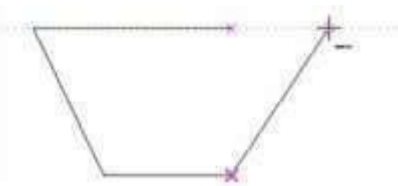
1. Start a polyline with two linear segments, then start the third segment like this:



To end the third segment, first hover over the start point. This displays an auxiliary line extending vertically from that point. You can snap to points along this line.



Horizontal auxiliary lines can also extend from this point, when the cursor is to the right or left.



Note: If **Always show auxiliary lines** is checked in the **Drawing Aids** window, the auxiliary lines will be displayed, even when you move the cursor away from them.

Grid Snap

SEKE: G



Default UI Menu: Modes/Snaps/Grid

Ribbon UI Menu:

Snaps to the nearest grid point. See [Grid](#). For details on grid settings, see [Grid Options](#). If the **Advanced Grid** option of **Frequency** is set to a figure greater than one, invisible grid lines will also be detected by this snap mode. See [Advanced Grid Options](#)

Intersection Snap

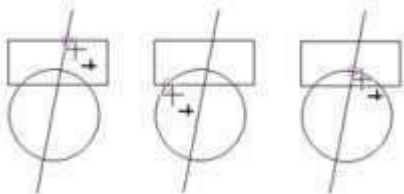
SEKE: I



Default UI Menu: Modes/Snaps/Intersection

Ribbon UI Menu:

Snaps to the intersection of two objects.



Middle Point Snap

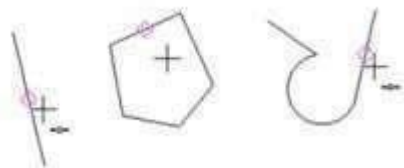
SEKE: M



Default UI Menu: Modes/Snaps/Middle Point

Ribbon UI Menu:

Snap to the midpoint of a line or line segment.



Nearest on Facet Snap



Default UI Menu: Modes/Snaps/Nearest On Facet

Ribbon UI Menu:

Snap to the nearest point on a facet, or to the projection of this point onto the current workplane.

Nearest on Graphic Snap

SEKE: N



Default UI Menu: Modes/Snaps/Nearest On Graphic

Ribbon UI Menu:

Snap to the point on an object closest to the cursor, within the snap aperture.

No Snap

SEKE: S



Default UI Menu: Modes/Snaps/No Snap

Ribbon UI Menu:

When on, turns off all snap modes. With snap modes off, you define points by simply clicking, or by using the Inspector Bar and Coordinate Fields.

Opposite (Mirror Point) Snap

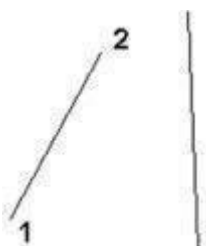
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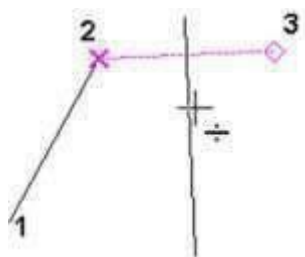
Default UI Menu: Modes/Snaps/Opposite

Ribbon UI Menu:

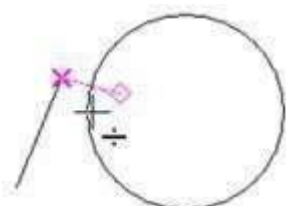
Snaps to a point that is the mirror image of the last point drawn, in reference to a selected object. In this example, the line on the right is the mirror line. The last point drawn is Point 2, when creating Line 1-2.



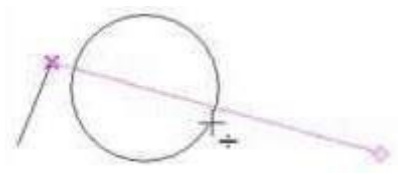
Activate the **Opposite** snap and hover over the mirror line. Point 3 is the snap point.



In some cases it is important where you select the mirror object. If the circle is selected on the left side, the mirror point will be opposite that side.



If you select the circle on its right side, the mirror point will be opposite that side.



Ortho Snap

Hotkey: Shift + O



Default UI Menu: Modes/Snaps/Ortho

Ribbon UI Menu:

When in use, the cursor can only move horizontally or vertically.

Projection Snap

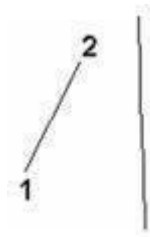


Default UI Menu: Modes/Snaps/Projection

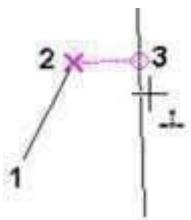
Ribbon UI Menu:

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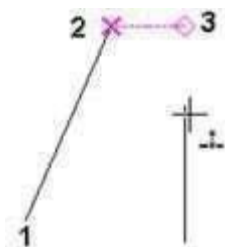
Snap to a point that is the perpendicular projection image of the last point drawn onto a selected object. In this example, the line on the right is the projection line. The last point drawn is Point 2, when creating Line 1-2.



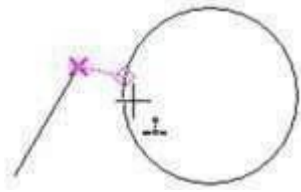
Activate the **Projection** snap and hover over the projection line. Point 3 is the snap point, which is the perpendicular projection of Point 2.



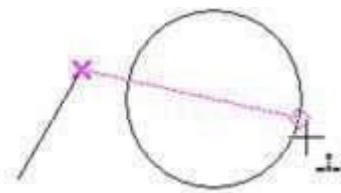
The snap point does not have to lie on the projection line; it can be placed on the extension of the projection line.



In the case of a circle, the projection point is along the radial from the circle center. If the circle is selected on the left side, the snap point will be on that side.



If you select the circle on its right side, the snap point will be on that side.



Quadrant Point Snap

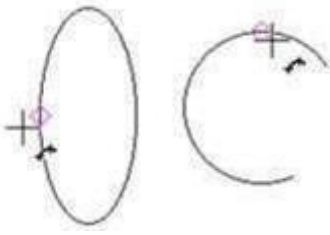
SEKE: Q



Default UI Menu: Modes/Snaps/Quadrant Point

Ribbon UI Menu:

Snap to the nearest quadrant point (0, 90, 180, and 270 degrees) on a circle, arc, or ellipse.



Show Magnetic Point



Default UI Menu: Modes/Snaps/Show Magnetic Point

Ribbon UI Menu:

When in use, a diamond-shaped point will be displayed for the locations of all running snaps.

Tangent Snap

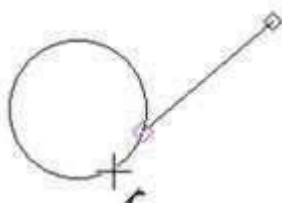
SEKE: T



Default UI Menu: Modes/Snaps/Tangent

Ribbon UI Menu:

Snap to a tangent point on an arc, circle, or ellipse, relative to the previous point.



Vertex Snap

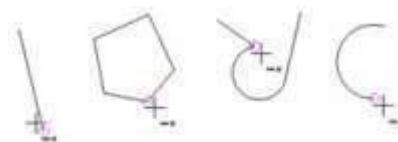
SEKE: V



Default UI Menu: Modes/Snaps/Vertex

Ribbon UI Menu:

Snap to the nearest vertex. A vertex can be an endpoint of a line or line segment,



corner of a polygon, or endpoint of an arc or curve.

LTE Snaps

In the LTE Workspace Snaps and SEKEs behave with slight differences to standard TurboCAD methods. In the LTE Workspace the Vertex snap is called Endpoint.

To use a SEKE you must hold down the ALT key. E.g. the Vertex snap becomes ALT+V

while in the LTE Workspace.

Workplane Intersections Snap

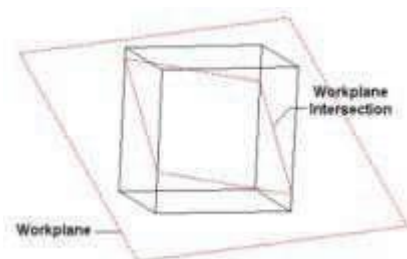


Default UI Menu: Modes/Snaps/Use Workplane Intersections

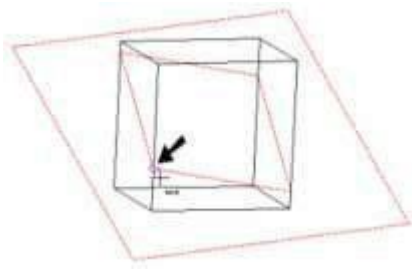
Ribbon UI Menu:

Snaps to intersection curves between 3D objects and the current workplane. (For information on workplanes, see [Workplanes](#))

The workplane must be displayed (see [Displaying the Workplane](#)), and intersection lines must be displayed as well (see [Display Intersections with 3D Objects](#)).



Activate the **Workplane Intersections snap** along with the snaps you want to use on the intersection lines and curves. In this case **Vertex** snap is active, and you can snap to an endpoint of the intersection line.



3D Snap – Snap to Workplane:

Snap to WorkPlane Intersections also works with 2D objects (arcs, curves, polylines) that do not lie both lie on the XY plane of the current workplane/UCS.

Mid 2 Points Snap



Default UI Menu: Modes/Mid 2 Points Snap

Ribbon UI Menu:

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Snap mid two points that once chosen allows you to have two subsequent points and will then snap to the middle of them.

1. Enable Mid two point snap and select two points
2. Mid two point will snap to middle of two selected point s.

Demo Video:

<https://youtu.be/vm3QboOOtl>

Snap Settings

The **Drawing Aids** window contains options for setting snaps and their priority levels. Open this window by right-clicking on the SNAP or GEO button next to the Coordinate Fields.



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Modes

No Snap: No snaps will be permanently active (running).

Ortho Mode: The mouse will move only horizontally and vertically.

Workplane by Face mode: The workplane will be set by the facet of the solid where your cursor is. See [Workplanes](#)

Use Workplane Intersection: Enables you to snap to intersection lines and curves between a 3D object and the current workplane. See [Snap Modes](#) . Show Magnetic Point and auxiliary lines will also appear.

Always show auxiliary lines: Once displayed, auxiliary lines will remain onscreen, even when the cursor moves outside the snap aperture.

Use Mouse Position if Required Snaps are not Found: If you click on a point at which there is no snap, the point will be used anyway.

Running Snaps: These are snaps that are permanently active, as opposed to snaps that are used only once.

In Aperture Only: The associated snap will function only when the snap point is within the snap aperture. If more than one object is located within the snap aperture, the point will be defined at the closest qualifying snap point, unless **Priority** is set. If not checked, the snap point itself does not need to be located within the snap aperture. You can set the radius of the snap aperture in the **General** page of the **Program Setup (Options / General)**.

Show Hit Graphic: This will highlight edges for snaps when you move a 3D object or connect two, 3D objects together. Settings can be changed from Program Setup/Preferences window.

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Tip: To see the snap aperture while you are working, set it in the **Preference** page of **Options**.

Priority: Establishes which snap take precedence if more than one snap is available. A value of 1 has the highest priority.

3.10. The Design Director

(Available in Platinum, Professional, Deluxe)

Hotkey: F3

Default UI Menu: Tools/Palettes/Design Director

Ribbon UI Menu:

The Design Director is a convenient place to perform commands related to layers,

workplanes, views, cameras, and graphic objects or object groups.

Design Director-Cameras

Default UI Menu: Tools/Palettes/Design Director/Camera

Ribbon UI Menu:

Enables you to manipulate all the cameras and camera sets in the drawing (see [Camera Objects](#)).

To create a new camera or camera set, select **Create New** in the Design Director toolbar or local menu. When **Cameras** or **Camera Sets** is selected in the upper pane of the Design Director, the lower pane has the following options:

Active: Displays the view taken by the camera, and attaches the window to the camera.

Visible: Displays or hides the camera symbol in the drawing.

Attached: Creates a new window associated and synchronized with the camera. **Draft**

Rendering: Sets the rendering type when the view is rendered in **Draft** rendering mode. See [Creating a Rendered View](#) and [Render \(Camera\) Properties](#)

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Quality Rendering: Sets the rendering type when the view is rendered in **Quality** rendering mode. See [Render \(Camera\) Properties](#)

Perspective: Activates **Perspective** mode. See [The Camera](#).

Angle: The view angle for perspective mode.

Position: The location of the camera position.

Target: The location of the point the camera is facing.

UP Vector: A point defining the up direction of the camera.

Camera Sets

A camera set is a group of cameras, which can be handy if you want to set rendering, perspective, and visibility parameters for multiple cameras.

Select **Cameras** in the upper pane of the Design Director.

On the lower pane, select the cameras you want to include in the camera set.

Select **Create Set** in the Design Director toolbar or local menu.

Select **Camera Sets** in the upper pane in order to see the defined camera sets in the lower pane.

Select the properties (visibility, render type, etc.) of the camera set. When the camera set is made **Active**, these properties will be automatically applied to all cameras in the set.

Design Director-Categories

Default UI Menu: Tools/Palettes/Design Director/Category

Ribbon UI Menu:

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In the Design Director, "Categories" refers to groups of objects - 2D, 3D, cameras, etc. You can insert a category into the Design Director in order to have easy access to zooming, visibility, and other properties of the object group.

Note: To add **single** objects to the Design Director, use **Graphics**.

To place a group of objects into the Design Director:

Select the objects in the drawing space.

In the Design Director select **Categories** in the upper pane and select **Create New** from the toolbar or local menu.

Accept the default name or assign one that will help you recognize the object.

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All object groups you add to the Design Director will appear in the lower pane. The lower pane contains the following options:

Name			Color	+	-	
Widgets			0,0,0	+	-	
components			255,255,25	+	-	
purchaed			255,255,25	+	-	

ZoomTo: Zooms to the extents of the objects.

Visible: Displays or hides the objects.

Color: Sets the object's pen color.

Add To Category: Adds the selected object to the category.

Remove From Category: Removes the selected object from the category.

Design Director-Graphics

Default UI Menu: Tools/Palettes/Design Director/Graphic

Ribbon UI Menu:

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In the Design Director, "Graphics" refers to objects - 2D, 3D, cameras, etc. You can insert **single** objects into the Design Director in order to have easy access to zooming, visibility, and other properties of the objects.

Note: To add **groups** of objects to the Design Director, use **Categories**.

To place a graphic object into the Design Director:

Select the object in the drawing space.

In the Design Director select **Graphics** in the upper pane and select **Create New** from the toolbar or local menu.

Accept the default name or assign one that will help you recognize the object.

All objects you add to the Design Director will appear in the lower pane. The lower pane

Name			 Color		
 Graphic_0			 0,0,0		
 tri			 0,0,0		
 Circle one			 0,0,0		

contains the following options:

ZoomTo: Zooms to the extents of the object.

Visible: Displays or hides the object.

Color: Sets the object's pen color.

View by WorkPlane: Displays the view by the workplane that is set as **WorkPlane** by Entity. See [Changing the Workplane](#).

Get WorkPlane: Sets the current workplane by this object.

Design Director-Layers

Default UI Menu: Tools/Palettes/Design Director/Layer

Ribbon UI Menu:

Enables you to manipulate all the layers and layer sets in the drawing (see [Snap Modes](#) and [Layer Sets](#)).

To create a new layer or layer set, select **Create New** in the Design Director toolbar or local menu.

When **Layers** or **Layer Sets** is selected in the upper pane of the Design Director, the lower pane has the following options:

Active: Sets the layer as active, or applies the layer set. Also indicates the layer for a selected object or tool.

Visible: Sets the layer visibility.

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Lock: Use this switch to lock layers, which means the objects on them will be read-only. You can add objects to locked layers, but you cannot edit or delete objects.

Color: Sets the layer color.

Order: The order number of the layer.

Line Style: specifies the Line Style for the layer.

Pen Width: Specifies the Pen Width for the layer.

Print Style: Specifies the Print Style for the layer.

Layer Sets

A layer set is a group of layers, which can be handy if you want to set visibility, locking, and other parameters for multiple layers.

Select **Layers** in the upper pane of the Design Director.

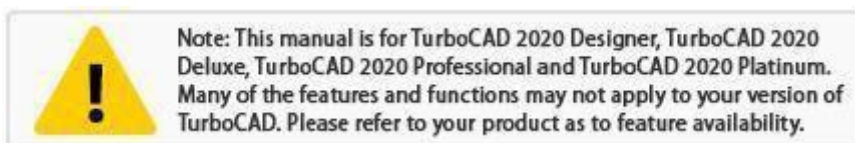
On the lower pane, select the layers you want to include in the layer set.

Select **Create Set** in the Design Director toolbar or local menu.

Select **Layer Sets** in the upper pane in order to see the defined layer sets in the lower pane.

Select the properties (visibility, locking, etc.) of the layer set. When the layer set is made **Active**, these properties will be automatically applied to all layers in the set.

Layer Filters



Layer filters provide you with a powerful tool to organize your work, workflow and drawing. This feature also allows you to import DWG file that have embedded layer filters

Layer filters will organize you layer by a defined set of parameters.

Layer Templates

Layer templates allow you to create and save alternate configurations for layers. Layer templates store how layers are setup, but do not store layers themselves. Layer templates are saved in a *.lrs file which can be stored anywhere in your system directory. All layers which meet the parameters defined by the filter will be grouped in the lower panel of the Design Director.

Note: The Show Filters and Group by Filters options must be turned on in the Design Director options for Filter groups to be displayed.

You can control all of the settings for the layers in the group. This means that if you change the color adjacent to the filter the color for all of its layer will be change to that color. If you change the visibility adjacent to the filter the visibility of all its layer will be changed to that visibility. If for any setting the layers contained in the filter have mixed settings for an option the filter will display a blank field or an ellipsis (...) for that setting. however you can still control the settings of the filtered layers.

To Change Layer Color Properties via the Filter

In the Design Director of a filtered drawing:

In the lower panel of the Design Director, hover the cursor over the color field adjacent to a filter.

Click on the color field.

When the Color dialog appears, select a color. The colors of all the filtered layers will be changed to the selected color.

To Create a Filter

1. Click the Create Layers Filter button in the toolbar of the Design Director.

When Prompted assign a name to the filter.

When the Filter Dialog appears, specify the parameters for the filter.

Click OK.

To Edit a Filters Parameters

Select the Filter in the top Panel of the Design Director.

Right click and Select Edit Filter Parameters..

When the Filter Dialog appears, reset the parameters for the filter.

Click OK.

Layer Filter Dialog

The Layers filter dialog allows you to specify how your filters work.

The dialog is divided into three sections: Toolbar, Filter Parameters table, Filtering Results table.

Filter Toolbar

Parent Filter: Filter can have sub-filters. This setting allows you to specify a parent filter that will constrain the list of layer that the current filter will apply to.

Insert Wildcard: This button opens the Wildcard dialog which allow you to insert a

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wildcard into the currently selected field. The dialog also defines how each of the wildcards function.

Duplicate Row: This button duplicates the currently selected row in the Filter Parameters table.

Delete Row: This button deletes the currently selected row in the Filter Parameters table.

Filter Parameters

The Filter Parameters table is where you specify how the filter will work by entering a set of parameters. Parameters are defined by values and wildcards. Each field in takes specific types of values.

Name: The Name field uses the name of layers to filter. Full names or parts of names can be combined with wildcards to specify what layers you want the filter to catch. The name field is the most versatile in regard to using wildcards.

Active: The Active field uses the Active status of layers to filter. It can take two values TRUE or FALSE. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~TRUE means = is not true. Since there can be only one Active layer at anytime, this option will make the filter dynamic as to what layers are captured.

Visible: The Visible field uses the Visible status of layers to filter. It can take two values TRUE or FALSE. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~TRUE = is not true.

Locked: The Locked field uses the Locked status of layers to filter. It can take two values TRUE or FALSE. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~TRUE = is not true.

Color: The Color field uses the Colors of layers to filter. It takes values which identify

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colors, either as RGB (True Color) values expressed as three numbers separated by commas, e.g. 10,20,30, or as Index colors expressed as a single number ranging from 1 to 255. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~10 = is not color 10. You cannot embed wildcard within the RGB value.

Line Style: The Line Style field uses the Line Styles of layers to filter. It takes values which are the names of Line Styles, such as CONTINUOUS or BORDER. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~BORDER = is not BORDER.

Pen Width: The Pen Width field uses the Pen Widths of layers to filter. It takes values which are the numeric values of the pen widths along with the units. For Example 0.5in or 33 mm. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~0.5in = is not 0.5in.

Order: The Order field uses the Order of layers to filter. It takes values which are the numeric values of the Order. For Example 1 or 55. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~10 = is not 10.

Print Style: The Print Style field uses the Print Styles of layers to filter. It takes values which are the names of Print Styles, such as 'Normal' or New Style. These can be combined with the NOT (~) wildcard to signify the opposite meaning. E.g. ~BORDER = is not BORDER.

Wildcards

Wildcards are special characters that are combined with other values to constrain a result. The wildcards that are supported by the filters are:

(pound) Matches any single numeric digit. For example, if there are 1000 layers named 1 to 1000, you can build the filter to capture all of the layers from 500 to 599 by using 5##, or 550 to 559 by using 55#

(at) Matches any single alphabetic character. For example, if there four layers named EAST, WEST, BEST and TEST, you can build a filter to capture all four by using @@ST, or just the last three layer by using @EST

. (period) Matches any single non alphanumeric character. For example, if there were four layers named TOP_VIEW, TOP-VIEW, TOP VIEW and TOP:VIEW, you can build a filter to capture all four by using TOP.VIEW

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(asterisk) Matches any string (sequence of characters) and can be used anywhere in the search string, beginning, middle or end. For example, you have dozens of layers that include the term 1ST, you can capture all of them by using **1ST**

(question mark) Matches any single character; for example, ?BC matches ABC, 3BC, etc. As another example, if there four layers named EAST, 2EST, _EST and TEST, you can build a filter to capture all four by using ??ST, or just the last three layer by using ?EST

(tilde) Matches anything but the pattern, in other words it equals NOT. For example; ~AB matches all strings that don't contain AB. As another example, you have dozens of layers that include the term 1ST, you can exclude them all from being captured by using ~1ST

[] Matches any one of the characters enclosed; for example, [AB]C matches AC and BC. As another example, all of the layers begin with a single number, and you want to capture just those that begin with 5, 7 and 8. You can capture them by using [578]*

Matches any character not enclosed; for example, [~AB]C matches XC but not AC. As another example, all of the layers begin with a single number, and you want to exclude from capture just those that begin with 5, 7 and 8. You can do this by using [~578]*

[-] Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC. As another example, all of the text layers have a format like TXT followed by a number followed by more characters e.g. TXT1TOP or TXT2BOTTOM. You want to capture any with the numbers 2, 3, 4, or 5. You can capture all of them using TXT[2-5]*

(reverse quote) Reads the next character literally; for example, `~AB matches ~AB. This is especially useful if your layers have characters in their names that otherwise would be interpreted as wildcards.

Important Items to remember:

All items on the same row of a filter further refine (tighten) the results of the filter. For example, on one row, if under the Name field you have **1ST** and in the Color field you have 55, the filter will capture only those items that have both 1ST and are Color 55

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Items on separate rows expand the results of the filter. For example, if on one row in the Name field you have **1ST** and on the next row in the Color field you have 55, the filter will capture all layers with 1ST and all layers with Color 55.

Sub filters are restricted to capturing only layers which are contained within their parent filters.

Filter Results

The Filtering Results table displays results of the filter as defined in the Filter Parameters table, this gives you a preview of how your filter will operate.

There are several functions which are available if you right click a field in the Filtering Results table.

Refine using this value: If this option is chosen the value in the field will be added to each row of the Filter Parameters table with content. This further restricts the results of the filter.

Refine excluding this value: If this option is chosen the value in the field will be added to each row of the Filter Parameters table with content with a ~ prepending the value. This further restricts the results of the filter.

Add all with this value: If this option is chosen the value in the field will be added to a new row of the Filter Parameters table. This expands the results of the filter.

Add all without this value: If this option is chosen the value in the field will be added to a new row of the Filter Parameters table with a ~ prepending the value. This expands the results of the filter.

Filters and XREFs

When external references (XREFs) are added to a file all of the layers of the external referenced file are appended. The layers are listed under the XREF name as if the XREF was itself a filter. In addition the XREF is listed under the filters in the top panel of the Design Director. Within the Design Director lower panel you can treat the XREF as if it is a filter, including changing the values for the properties of all the layers by making changes at the filter level. However, you cannot edit the parameters of the XREF as if it were a filter.

Design Director-Lights

Default UI Menu: Tools/Palettes/Design Director/Light

Ribbon UI Menu:

Enables you to manipulate all the lights and light sets in the drawing (see [Lights](#)). To create a new light or light set, select **Create New** in the Design Director toolbar or local menu.

When **Lights** or **Light Sets** is selected in the upper pane of the Design Director, the

bottom pane has the following options:

The lower pane contains the following options:

On/Off: Turns the light on and off.

Visible: Displays or hides the light indicator in the drawing.

Color: Sets the color of the light.

Power: Specify the power of the light in units of Watts.

Type: Sets the type of light (headlight, spot light, etc.). See [Creating Lights-Light Types](#)

Position: The coordinates of the light source, relevant for Point, Spot, and Sky lights.

Target: The target point of a directional light, relevant for Directional, Spot, and Sky lights.

Light Sets

A light set is a group of lights, which can be handy if you want to set on/off, visibility, and other parameters for multiple lights.

Select **Lights** in the upper pane of the Design Director.

On the lower pane, select the lights you want to include in the camera set.

Select **Create Set** in the Design Director toolbar or local menu.

Select **Light Sets** in the upper pane in order to see the defined light sets in the lower pane.

Select the properties (on/off, visibility, etc.) of the light set.

Lights Dialog

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Refresh: Refresh the Light manager display.

New Light Template: Creates a new light template from the currently selected layers.

New Light: Creates a new light. The default name will include a prefix by default, but the name can be changed.

Delete Light: Delete the selected lights.

Activate: Sets the currently selected layer (only one) as the active layer. This will effect any currently active drawing tool or currently selected object. **Select By:** Selects the currently selected lights.

Edit Properties: Opens the light properties dialog.

Visible/Invisible All: If any layer is invisible all lights are made visible. If all lights are visible they are all made invisible.

Invert Visibility: Turns all visible lights invisible, and all invisible lights visible.

Exclusive Visible: Makes only the currently selected lights visible, all other become visible.

ON/OFF All: If any light is off all it is turned on. If all lights are on they are all turned off.

Invert On: Lights that are on are turned off, lights that are off are turned on.

Exclusive On: Turns on only the currently selected lights, all others are turned off.

Design Director Options

Default UI Menu: Tools/Palettes/Design Director

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Ribbon UI Menu:

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General options for items to display or hide, how the mouse will be used, and confirmation windows.

Show Tool Bar: Select to display the Tool bar.

Show Entries Tree: Select the items you want displayed in the Design Director.

Double-Click Action: Sets the action performed when double-clicking on either the right or left mouse button.

Show New Name Confirmation Dialog: If checked, you will be asked to confirm the name of each new item you create using **Create New**. If not checked, the names will be assigned automatically.

Replace Standard Dialogs: Select the items you want displayed using the Design Director components as dialogs instead of standard dialogs.

Design Director-Spaces

Default UI Menu: Tools/Palettes/Design Director/Space

Ribbon UI Menu:

In the Design Director, "Spaces" refers to the Model space and all of the Paper spaces. Using Design Director-Spaces serve as an alternative to using the workspace tabs at the bottom of the screen.

Model space contains a list of named views,
Each paper space has a list of viewports.

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All items have an "Activate" property:

Click on a model space item - the model space is activated with a current view

Click on a named view item - the model space is activated at first and the selected view

Click on a paper space item - the selected paper space is activated with a current view

Click on a viewport item - the selected paper space is activated at first and zoom to the selected viewport

All items have a local menu (Right Click):

Model space item - "Activate" and "Properties".

Paper space item - similarly the space tab local menu

Design Director Toolbar and Menu

The following options are available on the Design Director toolbar and on the local menu. The available options depending on the item chosen (layer, layer set , workplane, etc.)

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Refresh: Redraws the Design Director pane.

Activate: (hotkey: Ctrl+A): Applies, or makes active, the selected item.

Select By: (hotkey: Ctrl+S): Selects the selected item in the drawing space .

Edit Properties: (hotkey: Ctrl+P): Change properties of the selected item.

Create New: (hotkey: Ctrl+N): Adds a new item.

Create Set: Creates a set of all or selected items in the table. Relevant for layers, lights, and cameras.

Delete: (hotkey: Ctrl+D): Deletes the selected item.

Options: Invokes the **Options** window to customize the Design Director.

Open as Separate Palette: Open the selected item as separate palette. This is useful if you want to keep multiple sections of the Design Director open at once. The following options appear on the local menu only:

Load From File: (hotkey: Ctrl+Shift+O): Loads an item from the file with the corresponding extension.

Save To File: (hotkey: Ctrl+Shift+S): Saves an item to the file with the corresponding

extension.

Show Toolbar: Shows or hides the Design Director toolbar.

Design Director-Views

Default UI Menu: Tools/Palettes/Design Director/View

Ribbon UI Menu:

Enables you to manipulate and create new views. To create a new view from the current display, select **Create New** in the Design Director toolbar or local menu. When **Views** is selected in the upper pane of the Design Director, all views you saved (see [Saving Views](#)) will appear in the lower pane. The lower pane contains the following options:

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Name	✓	☒	◀ An...	Wirefr...	Hidden...	Draft R...	Quality...	Position	Target	Up Vec...	Extents
View_0	✓		IS	Native Wirefr	Redskd Redskd	Redskd Redskd	Redskd Raytra	x: 0 y: 0 z: 1	x: 0 y: 0 z: 0	x: 0 y: 1 z: 0	Left: 0 Top: 0
main view		☒	35.00000000	Native Wirefr	Redskd Redskd	Redskd Redskd	Redskd Raytra	x: 24.7476 y: 4	x: 23.5515 y: 3	x: -0.395481 y:	Left: -7.13129
standard view			35.00000000	Native Wirefr	Redskd Redskd	Redskd Redskd	Redskd Raytra	x: 24.7476 y: 4	x: 23.5515 y: 3	x: -0.395481 y:	Left: -7.13129
Top view			35.00000000	Native Wirefr	Redskd Redskd	Redskd Redskd	Redskd Raytra	x: 24.7476 y: 4	x: 23.5515 y: 3	x: -0.395481 y:	Left: -7.13129

Active: Sets the view as active.

Draft Rendering: Sets the rendering type when the view is rendered in **Draft** rendering mode. See [Creating a Rendered View](#) and [Render \(Camera\) Properties](#)

Quality Rendering: Sets the rendering type when the view is rendered in **Quality** rendering mode. See [Render \(Camera\) Properties](#)

Perspective: Activates **Perspective** mode. See [The Camera](#).

Angle: The view angle for perspective mode.

Position: The location of the camera position.

Target: The location of the point the camera is facing.

UP Vector: A point defining the up direction of the camera.

Extents: The boundaries of the view.

Design Director-Workplanes

Default UI Menu: Tools/Palettes/Design Director/WorkPlane

Ribbon UI Menu:

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Enables you to manipulate and create new workplanes. See [Workplanes](#)

To create a new workplane, select **Create New** in the Design Director toolbar or local menu.

When **Workplanes** is selected in the upper pane of the Design Director, all workplanes you saved using **Set Named Workplane** (see [Saving and Recalling WorkPlanes](#)) will appear in the lower pane. The lower pane contains the following options:

Active: Sets the workplane as active; all inserted objects will be located or based on this workplane. Also indicates the workplane for a selected object or tool.

View by WorkPlane: Sets the view normal to the workplane (workplane is against the screen).

Origin: The point that represents the UCS origin. See [3D Coordinate Systems](#)

X Vector: The point that defines the direction of the X-axis in the UCS.

Z Vector: The point that defines the direction of the Z-axis of the UCS.

3.11. Repeat Last Command

Repeat Last Command

(Available in all TurboCAD variants)

Default UI menu: Draw/Repeat

Ribbon UI menu:

The "Repeat" command allows the user to run the previous/most recently used command with one click.

Simply click on this icon and the previously used Tool or process will become active.

4. Inserting 2D Objects

(Available in all TurboCAD Variants)

The **Draw** menu contains all of the TurboCAD drawing tools.

Note: The tools covered in this section are 2D. For 3D geometry, see [Creating 3D Objects](#)

You can set object properties before they are created, or modify properties of existing objects. See [Object Properties](#). Before creating 2D objects, it is a good idea to familiarize yourself with the following concepts:

See [Coordinate Systems](#)

See [Grid](#)

See [Snaps](#)

See [Snap Modes](#)

See [Construction Geometry](#)

For most objects, you can define size and location by using the cursor, or by specifying exact sizes or dimensions. See and [Coordinate Fields](#)

Note: If you are inserting 2D objects while working in 3D, the objects will be placed on the current workplane. See [Workplanes](#)

LTE Type Objects

In the LTE Workspace 2D tools behave with slight differences to standard TurboCAD methods.

They allow use of dynamic input.

They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

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They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

You can complete drop the tool by pressing the either the Enter or Space keys.

You can re-activate the tool by pressing the Space Key again.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

4.1. Point

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Point

Ribbon UI Menu:

You can display the Point and Arrow toolbar by right-clicking in any toolbar area and selecting Point and Arrow from the toolbars.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.



Use **Point** to create several different types of point markers: dot, square, cross, star, or circle. By default, a point in 2D space. If you want to create a 3D point, select **3D Point** from the Inspector Bar or local menu.



Point Properties

The **Point** page is included in the **Properties** window when a point is selected, or when the point tool is active.

Color: Select the point color.

Point Type: Select dot, star, square, cross, circle, or the AutoCAD point.

Size: Sets the width and height of the point. This value affects all types of points except **Dot**, which is always displayed at a minimal size.

4.2. Line

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Line

Tools for drawing lines and linear objects. You can display the **Line** toolbar by right-clicking in any toolbar area and selecting **Line**.

Ribbon UI Menu:

LTE T ype Objects



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

In the LTE Workspace 2D tools behave with slight differences to standard TurboCAD methods.

- They allow use of dynamic input.

- They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

- They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

- You can complete drop the tool by pressing the either the Enter or Space keys.

- You can re-activate the tool by pressing the Space Key again.

With the exception of the Single line tool, all LTE Workspace Line tools work exactly as the equivalent TC tools except as noted in the above list.

Bisector

Default UI Menu: Draw/Line/Bisector

Ribbon UI Menu:

The Bisector line allows you to create a line which bisects an angle. It has two modes:
By two lines, and by three points.

By two lines:

Click on the first line.

Click on the second line

Click between the two lines.

Drag and click to create a length or type a value into the Length field and press Enter.

By three points:

Click on the first point. this will be the first edge of the angle to be bisected.

Click on the second point. this will be the vertex of the angle to be bisected.

Click on the third point. this will be the second edge of the angle to be bisected.

Drag and click to create a length or type a value into the Length field and press Enter.

Irregular Polygon



Default UI Menu: Draw/Irregular Polygon

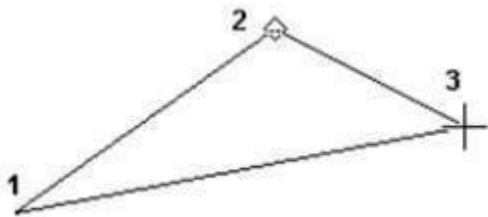
Ribbon UI Menu:

Creates a polygon (closed) with irregular sides and angles. This is equivalent to using the **Polyline** tool to create line segments in "closed" mode.

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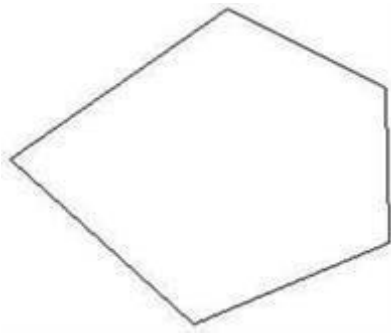
Define the first two vertices, creating the first segment. You can also use the Inspector Bar to enter a length and angle.

Define the next segment. From this new vertex, a closing segment is created back to the first vertex.



Create as many segments as needed. A closing segment is always created from the last vertex.

4. Select **Finish** from the local menu or Inspector Bar, or press Alt+F.



Note: To undo polygon segments in reverse order, select **One Step Back** from the local menu or Inspector Bar (or press Backspace).

Options Fillet Corners: If this option is turned on a the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.



Minimal Distance



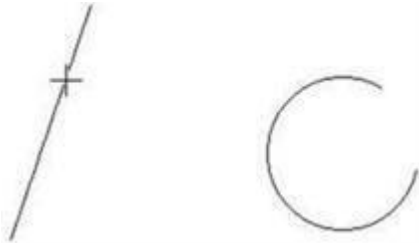
Default UI Menu: Draw/Line/Minimal Distance

Ribbon UI Menu:

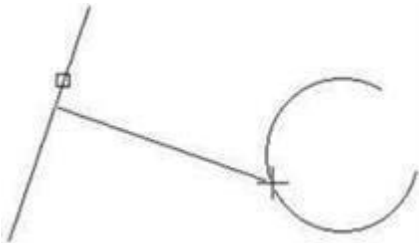
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Creates the line representing the shortest distance between two 2D objects. The objects must be on the same workplane.

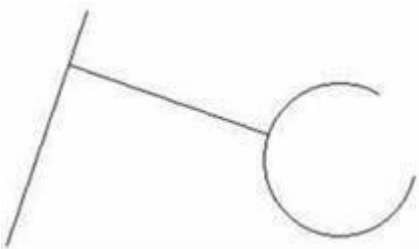
1. Select the first object from which you want to draw the shortestline.



2. Select the second object.

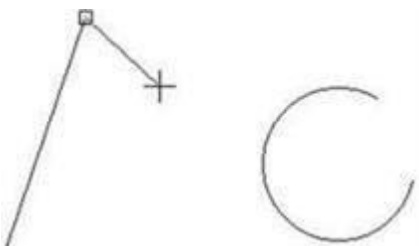


3. The shortest-distance line is created.

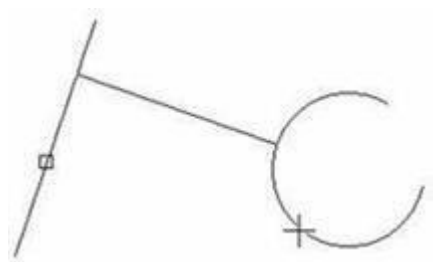


Local menu options:

Through Point: Creates a line passing through a specified point. The through point should only be specified for one of the objects, otherwise a standard single line is created.



Show beforehand displays the shortest line to an object when you pass the cursor over it. This allows you to preview the line before creating it.



Parallel



Default UI Menu: Draw/Line/Parallel

Ribbon UI Menu:

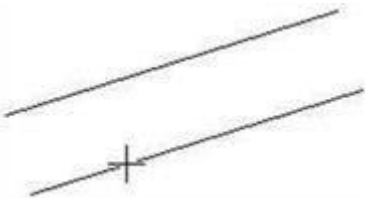
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Creates a line parallel to a existing line.

Select an existing line.

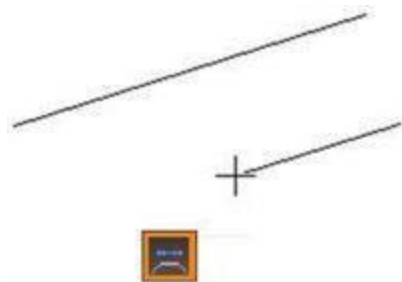


Select the location of the parallel line, or enter the offset in the Inspector Bar. By default, the parallel line will have the same length as the original line.



Local menu option:

Keep Length (default): If not used, you can create a parallel line with a different length than the original line. You can only adjust one endpoint of the line - the endpoint closest to where you selected the original line. You can enter both the offset and length in the Inspector Bar.



Perpendicular



Creates a line perpendicular to an existing line.

Default UI Menu: Draw/Line/Perpendicular

Ribbon UI Menu:

Tip: This tool creates a line **from** an existing line. To create a perpendicular line **to** a line, you can use the "J" SEKE (see [Running and Local Snaps](#))

Select an existing line.



Select the point on the existing line where the perpendicular line will start. You can select a point past the endpoints.



Select a third point to set the length of the perpendicular line, or enter the length in the Inspector Bar.



Local menu option:

Limit to Segment: The perpendicular line cannot extend past the endpoints of the existing line.

Tip: To draw a line perpendicular to an arc or circle, draw a single **Line** from the center of the arc or circle to its destination, then trim the line.

Polygon



Default UI Menu: Draw/Polygon

Ribbon UI Menu:

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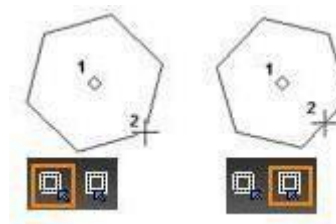
Creates a regular (equal-length sides) polygon.

Enter the number of sides in the Inspector Bar. If desired, you can also specify the **Angle** (angle of control point from the center), and **Radius** or **Side** (length of one side).

Select the mode: Vertex, Segment or Edge.

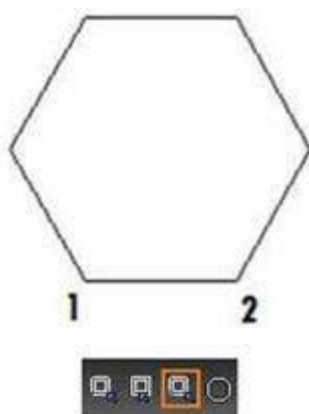
If you are using Vertex or Segment mode, select the polygon centerpoint.

Use the **Segment Mode** from the local menu or Inspector Bar to use the midpoint



of one segment as a control point.

Or, select **Edge Mode** from the local menu or Inspector Bar to use the midpoint of one segment as a control point.



Options

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Fillet Corners: If this option is turned on the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.



Create Pattern Constraint: (Mechanical Edition Only) If this option is turned on a pattern constraint will be created from the resulting polygon. Auto Add Constraints must be turned on for this feature to work.

Polyline

Default UI Menu: Draw/Polyline

Ribbon UI Menu:

Creates a series of connected straight line or arc segments that comprise one object. The segments can have different and/or tapered line widths.

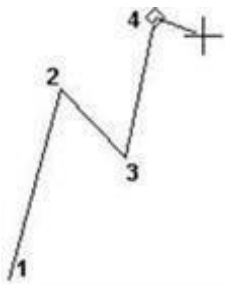


Note: If you want to create a polyline by joining a series of existing line or arc segments, see [Join Polyline](#) or [Chain Polyline](#)

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By default, line segments are created (as opposed to arcs). Select each endpoint, or enter segment lengths and angles in the Inspector Bar.

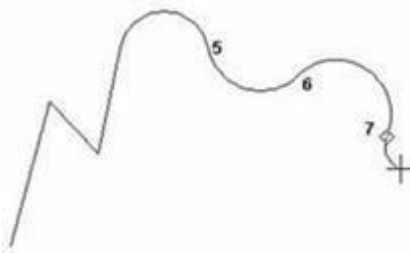
Note: You can choose to enter an Angle which is absolute in relation to the world or a Relative Angle which is the angle from the immediately previous segment (arc or line).



To draw arc segments, select **Arc Segment** from the local menu or Inspector Bar.



By default, each arc segment is tangent to the previous segment, but the tangency can be changed via the local menu options. Select each arc endpoint, or enter

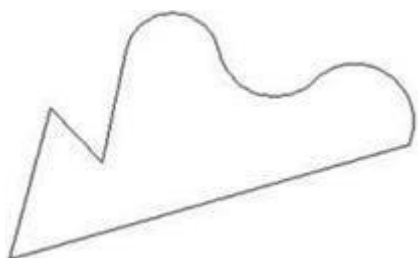


parameters in the Inspector Bar.

To switch back to lines, select **Line Segment** from the local menu or Inspector Bar.



When all segments are created, select **Finish** (Alt+F) from the local menu or Inspector Bar, or double-click the last point. If you want to create a line or arc segment connecting the first and last points, select **Close** (Alt+C).



Local menu options (Arcs):

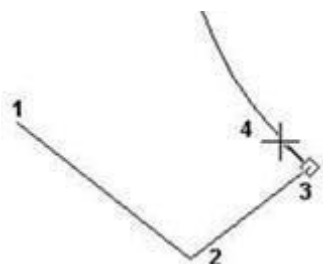
Direction: Rather than creating an arc segment tangent to the last segment, you can set the start angle of the arc segment.



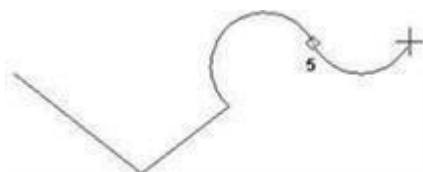
After selecting Direction, you can set an Angle which is absolute in relation to the world and a Relative Angle which is the angle from the immediately previous segment.



For the arc segment, move the mouse to set the start angle, or enter the angle in the Inspector Bar.



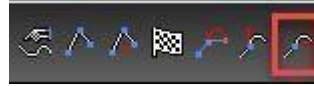
Select the arc endpoint, or enter its parameters in the Inspector Bar. The next arc segment you create will revert to the default - tangent to the previous segment.



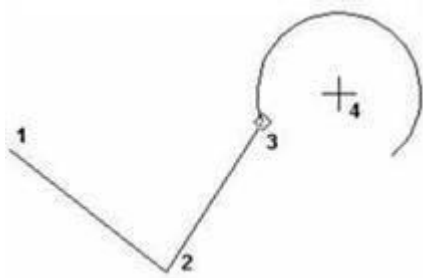
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Center: Rather than creating the arc tangent to the last segment, you can set the start

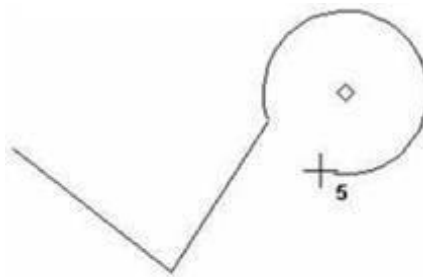
angle and size by selecting the arc center.



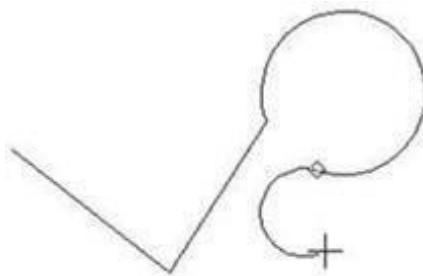
For the arc segment, select the arc center, or enter the arc length in the Inspector Bar.



Size the arc by selecting its endpoint, or enter its angle or arc length in the Inspector Bar.



The next arc segment you create will revert to the default - tangent to the previous segment.

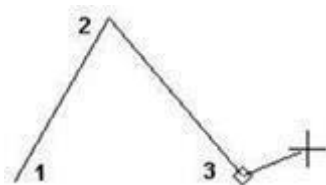


Demo Video:

<https://youtu.be/8lZESNDfJvA>

Polyline Line Widths

. By default, polyline segments have the default line thickness (which can be set in the **Pen Width** box on the **Property** toolbar.)



For any segment, you can enter values for **Start** and **End Width**.

Start Width	End Width
0.05 in	0.12 in

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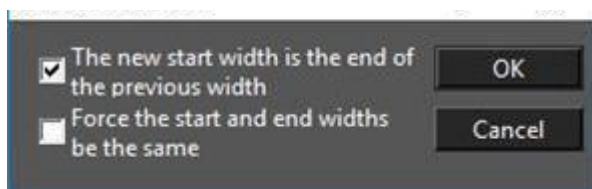
The next segment will have these properties, but the subsequent segments, by default, will revert to the default thickness.



To change the widths of adjacent segments you can open the **Polyline Properties** from the local menu or Inspector Bar.



In this window, you can force start and end widths to be equal, and to set the start width of each segment to be equal to the end width of the previous segment.



Activate both options if you want to apply a uniform thickness to the entire polyline, or to a contiguous group of segments.

Note: If you use **Start** and **End Width** to create a segment with tapering thickness, this taper will be lost when the polyline is exploded, and then the tapered segment is exploded.

Rectangle



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Default UI Menu: Draw/Rectangle

Ribbon UI Menu:

Creates an orthogonal rectangle by defining two diagonally opposite corners.

Select the first corner.



Select the diagonally opposite corner. You can also enter the size in the Inspector Bar.



Options

Fillet Corners: If this option is turned on a the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.



Rotated Rectangle



Default UI Menu: Draw/Rotated Rectangle

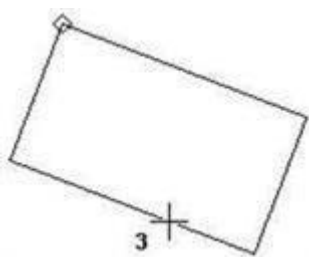
Ribbon UI Menu:

Creates a non-orthogonal rectangle.

Select two points to define the base. The angle of this line defines the orientation of the rectangle.



2. Define a third point that specifies the distance from the base to the top.

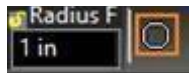


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You can also enter the length of both sides and the orientation angle in the Inspector Bar.

Options

Fillet Corners: If this option is turned on a the corners of the polygon will be rounded by filleting. A Radius F field appears in the Inspector bar that allows you to specify the radius of the fillets.



Single Line



Default UI Menu: Draw/Single

Ribbon UI Menu:

Creates a single line segment.

Define the segment start point.



Define the endpoint, or specify the length and angle in the Inspector Bar.



Best Fit Line



(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Draw/Line/Best Fit Line

Ribbon UI Menu:

The Best Fit Line tool draws a line that is best-fitted within a set of user-defined points. Points can be extracted from a specified location or from a specific object.

To draw this, select "Best Fit Line" and define at least 2 points on the work plane.

Select Finish and the line which fits best within the points is drawn.

LTE Wokspace

In the LTE Workspace the line tool creates a series of discrete line segments until you select Finish, or close from the local menu (or Command Line) or until you press the either the Enter or Space keys.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Tangent Arc Point

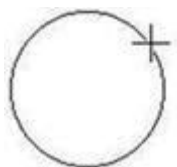


Default UI Menu: Draw/Line/Tangent Arc Point

Ribbon UI Menu:

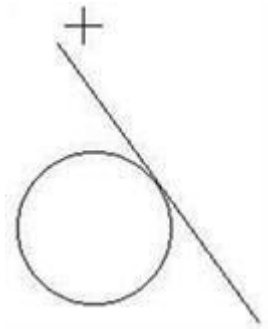
Creates a line tangent to an arc, circle, or ellipse, with the midpoint of the line located at the point of tangency.

1. Select the tangent arc, circle, or ellipse.

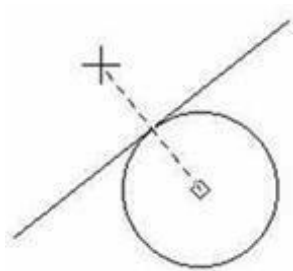


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Move the mouse to adjust the length of the line, or set the length in the Inspector Bar.
The line midpoint always remains at the tangent point.



Move the mouse to change the angle of the line and the point of tangency, or set the angle in the Inspector Bar.



Tangent from Arc

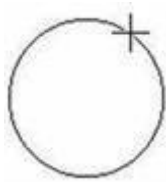


Default UI Menu: Draw/Line/Tangent from Arc or Curve

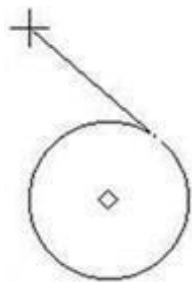
Ribbon UI Menu:

Creates a tangent line from an arc, circle, or ellipse.

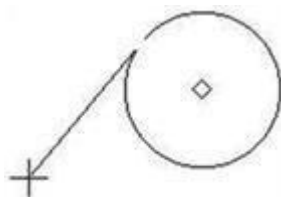
Click on the side of the arc or circle from which you want to draw the tangent line.



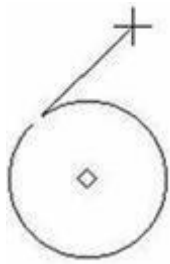
A tangent line appears from the selected point. Click to set the length, or enter the length in the Inspector Bar.



3. Move the cursor to set the line angle, or enter the angle in the Inspector Bar.



To switch the side of the tangent line, pass the cursor through the tangency point.



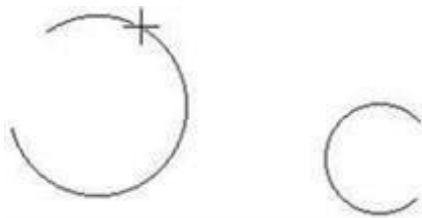
Tangent to 2 Arcs

Default UI Menu: Draw/Line/Tangent to 2 Arcs

Ribbon UI Menu:

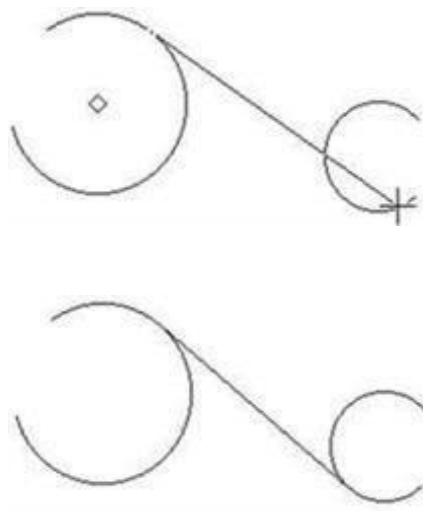
Creates a line tangent to two arcs or circles.

1. Select one of the tangent objects - arc, circle, or ellipse.



The tangent line extends from this object. To switch the side of the line, pass the cursor through the point of tangency. Adjust the line so that it is close to the desired point of tangency.

Select the second tangent object, close to the point of tangency. The tangent line is created.



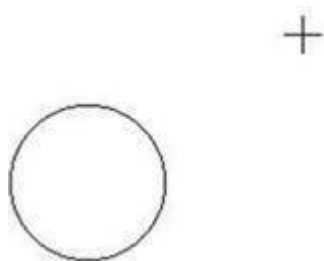
Tangent to Arc

Default UI Menu: Draw/Line/Tangent to Arc or Curve

Ribbon UI Menu:

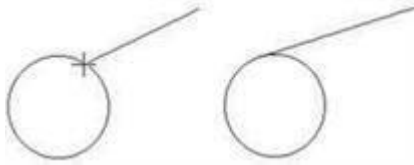
Creates a line tangent to an arc, circle, or ellipse, starting from a selected point.

1. Select the start point of the line.



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Select the tangent arc, circle, or ellipse, close to the point of tangency. The tangent line is created.

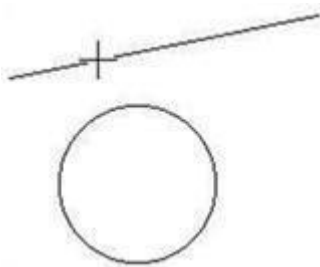


In this example, clicking on the other side of the circle would have created a different tangent line.

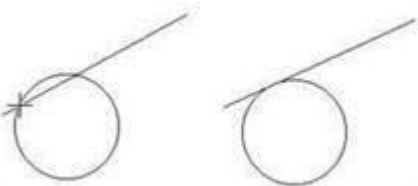


Tangent Line of Fixed Length

By default, **Tangent to Arc** creates a line from the start point, ending at the tangent object. To fix the length of the line, enter the length in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the line length remains constant.



Select the tangent object, close to the point of tangency. The fixed-length tangent line is created.



4.3. Circle-Ellipse

Circle-Ellipse

(Available in all TurboCAD Variants)

Default UI menu: Draw/Circle , Draw/Ellipse

Ribbon UI menu:

Tools for drawing circles and ellipses. You can display the **Circle/Ellipse** toolbar by right-clicking in any toolbar area and selecting **Circle/Ellipse**.



Circle - Center and Point

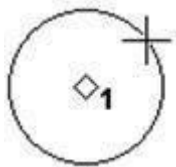


Default UI menu: Draw/Circle/Center and Point

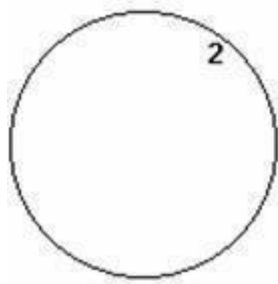
Ribbon UI menu:

Creates a circle by defining its center and a point on its circumference.

1. Select the circle center point. Move the cursor to size the circle.



Click to create the circle, or enter the radius, diameter, or circumference in the Inspector Bar.



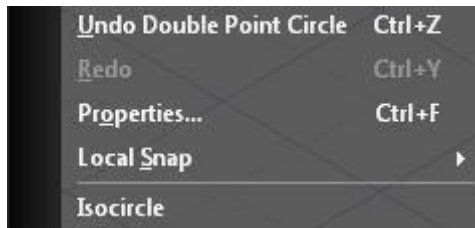
Isocircle

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

The Isocircle option is available in the Circle - Center and Point tool when an [Iso Grid](#) is active. It allows you to draw true isometric circles instead of ellipses.



The tool draws circles using the standard procedure.

Circle-Concentric



Default UI menu: Draw/Circle/Concentric

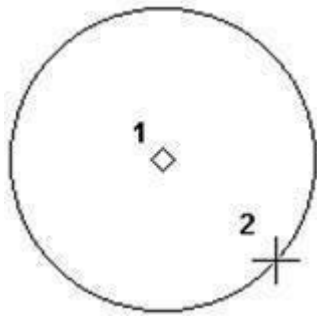
Ribbon UI menu:

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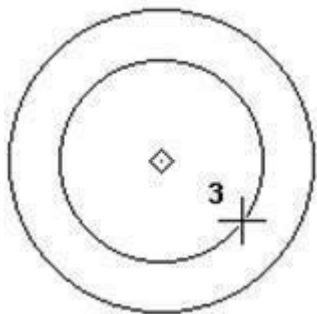
Creates circles that share a center point.

Select a centerpoint for the circles.

Size the first circle, or enter the radius, diameter, or circumference in the Inspector Bar.



3. Create the second circle the same way.



4. Create more circles as needed.

5. Finish by selecting **Finish** from the local menu or Inspector Bar, or press Alt+F.

Circle-Double Point

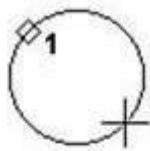


Default UI menu: Draw/Circle/Double Point

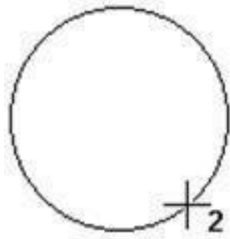
Ribbon UI menu:

Creates a circle by defining two endpoints of its diameter.

Select a point on the circle circumference.



Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.



Circle-Tangent to 3 Arcs



Default UI menu: Draw/Circle/Tangent to 3 Arcs

Ribbon UI menu:

Creates a circle tangent to three arcs or circles. You can specify whether the existing arcs will be inside or outside the new circle.

Select the first tangent object. Click slightly outside the object to keep it outside the new circle. Click inside if you want the tangent object to be inside the new circle.

If the object is clicked slightly outside, an outward-pointing arrow appears.

Select the second tangent object. If the object is clicked slightly inside, an inward-pointing arrow appears.

4. Select the third object. The tangent circle is created.

Circle-Tangent to Arc

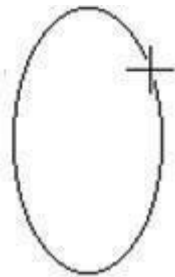


Default UI menu: Draw/Circle/Tangent to Arc or Curve

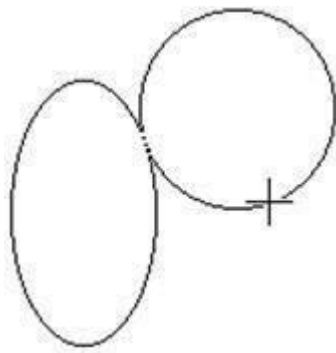
Ribbon UI menu:

Creates a circle tangent to an arc, circle, or ellipse.

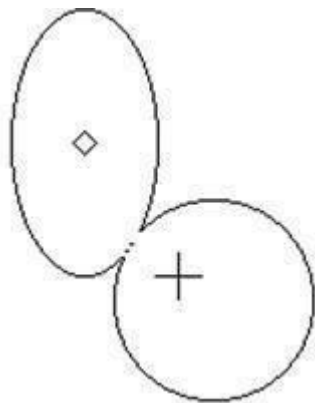
1. Select the existing arc, circle, or ellipse to which you want the circle to be tangent.



Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.) The circle will be created from the point you selected on the tangent object.



3. Move the mouse to position the circle, either inside or outside the tangent object.



Circle-Tangent to Entities



Default UI menu: Draw/Circle/Tangent to Entities

Ribbon UI menu:

Creates a circle tangent to three objects (rectangles, polygons, arcs, etc.)

1. Select the first tangent object.

2. Select the second tangent object.

3. Select the third tangent object. The tangent circle is created.

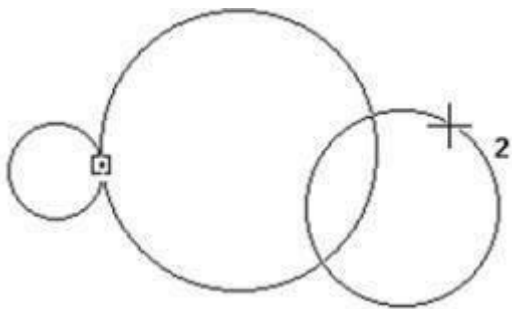
Local menu option:

Through point: Makes the circle pass through a specific point. In the example below, a vertex of the polygon was selected, rather than the polygon itself.

Tangent Circle of Fixed Size

By default, **Tangent to Entities** creates a circle that touches three objects. To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.

Select the second tangent object, close to the point of tangency.



The fixed-sized tangent circle is created.

Circle-Tangent to Line

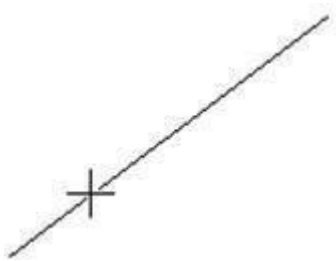


Default UI menu: Draw/Circle/Tangent to Line

Ribbon UI menu:

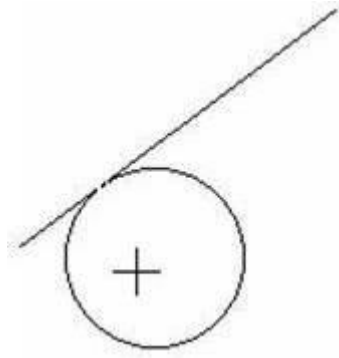
Creates a circle tangent to a line.

1 .Select the existing line to which you want the circle to be tangent.

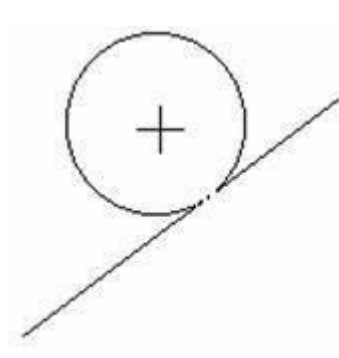


Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be

able to reposition the circle.) The circle will be created from the point you selected on the line.



3. Move the mouse to position the circle, on either side of the line.



Circle-Triple Point

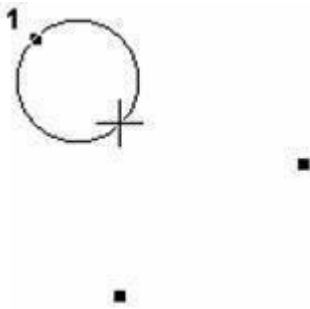


Default UI menu: Draw/Circle/Triple Point

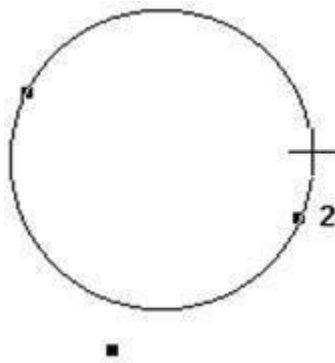
Ribbon UI menu:

Creates a circle that passes through three points.

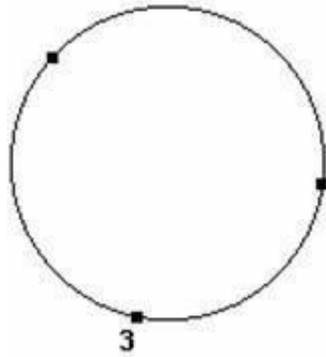
1. Select the first point on the circumference.



2. Select the second point.



3. Select the third point.



Best Fit Circle



(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Draw/Circle/Best Fit Circle

Ribbon UI Menu:

The Best Circle tool draws a circle that is best-fitted within a set of user-defined points. Points can be extracted from a specified location or from a specific object. To draw this, select "Best Fit Circle" and define at least 3 points on the work plane. Select Finish and the circle which fits best within the points is drawn.

To draw a best f it circle:

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Draw first point

Draw second point

Draw third point and/or other points

Select Finish

Ellipse



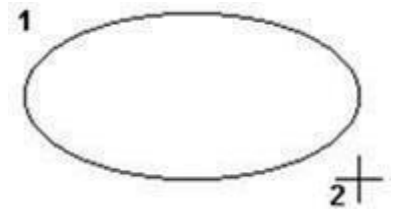
Default UI menu: Draw/Ellipse/Ellipse

Ribbon UI menu:

Creates an ellipse by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal.

Select the first corner of the bounding rectangle.

Define the diagonally opposed corner of the bounding rectangle, or enter the



major and minor axis lengths in the Inspector Bar.

Ellipse Fixed Ratio



Default UI menu: Draw/Ellipse/Ellipse Fixed Ratio

Ribbon UI menu:

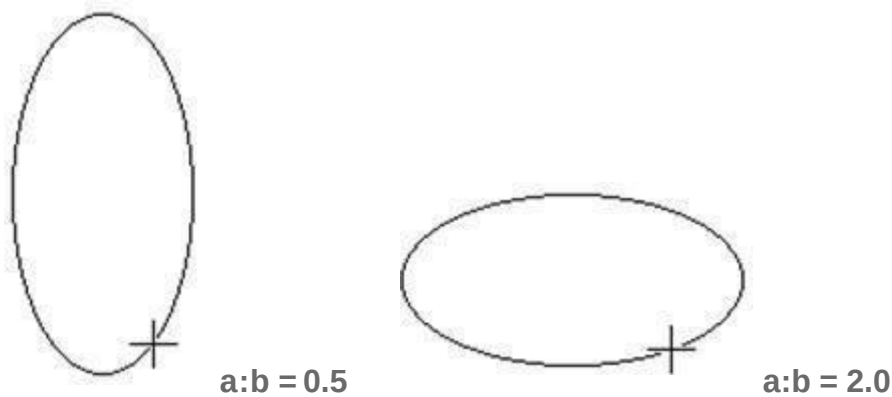
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Creates an ellipse by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (Lock this value if you want to repeat it later, otherwise the field reverts back to the default.)

Select the centerpoint of the ellipse.

Move the mouse to size the ellipse.



Tip: The fixed ratio for an "IsoCircle" (an isometric circle on a cube) is 1.73 (the square root of 3).

Rotated Ellipse



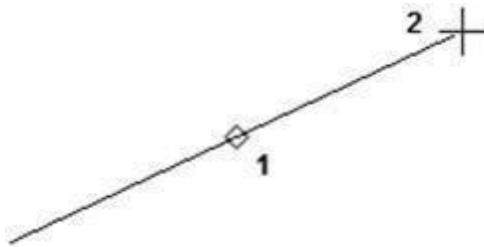
Default UI menu: Draw/Ellipse/Rotated Ellipse

Ribbon UI menu:

Creates a tilted ellipse.

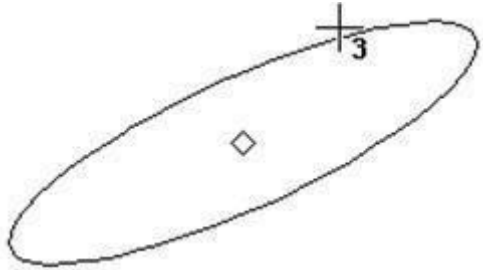
Select the centerpoint of the ellipse.

Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



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Set the length of the minor axis, which is always perpendicular to the major axis.
You can also enter the axis length in the Inspector Bar.



LTE Workspace Circle_Ellipse Tools



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In the LTE Workspace 2D tools behave with slight differences to standard TurboCAD methods.

They allow use of dynamic input.

They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

You can complete drop the tool by pressing the either the Enter or Space keys.

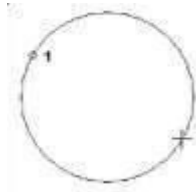
You can re-activate the tool by pressing the Space Key again.

Note: LTE Workspace is not a feature of TurboCAD Deluxe

Circle_2 Points

Creates a circle by defining two endpoints of its diameter.

Select a point on the circle circumference.



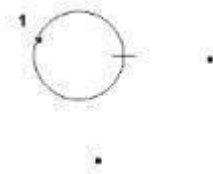
Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.



Circle_3 Points

Creates a circle that passes through three points.

Select the first point on the circumference.



Select the second point.



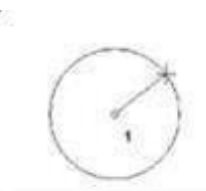
Select the third point.



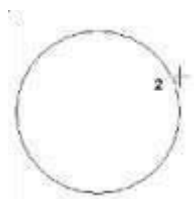
Circle_Center, Radius

Creates a circle by defining its center and a point on its circumference.

Select the circle centerpoint. Move the cursor to size the circle.



Click to create the circle, or enter the radius, diameter, or circumference in the Inspector Bar.



Local menu options:

The local menu contains options to switch to any of the other circle tools.

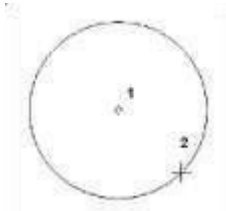


Circle_Concentric

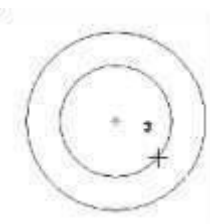
Creates circles that share a centerpoint.

Select a centerpoint for the circles.

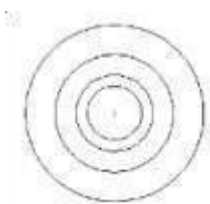
Size the first circle, or enter the radius, diameter, or circumference in the Inspector Bar.



Create the second circle the same way.



Create more circles as needed.

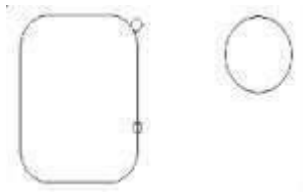


Finish by selecting **Finish** from the local menu or Inspector Bar, or press Alt+F.

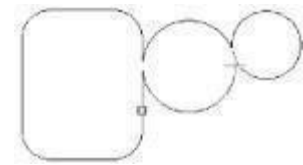
Circle_Tan, Tan, Rad

Creates a circle tangent to two objects, with a fixed radius.

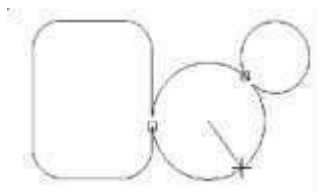
Select the first tangent object.



Select the second tangent object.



Move the mouse to create the tangent circle, or enter the Radius in the Inspector Bar.



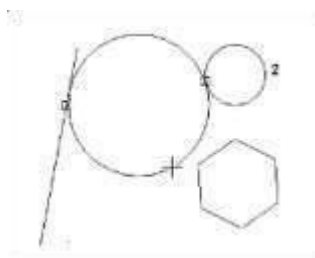
Circle_Tan, Tan, Tan

Creates a circle tangent to three objects (rectangles, polygons, arcs, etc.)

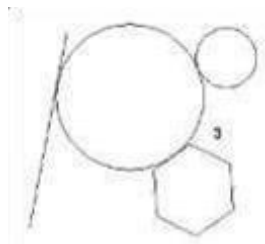
Select the first tangent object.



Select the second tangent object.



Select the third tangent object. The tangent circle is created.

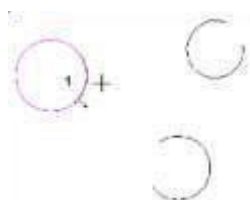


Circle_Tangent to 3 Arcs

Creates a circle tangent to three arcs or circles. You can specify whether the existing arcs will be inside or outside the new circle.

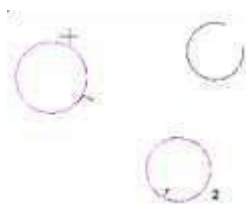
Select the first tangent object. Click slightly outside the object to keep it outside the new circle. Click inside if you want the tangent object to be inside the new circle.

If the object is clicked slightly outside, an outward-pointing arrow appears.

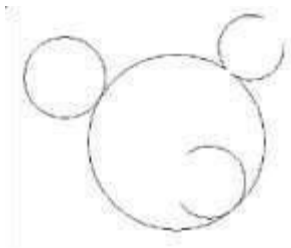


Select the second tangent object.

If the object is clicked slightly inside, an inward-pointing arrow appears.



Select the third object. The tangent circle is created.



Circle_Tangent to Arc or Curve

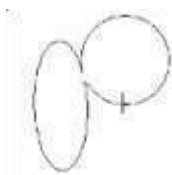
Creates a circle tangent to an arc, circle, or ellipse.

Select the existing arc, circle, or ellipse to which you want the circle to be tangent.



Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.)

The circle will be created from the point you selected on the tangent object.



Move the mouse to position the circle, either inside or outside the tangent object.



Circle_Tangent to Line

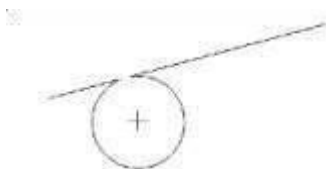
Creates a circle tangent to a line.

Select the existing line to which you want the circle to be tangent.



Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.)

The circle will be created from the point you selected on the line.



Move the mouse to position the circle, on either side of the line.



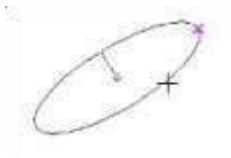
Ellipse_Axis, End

Creates an ellipse by the endpoints of the major axis, and an endpoint of the minor axis.

Select the endpoints of the major axis, or enter the major axis length and angle in the Inspector Bar.



Set the length of the minor axis, which is always perpendicular to the major axis. You can also enter the axis length in the Inspector Bar.



Local menu option:

Center: Enables you to set the center of the ellipse, which is the same as using the **Ellipse: Center** tool.



Ellipse_Center

Creates an ellipse by its centerpoint, major axis and minor axis.

Select the centerpoint of the ellipse.

Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



Set the length of the minor axis, which is always perpendicular to the major axis.
You can also enter the axis length in the Inspector Bar.



Local menu option:

Rotation: Sets the minor axis by rotation from the horizontal axis, rather than perpendicular from the major axis.



Ellipse_Ellipse (Standard)

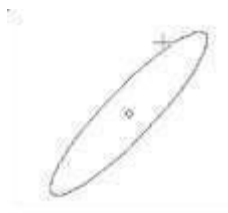
Creates a tilted ellipse.

Select the centerpoint of the ellipse.

Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



Set the length of the minor axis, which is always perpendicular to the major axis.
You can also enter the axis length in the Inspector Bar.



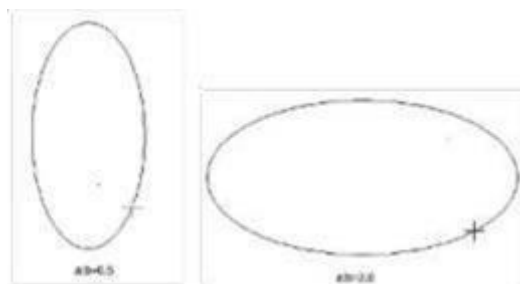
Ellipse_Fixed Ratio

Creates an ellipse by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (Lock this value if you want to repeat it later, otherwise the field reverts back to the default.)

Select the centerpoint of the ellipse.

Move the mouse to size the ellipse.



Tip: The fixed ratio for an "IsoCircle" (an isometric circle on a cube) is 1.73 (the square root of 3).

Ellipse, Elliptical Arc

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Creates an elliptical arc by defining the major and minor axes.

Click the endpoints of the major axis, or enter the major axis length and angle in the Inspector Bar.



Set the length of the minor axis, which is always perpendicular to the major axis. You can also enter the axis length in the Inspector Bar.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar. You can also enter the included angle.



Local menu options:

Center: Enables you to define the major axis from the center.



Rotation: Enables you to define the minor axis at an angle from the major axis.



Ellipse, Elliptical Arc, Fixed Ratio

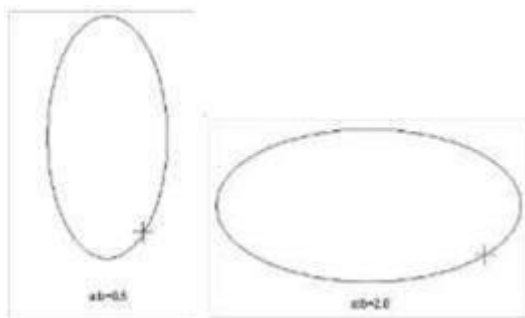
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Creates an elliptical arc by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

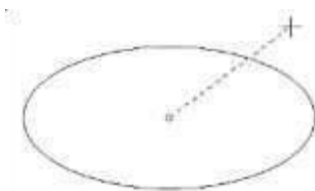
Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (If you want to repeat this value later then lock it, otherwise the field reverts back to the default.)

Select the centerpoint of the ellipse.

Move the mouse to size the ellipse.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Ellipse, Rotated Elliptical Arc

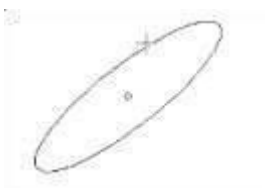
Creates a tilted elliptical arc.

Select the centerpoint of the ellipse.

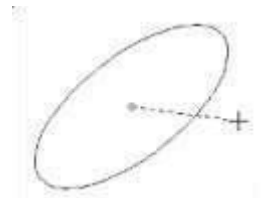
Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



Set the length of the minor axis, which is always perpendicular to the major axis. You can also enter the axis length in the Inspector Bar.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



4.4. Arc

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Arc

Ribbon UI Menu:

Most of the **Arc** tools are identical to **Circle/Ellipse** tools. After the initial circle is created, the arc is cut from it by defining the start and end angles.

You can display the **Arc** toolbar by right-clicking in any toolbar area and selecting **Arc**.



Start and end angles can be changed with the **Edit Tool** (node editing). See [Editing Circles, Ellipses, and Arcs](#)

Arc-Center and Radius

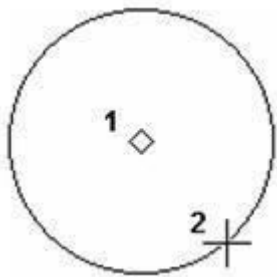


Default UI Menu: Draw/Arc/Center and Radius

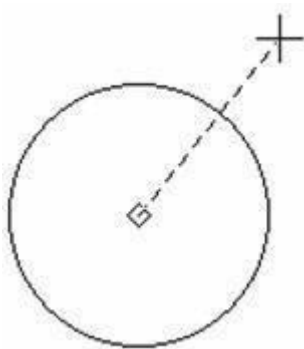
Ribbon UI Menu:

Creates an arc by defining its center, a point on its circumference, and the start and end angles.

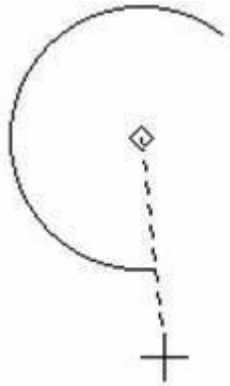
Select the arc centerpoint and a point on the circumference of the circle from which the arc will be cut. You can also specify the radius, diameter, or circumference in the Inspector Bar.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** (you can change this to clockwise with the Angle options) to draw the arc, or enter the end angle or arc length in the InspectorBar.



Arc-Concentric



Default UI Menu: Draw/Arc/Concentric

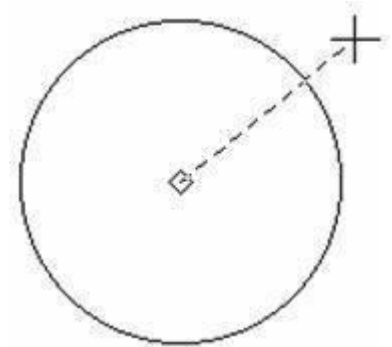
Ribbon UI Menu:

Creates arcs that share the same center point.

Create the first arc by selecting the centerpoint, then moving the mouse and clicking to define its size. You can also enter the radius, diameter, or circumference in the Inspector Bar.

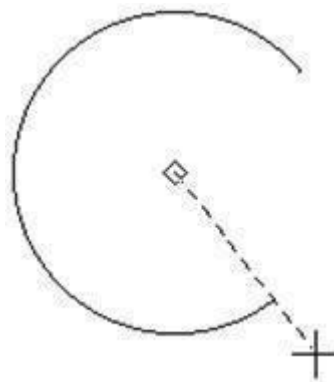
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A dotted line appears from the centerpoint. Move the cursor to set the arc start



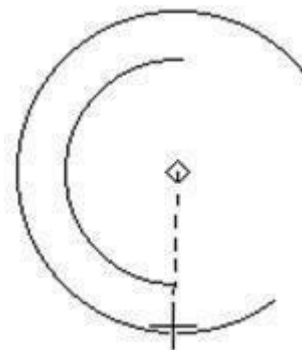
angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc



length in the Inspector Bar.

Create the next arc the same way. The centerpoint is already defined, you just



need to define the size, and start and end angles.

5. Repeat these steps to draw multiple concentric arcs.

Double-click to finish, select **Finish** from the local menu or Inspector Bar, or press Alt+F.

Arc-Double Point

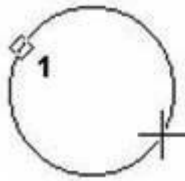


Default UI Menu: Draw/Arc/Double Point

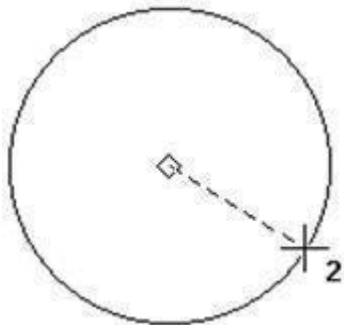
Ribbon UI Menu:

Creates an arc by defining two endpoints of its diameter.

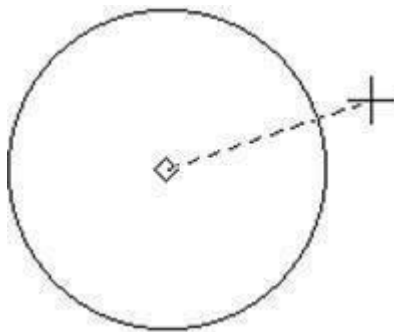
1. Select a point on the circumference of the circle from which the arc will be cut.



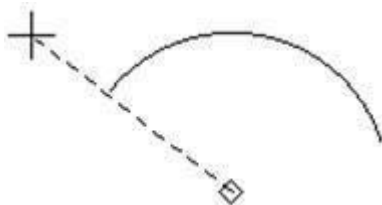
Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc-Elliptical



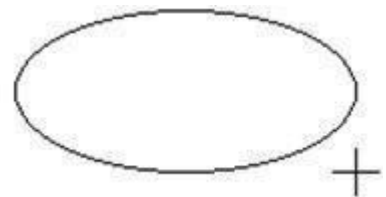
Default UI Menu: Draw/Ellipse/Elliptical Arc

Ribbon UI Menu:

Creates an elliptical arc by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal.

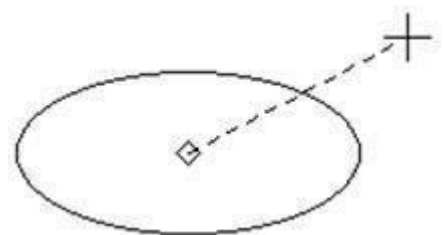
Select the first corner of the bounding rectangle.

Define the diagonally opposite corner of the bounding rectangle, or enter the



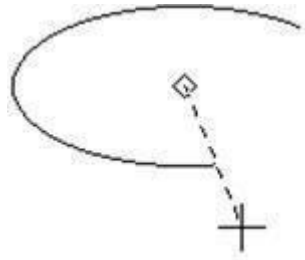
major and minor axis lengths in the Inspector Bar.

A dotted line appears from the centerpoint. Move the cursor to set the arc start



angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc



length in the Inspector Bar.

Arc-Elliptical Fixed Ratio



Default UI Menu: Draw/Ellipse/Elliptical Fixed Ratio Arc

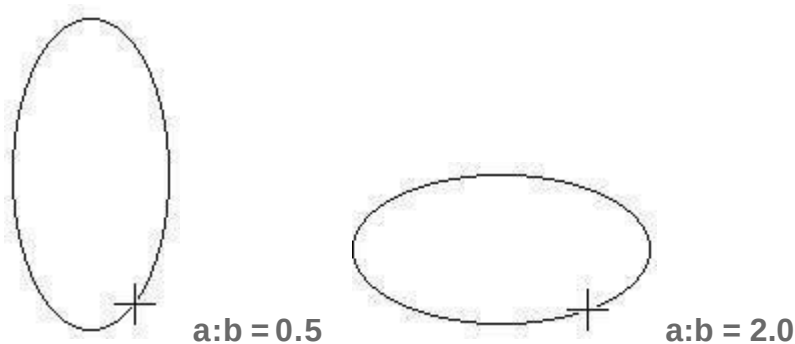
Ribbon UI Menu:

Creates an elliptical arc by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

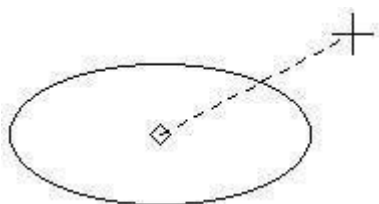
Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (If you want to repeat this value later then lock it, otherwise the field reverts back to the default.)

Select the centerpoint of the ellipse.

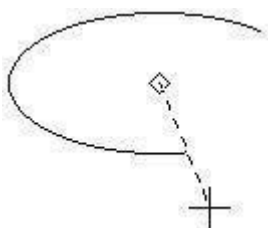
Move the mouse to size the ellipse.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc-Rotated Elliptical



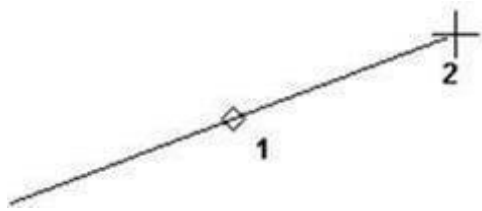
Default UI Menu: Draw/Ellipse/Rotated Elliptical Arc

Ribbon UI Menu:

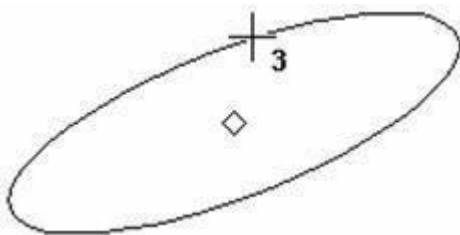
Creates a tilted elliptical arc.

Select the centerpoint of the ellipse.

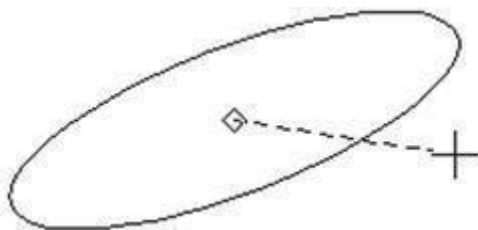
Move the mouse to set the length and angle of the major axis, or enter the major axis length and angle in the Inspector Bar.



Set the length of the minor axis, which is always perpendicular to the major axis. You can also enter the axis length in the Inspector Bar.

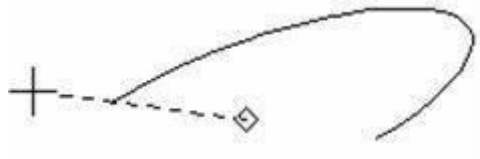


A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



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Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc-Start, End, Included



Default UI Menu: Draw/Arc/Start / End / Included

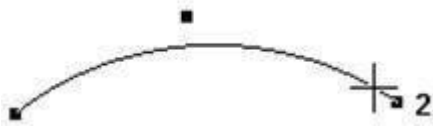
Ribbon UI Menu:

Creates an arc by defining the start point, the endpoint, and a point on the arc.

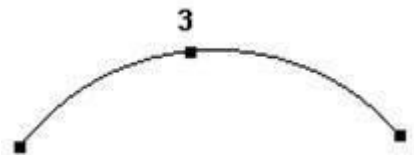
Select the start point.



Select the endpoint.



3. Select a point through which the arc will pass.



Arc-Start, Included, End



Default UI Menu: Draw/Arc/Start / Included / End

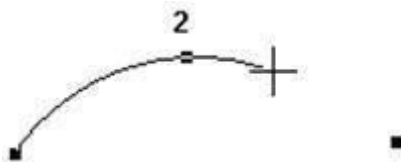
Ribbon UI Menu:

Creates an arc by defining the start point, a point on the arc, and the endpoint.

Select the start point.



Select a point through which the arc will pass.



3. Select the endpoint.



Arc-Tangent to 2 Entities



Default UI Menu: Draw/Arc/Tangent to 2 Entities

Ribbon UI Menu:

Creates an arc tangent to two objects - lines, arcs, circles, or ellipses. You can select the arc segment that will remain on either side of the tangent points.

1. Select the first tangent object, close to the point of tangency.

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2. Select the second tangent object, close to the point of tangency.

Size the circle with the mouse; the arc will be cut from this circle. Or enter the radius, diameter, or circumference in the Inspector Bar.

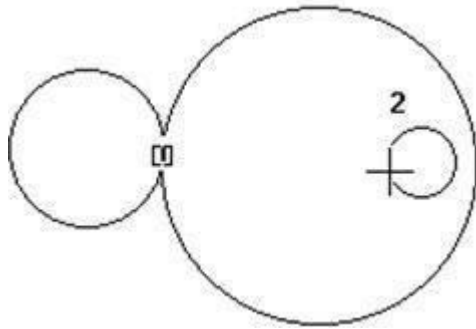
Move the cursor to either side of the arc to determine the arc segment that will remain.

Tangent Arc of Fixed Size

By default, **Tangent to 2 Entities** creates the smallest circle that touches two objects, and the arc is cut from this circle.

To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.

2. Select the second tangent object.



3. The fixed-sized tangent circle, then arc, is created.

Arc-Tangent to 3 Arcs



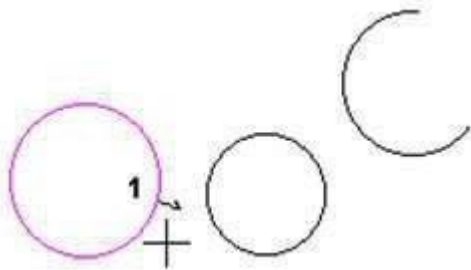
Default UI Menu: Draw/Arc/Tangent to 3 Arcs

Ribbon UI Menu:

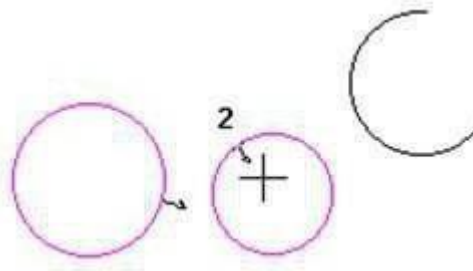
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Creates an arc tangent to three arcs or circles. You can specify whether the existing arcs will be inside or outside the new arc.

Select the first tangent object. Click slightly outside the object to keep it outside the new circle. Click inside if you want the tangent object to be inside the new circle. If the object is clicked slightly outside, an outward-pointing arrow appears.



Select the second tangent object. If the object is clicked slightly inside, an



inward-pointing arrow appears.

Select the third object. The tangent circle is created. The arc will be cut from this

circle.

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A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc

length in the Inspector Bar.

Arc-Tangent to Arc

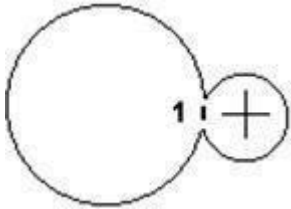


Default UI Menu: Draw/Arc/Tangent to Arc or Curve

Ribbon UI Menu:

Creates an arc tangent to an arc, circle, or ellipse.

Select the existing arc, circle, or ellipse to which you want the arc to be tangent. The tangent circle, from which the arc will be cut, extends from the point you selected.



Select the centerpoint of the tangent circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.). The circle will be created from the point you selected on the tangent object.

3. Move the mouse to position the circle, either inside or outside the tangent object.

A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

Arc-Tangent to Entities



Default UI Menu: Draw/Arc/Tangent to Entities

Ribbon UI Menu:

Creates an arc tangent to three objects (rectangles, polygons, arcs, etc.)

1 Select the first tangent object.

2 Select the second tangent object.

3Select the third tangent object. The tangent circle is created. The arc will be cut from this circle.

4A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

5Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

Local menu option:

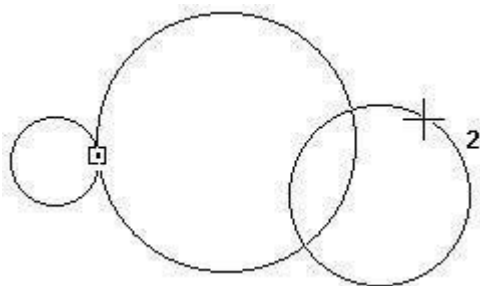
Through point: Makes the tangent circle pass through a specific point. In the example below, the line endpoint was selected, rather than the line itself.

Tangent Arc of Fixed Size

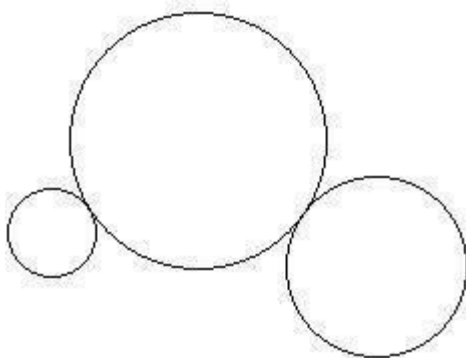
By default, **Tangent to Entities** creates a circle that touches three objects, and the arc is cut from this circle.

To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.

2. Select the second tangent object, close to the point of tangency.



3. The fixed-sized tangent circle is created.



Arc-Tangent to Line

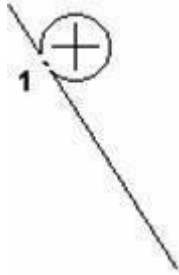


Default UI Menu: Draw/Arc/Tangent to Line

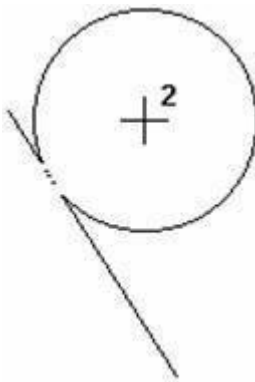
Ribbon UI Menu:

Creates an arc tangent to a line.

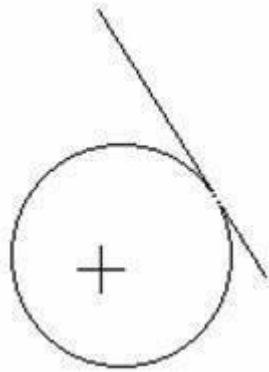
Select the existing line to which you want the arc to be tangent. The tangent circle, from which the arc will be cut, extends from the point you selected.



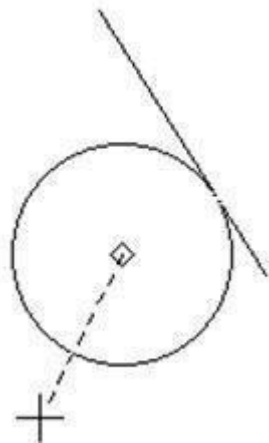
Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.) The circle will be created from the point you selected on the line.



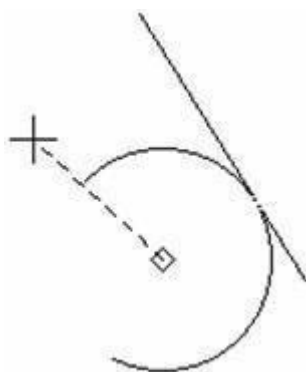
3. Move the mouse to position the circle, on either side of the line.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

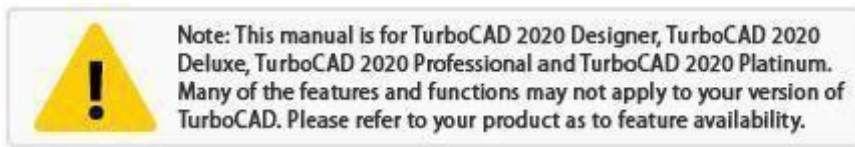


Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc tools for the LTE Workspace

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There are a specific set of tools designed to work in concert with the LTE Workspace In the LTE Workspace 2D tools behave with slight differences to standard TurboCAD methods.

They allow use of dynamic input.

They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

You can complete drop the tool by pressing the either the Enter or Space keys.

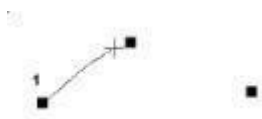
You can re-activate the tool by pressing the Space Key again.

Note: LTE Workspace is not a feature of TurboCAD Deluxe

Arc_3 Points

Creates an arc by defining the start point, a point on the arc, and the endpoint. This is a "generic" arc tool, because the local menu options enable you to define the arc in a number of different ways.

Select the start point.



Select a point through which the arc will pass.



Select the endpoint.



Local Menu Options:

These options are available after the first point of the arc is placed:

Center: Before you click the arc start point, you can select **Center** to define the center point first. You can then click the center point and start point.



○ **Angle:** Rather than clicking the end point, you can select the **Angle** or

Chord Length local option.



End: With this option, after you click the arc start point, you can select **End**

to click the end point.



Radius: Then you can click the center point, or use the **Angle**, **Direction**,

or **Radius** local option.



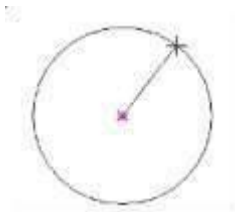
Arc_Center, Start, Angle

Note: You can use the local menu options to create this arc differently.

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Creates an arc by defining its center, a point on its circumference where the arc starts, and the arc angle.

Select the arc centerpoint and a point on the circumference of the circle where the arc will start. You can also specify the Radius and Start Angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the Angle in the Inspector Bar.



Arc_Center, Start, End

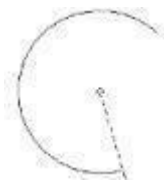
Note: You can use the local menu options to create this arc differently.

Creates an arc by defining its center, a point on its circumference where the arc starts, and the end point.

Select the arc centerpoint and a point on the circumference of the circle where the arc will start. You can also specify the Radius and Start Angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the End Angle in the Inspector Bar.

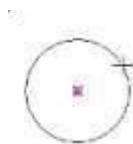


Arc_Center, Start, Length

Note: You can use the local menu options to create this arc differently.

Creates an arc by defining its center, a point on its circumference where the arc starts, and the chord length.

Select the arc centerpoint and a point on the circumference of the circle where the arc will start. You can also specify the Radius and Start Angle in the Inspector Bar.



Move the cursor to draw the arc, or enter the Length in the InspectorBar.

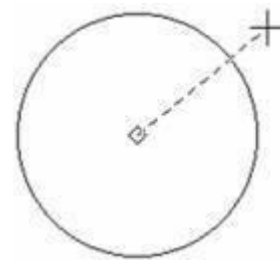


Arc_Concentric

Creates arcs that share the same centerpoint.

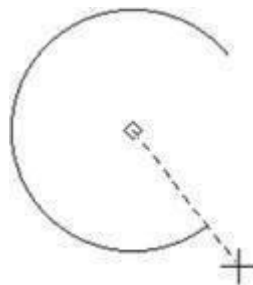
Create the first arc by selecting the centerpoint, then moving the mouse and clicking to define its size. You can also enter the radius, diameter, or circumference in the Inspector Bar.

A dotted line appears from the centerpoint. Move the cursor to set the arc start



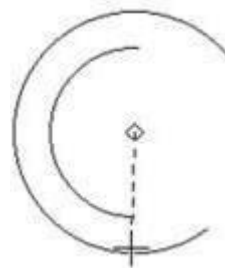
angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc



length in the Inspector Bar.

Create the next arc the same way. The centerpoint is already defined, you just



need to define the size, and start and end angles.

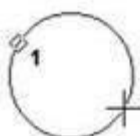


Double-click to finish, select **Finish** from the local menu or Inspector Bar, or press Alt+F.

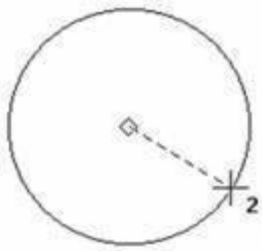
Arc_Double Point

Creates an arc by defining two endpoints of its diameter.

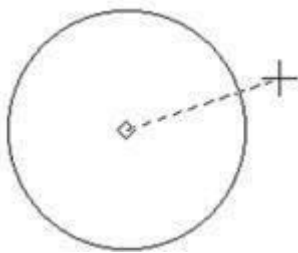
Select a point on the circumference of the circle from which the arc will be cut.



Select the point on the opposite end of the diameter line, or enter the radius, diameter or circumference, and angle of the diameter line in the Inspector Bar.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc_Elliptical

Creates an elliptical arc by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal.

1. Select the first corner of the bounding rectangle.

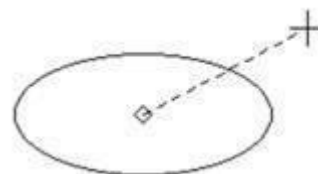
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Define the diagonally opposite corner of the bounding rectangle, or enter the



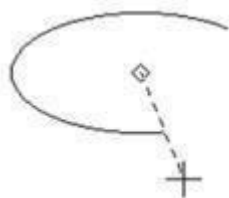
major and minor axis lengths in the Inspector Bar.

A dotted line appears from the centerpoint. Move the cursor to set the arc start



angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc



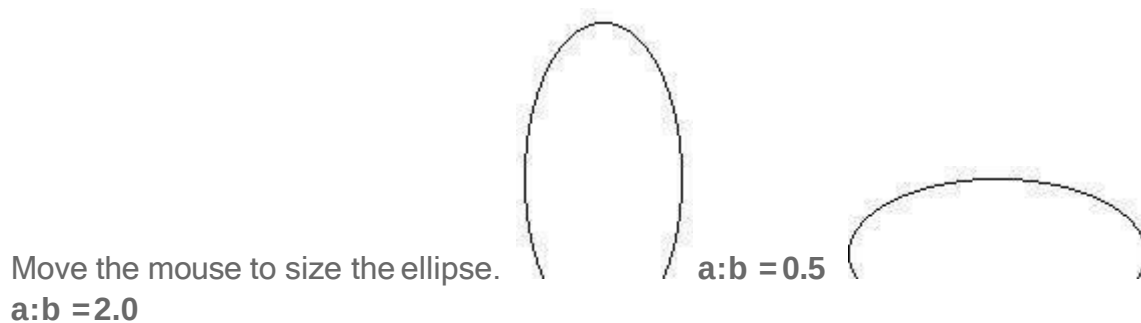
length in the Inspector Bar.

Arc_Elliptical Fixed Ratio

Creates an elliptical arc by specifying its aspect ratio - the ratio of the major axis to the minor axis. The axes of the ellipse are orthogonal.

Enter the aspect ratio in the **a:b** ratio field on the Inspector Bar. (If you want to repeat this value later then lock it, otherwise the field reverts back to the default.)

Select the centerpoint of the ellipse.



A dotted line appears from the centerpoint. Move the cursor to set the arc start

angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc

length in the Inspector Bar.

Arc_Rotated Elliptical

Creates a tilted elliptical arc.

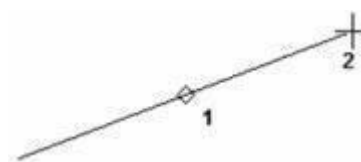
Select the centerpoint of the ellipse.

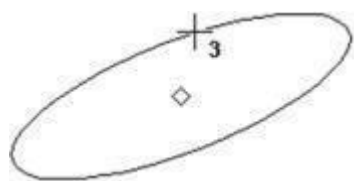
Move the mouse to set the length and angle of the major axis, or enter the major

axis length and angle in the Inspector Bar.

Set the length of the minor axis, which is always perpendicular to the major axis.

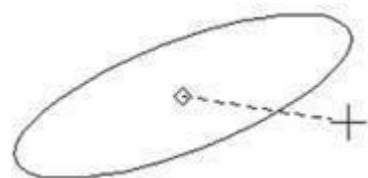
You can also enter the axis length in the Inspector Bar.





A dotted line appears from the centerpoint. Move the cursor to set the arc start

angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc

length in the Inspector Bar.



Arc_Start, Center, Angle

Creates an arc by defining its start point, the center point, and the angle. Note: You can use the local menu options to create this arc differently.

Select the arc start point and then the center point. You can also specify the Radius and Angle in the Inspector Bar.

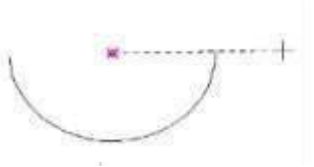
Arc_Start, Center, End

Creates an arc by defining its start point, the center point, and the end point. Note: You can use the local menu options to create this arc differently. .

Select the arc start point and then the center point. You can also specify the Radius and Angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the End Angle in the Inspector Bar.



Arc_Start, Center, Length

Creates an arc by defining its start point, the center point, and the chord length. Note: You can use the local menu options to create this arc differently.

Select the arc start point and then the center point. You can also specify the Radius and Angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the Length in the Inspector Bar.



Arc_Start, End, Angle

Creates an arc by defining its start point, end point, and angle. Note: You can use the local menu options to create this arc differently.

Select the arc start point and then the end point. You can also specify the Length and Angle in the Inspector Bar.



Move the cursor to draw the arc, or enter the Angle in the Inspector Bar.



Arc_Start, End, Direction

Creates an arc by defining its start point, end point, and tangent direction. Note: You can use the local menu options to create this arc differently.

Select the arc start point and then the end point. You can also specify the Length and Angle in the Inspector Bar.



Move the cursor to set the tangent direction of the arc, or enter the Angle in the Inspector Bar.



Arc_Start, End, Included

Creates an arc by defining the start point, the endpoint, and a point on the arc.

Select the start point.



Select the endpoint.



Select a point through which the arc will pass.



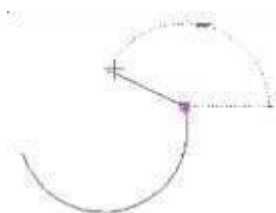
Arc_Start, End, Radius

Creates an arc by defining its start point, end point, and arc radius. Note: You can use the local menu options to create this arc differently.

Select the arc start point and then the end point. You can also specify the Length and Angle in the Inspector Bar.



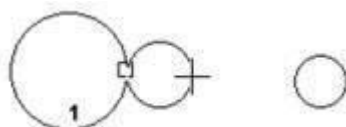
Move the cursor to set the radius arc, or enter the Radius in the Inspector Bar.



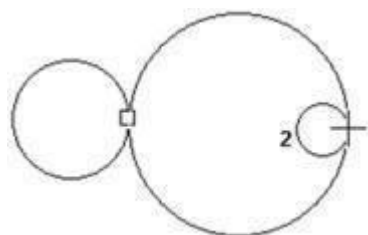
Arc_Tangent to 2 Entities

Creates an arc tangent to two objects - lines, arcs, circles, or ellipses. You can select the arc segment that will remain on either side of the tangent points.

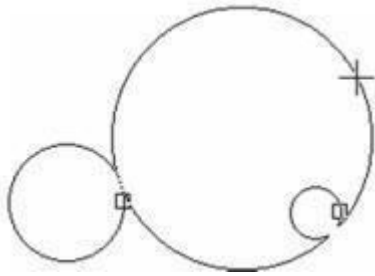
Select the first tangent object, close to the point of tangency.



Select the second tangent object, close to the point of tangency.



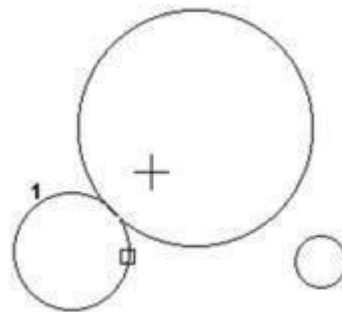
Size the circle with the mouse; the arc will be cut from this circle. Or enter the radius, diameter, or circumference in the Inspector Bar.



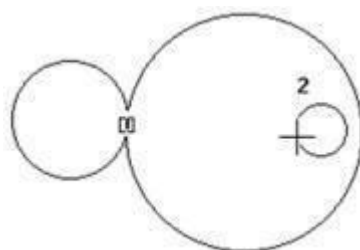
Move the cursor to either side of the arc to determine the arc segment that will remain.

Tangent Arc of Fixed Size

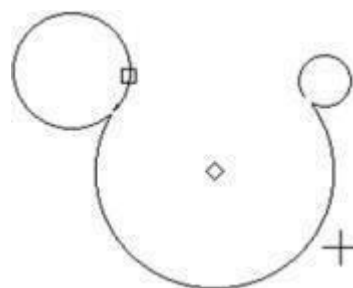
By default, **Tangent to 2 Entities** creates the smallest circle that touches two objects, and the arc is cut from this circle. To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you



move the cursor, the circle size remains constant.



Select the second tangent object.

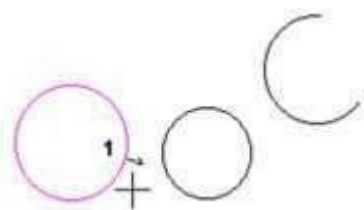


The fixed-sized tangent circle, then arc, is created.

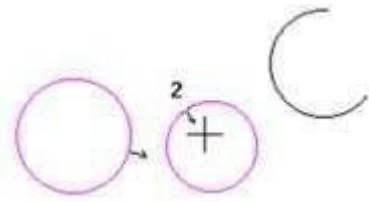
Arc_Tangent to 3 Arcs

Creates an arc tangent to three arcs or circles. You can specify whether the existing arcs will be inside or outside the new arc.

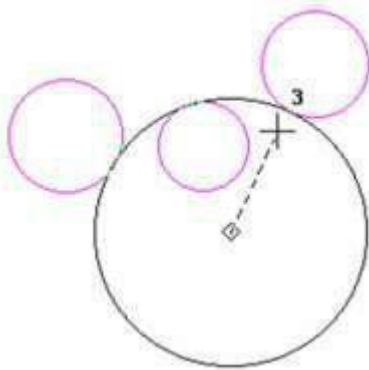
Select the first tangent object. Click slightly outside the object to keep it outside the new circle. Click inside if you want the tangent object to be inside the new circle. If the object is clicked slightly outside, an outward-pointing arrow appears.



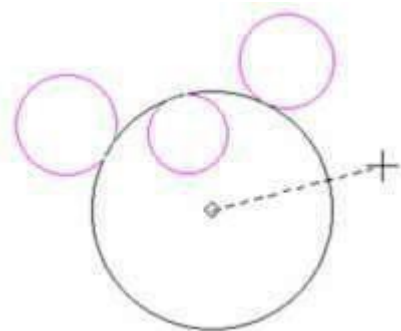
Select the second tangent object. If the object is clicked slightly inside, an inward-pointing arrow appears.



Select the third object. The tangent circle is created. The arc will be cut from this circle.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

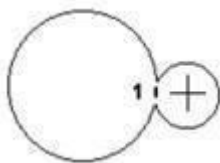


Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.

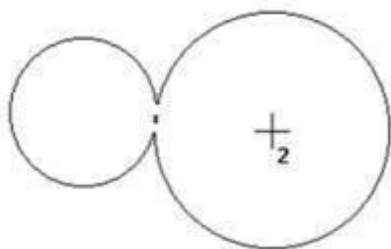
Arc_Tangent to Arc

Creates an arc tangent to an arc, circle, or ellipse.

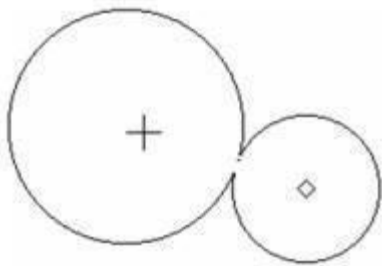
Select the existing arc, circle, or ellipse to which you want the arc to be tangent. The tangent circle, from which the arc will be cut, extends from the point you selected.



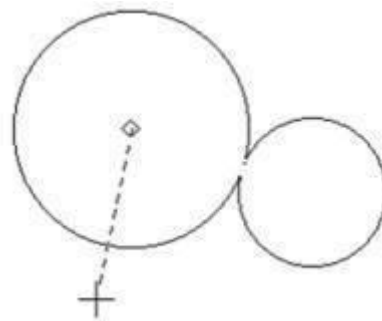
Select the centerpoint of the tangent circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.) The circle will be created from the point you selected on the tangent object.



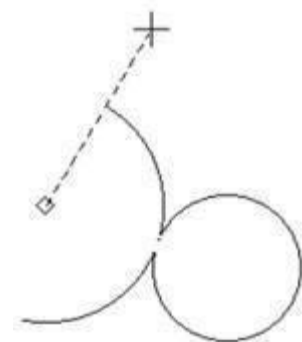
3. Move the mouse to position the circle, either inside or outside the tangent object.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



Arc_Tangent to Line

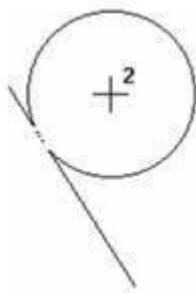
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Creates an arc tangent to a line.

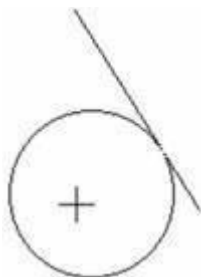
Select the existing line to which you want the arc to be tangent. The tangent circle, from which the arc will be cut, extends from the point you selected.



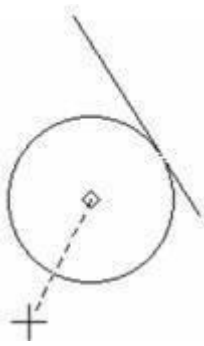
Select the centerpoint of the circle, or enter the radius, diameter, or circumference in the Inspector Bar. (If you use the Inspector Bar and then press Enter, you will not be able to reposition the circle.) The circle will be created from the point you selected on the line.



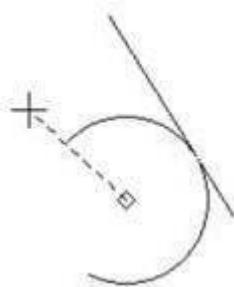
3. Move the mouse to position the circle, on either side of the line.



A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



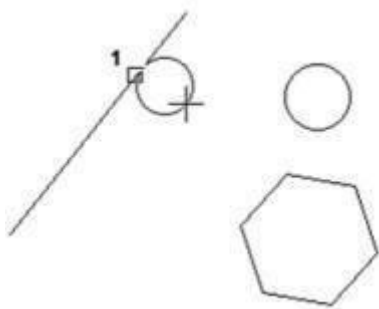
Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



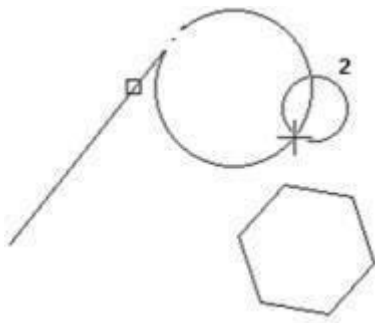
Arc_Tangent to Entities

Creates an arc tangent to three objects (rectangles, polygons, arcs, etc.)

1. Select the first tangent object.

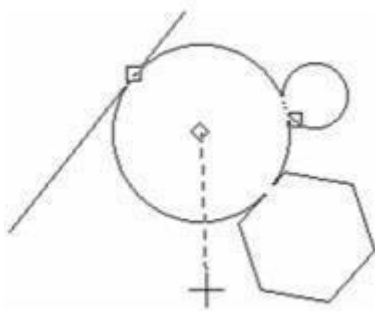


2. Select the second tangent object.

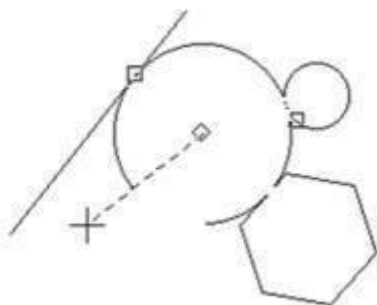


Select the third tangent object. The tangent circle is created. The arc will be cut from this circle.

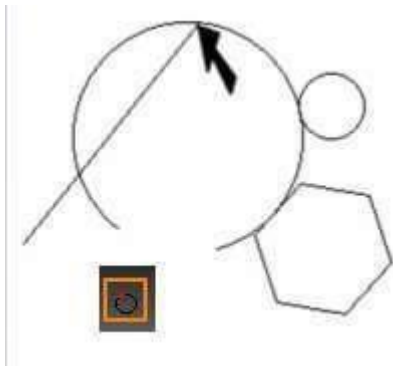
A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.



Move the cursor **counterclockwise** to draw the arc, or enter the end angle or arc length in the Inspector Bar.



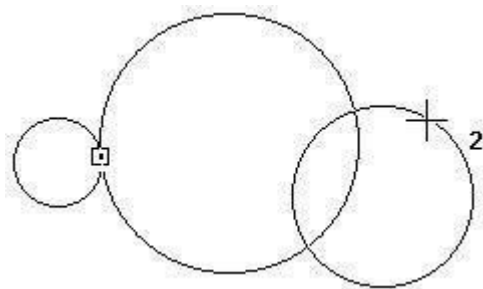
Local menu option: Through point: Makes the tangent circle pass through a specific point. In the example below, the line endpoint was selected, rather than the line itself.



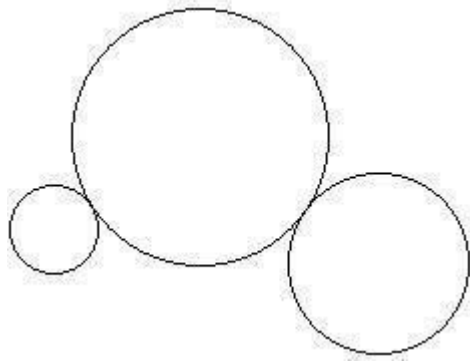
Tangent Arc of Fixed Size

By default, **Tangent to Entities** creates a circle that touches three objects, and the arc is cut from this circle. To fix the size of the circle, enter the radius, diameter, or circumference in the Inspector Bar and lock it, but do not press Enter. When you move the cursor, the circle size remains constant.

Select the second tangent object, close to the point of tangency.



The fixed-sized tangent circle is created.



4.5. Curves

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Curve

Ribbon UI Menu:

Tools for drawing splines, Bezier curves, sketches, and revision clouds.

You can display the **Curve** toolbar by right-clicking in any toolbar area and selecting **Curve**.



Autoshape

Default UI Menu: Draw/Curve/Autoshape

Ribbon UI Menu:

The Autoshape tool allows you to quickly create standard shapes just by sketching. The sketched shape will automatically recompose into a standard object depending on the shape of the sketch.

The objects include:

Lines, Polylines, Rectangles, Polygons, Triangles

Arcs, Circles, Ellipses,

Ellipse / Circles

If the sketch closely approximates a elliptical path an ellipse will be generated.



If the radii of the computed ellipse are almost equal, then a circle is drawn.



If the sketch closely approximates a circular path but is open an arc will be generated.

Line

A single quick dash will result in a line.



Polyline

If the end points are closely located the resulting polyline will be closed.

If the angle between the nearest computed line segments is close to 90 degrees, right angle is created.

A closed polyline with 3 points becomes a triangular Polygon.

A closed polyline with 4 points becomes a rectangular Polygon.

Bezier Curve

If the sketch does not approximate a regular shape a Bezier curve will be created. The curve can be closed or open.

Curve Properties

Curve properties can be set on the **Curves** page of the **Properties** window.

Number of segments between adjacent control points: Curves are composed of many small line segments drawn between control points. A high number of segments will yield a smoother curve; a low number will make the curve appear more jointed.

High number of segments



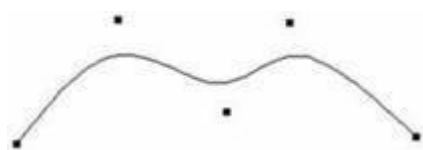
Low number of segments



Bezier Curve



Spline Curve



Bezier Curve

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Options: If the curve is a Bezier curve, the following options are available:

Show additional control points: Additional control points will be displayed when **Edit Node** mode is active (see [Edit Tool](#) and [Editing Splines and Bezier Curves](#)). This also enables the **Keep the Curve Smooth** option.

Regular control points in Edit Node mode



Additional control points in Edit Node mode

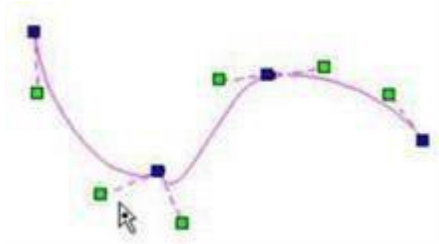


Keep the curve smooth: If checked, the additional control points at each node remain linear, so that no sharp corners can be created at a node. If not checked, you can move each additional node independently.

Keep Curves Smooth - additional nodes remain linear



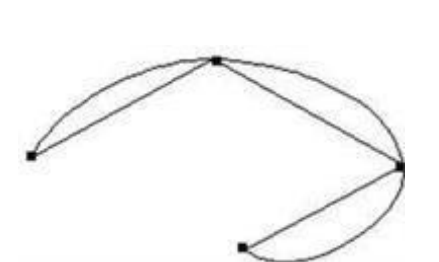
Keep Curves Smooth disabled



Warning: Modifying tools such as **Split** or **Trim** may alter a curve drastically, because the curve loses the influence of deleted control points.

Spline Curve Options: The degree value creates a spline of n-th order, which affects the spline smoothness.

Show Frame: If checked, a polyline frame will be displayed connecting the curve's control points. This provides visual feedback about how the curve is drawn.



Revision Cloud



Default UI Menu: Draw/Curve/Revision Cloud

Ribbon UI Menu:

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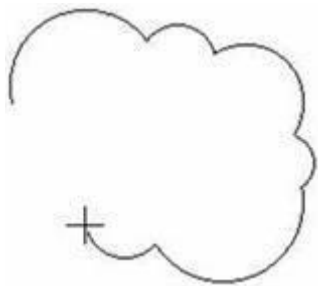
Creates uniform revision clouds. These are most commonly used in Paper Space, but the tool is available in Model Space as well.

Before selecting the start point, select **Number of segments** from the local menu. The default number is 1, and each segment consists of two arcs.

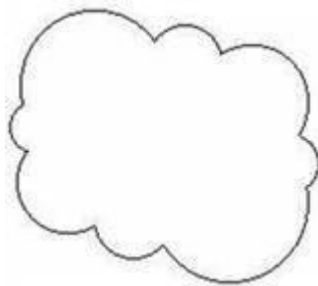
Select the start and endpoints of the first segment.



Continue selecting points for more segments.



When finished, select **Finish** (Alt+F) from the local menu, or select **Close** (Alt+C) to join the endpoints.



Sketch

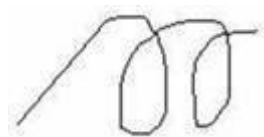


Default UI Menu: Draw/Curve/Sketch

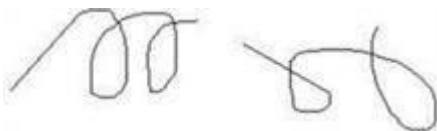
Ribbon UI Menu:

Creates a freehand drawing.

Press and hold the mouse button to draw a freehand curve. Release the mouse when finished.



As long as the tool is still active, you can continue drawing new curves.



Splines and Bezier Curves

These three tools each use a series of points to create a curve.

Spline by Control Points



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Default UI Menu: Draw/Curve/By Control Points

Ribbon UI Menu:

Spline by Fit Points



Default UI Menu: Draw/Spline by Fit Points

Ribbon UI Menu:

Bezier



Default UI Menu: Draw/Curve/Bezier

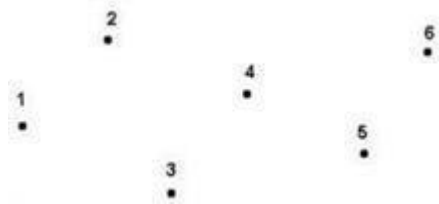
Ribbon UI Menu:

For **Spline by Control Points**, the points act as a guide for the curve; the spline does not actually pass through all of the points. For **Spline by Fit Points** and **Bezier**, the curve does pass through each point. These two tools produce similar results; the main difference is the algorithm behind them, and how they are edited.

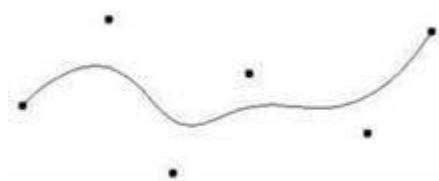
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Note: To create splines in 3D, [3D Spline by Fit Points](#)

The three curves will be shown for this series of points:



Select the control points in the desired order. You can also enter the length and angle between points in the Inspector Bar. After selecting the last point, select **Finish** from the local menu or press Alt+F. You can also double-click the last point.



Spline by Control Points

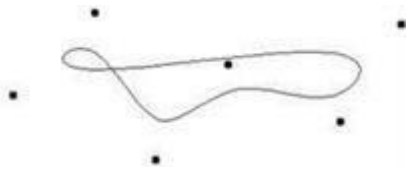


Spline by Fit Points

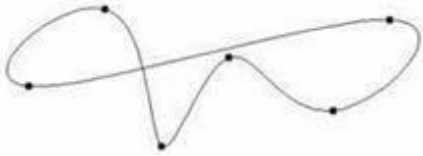


Bezier

If you want to close the curve, select **Close** from the local menu instead of **Finish**.



Closed Spline by Control Points



Closed Spline by Fit Points

Once created, you can use the **Edit Tool** to change the shape of a spline and add knots.

See [Editing Splines and Bezier Curves](#)

Tip: You can change a spline into a Bezier curve, and vice-versa, by opening the **Properties** window and editing the **Curves** page. If you change a Bezier curve into a spline, it will be a **Spline by Control Points**. You can also convert 2D objects into a Bezier curve - see [Convert to Curve](#)

4.6. Double Line

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Double Line

Ribbon UI Menu:

Tools for drawing double (parallel) lines and linear objects. The **Double Line** tools are essentially similar to the **Line** tools, but there are no tangent tools. You can display the **Double Line** toolbar by right-clicking in any toolbar area and selecting **Double Line**.



Note: If you want to use double lines to create representations of walls, there is a special tool for this purpose. See [Wall Tools](#)

LTE T ype Objects

In the LTE Workspace 2D tools behave with slight differences to standard TurboCAD methods.

- They allow use of dynamic input.

- They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

- They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

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You can complete drop the tool by pressing the either the Enter or Space keys.

You can re-activate the tool by pressing the Space Key again.

All LTE Workspace Double Line tools work exactly as the equivalent TC tools except as noted in the above list.

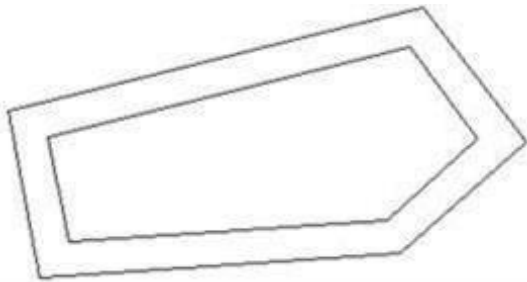
Double Line-Irregular Polygon



Default UI Menu: Draw/Double Line/Irregular Polygon

Ribbon UI Menu:

Creates a double-line polygon (closed) with irregular sides and angles. For the interaction, see [Irregular Polygon](#)



Double Line-Parallel



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Default UI Menu: Draw/Double Line/Parallel

Ribbon UI Menu:

Creates a double line parallel to a existing line. For the interaction, see [Parallel](#)



Double Line-Perpendicular

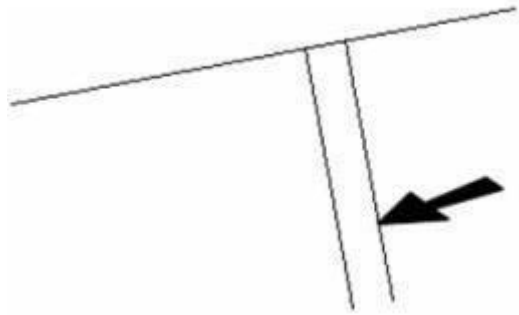


Default UI Menu: Draw/Double Line/Perpendicular

Ribbon UI Menu:

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Creates a double line perpendicular to an existing line. For the interaction, see [Perpendicular](#)



Double Line-Polygon



Default UI Menu: Draw/Double Line/Polygon

Ribbon UI Menu:

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Creates a regular (equal-length sides) double-line polygon. For the interaction, see [Polygon](#)

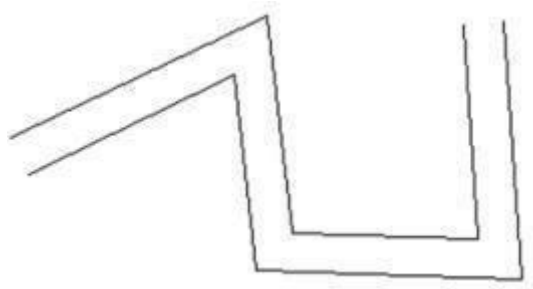
Double Line-Polyline



Default UI Menu: Draw/Double Line/Polyline

Ribbon UI Menu:

Creates a series of connected straight double line segments (no arc segments) that comprise one object. For the interaction, see [Polyline](#)

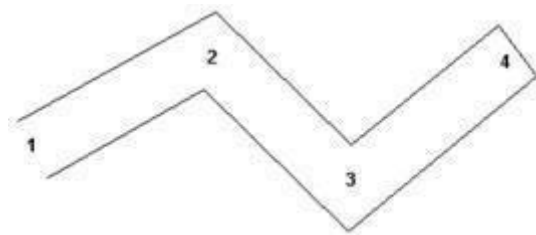


Double Line Properties

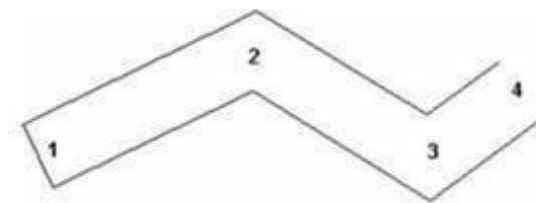
Double Lines have unique properties that you can set in the **Double Line** tab of the **Properties** window.

Separation: The distance between the lines. If you stretch or scale a double line, this distance does not change.

End Caps: Closes the endpoints of the double line. **End Cap at End**



End Cap at Start



Tip: Another way to create end caps is to use the **Edit Tool** (node editing), right-click on the end node and select **Closed Double End**.

Double Line-Rectangle



Default UI Menu: Draw/Double Line/Rectangle

Ribbon UI Menu:

Creates an orthogonal double-line rectangle by defining two diagonally opposite corners. For the interaction, see [Rectangle](#)



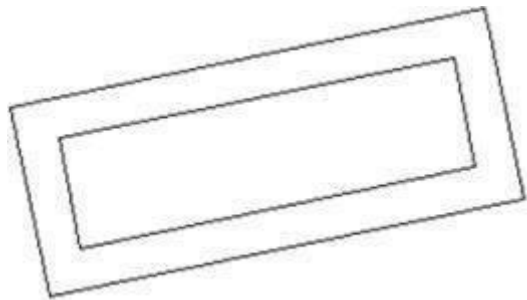
Double Line-Rotated Rectangle



Default UI Menu: Draw/Double Line/Rotated Rectangle

Ribbon UI Menu:

Creates a non-orthogonal double-line rectangle. For the interaction, see [Rotated Rectangle](#)



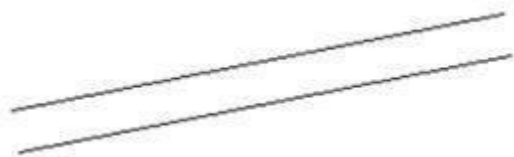
Double Line-Single Line



Default UI Menu: Draw/Single

Ribbon UI Menu:

Creates one double line. For the interaction, see [Single Line](#)



4.7. Multi Line

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Multi Line

Ribbon UI Menu:

Tools for drawing multi lines and linear objects.

Multi Line objects consist of two or more parallel lines, created as one object. The **Multi Line** tools are essentially similar to the **Line** tools, except that there are no tangent tools. You can display the **Multi Line** toolbar by right-clicking in any toolbar area and selecting **Multi Line**.



LTE Type Objects

In the LTE Workspace 2D tools behave with slight differences to standard TurboCAD methods.

- They allow use of dynamic input.

- They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

- They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

- You can complete drop the tool by pressing the either the Enter or Space keys.

- You can re-activate the tool by pressing the Space Key again.

All LTE Workspace MultiLine tools work exactly as the equivalent TC tools except as noted in the above list.

Multi Line-Irregular Polygon



Default UI Menu: Draw/Multi Line/Irregular Polygon

Ribbon UI Menu:

Creates a multi line polygon (closed) with irregular sides and angles. For the interaction, see [Irregular Polygon](#)

Multi Line-Parallel

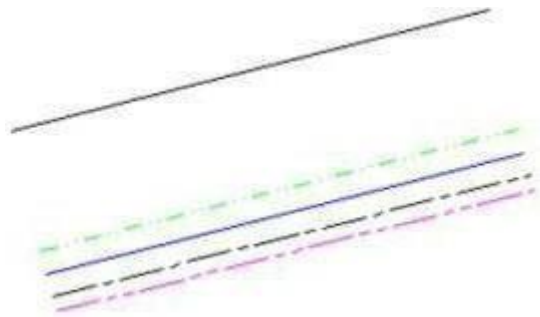


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Default UI Menu: Draw/Multi Line/Parallel

Ribbon UI Menu:

Creates a multi line parallel to a existing line. For the interaction, see [Parallel](#)



Multi Line-Perpendicular

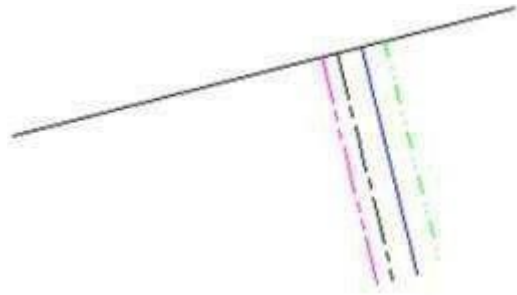


Default UI Menu: Draw/Multi Line/Perpendicular

Ribbon UI Menu:

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Creates a multi line perpendicular to an existing line. For the interaction, see [Perpendicular](#)



Multi Line-Polygon

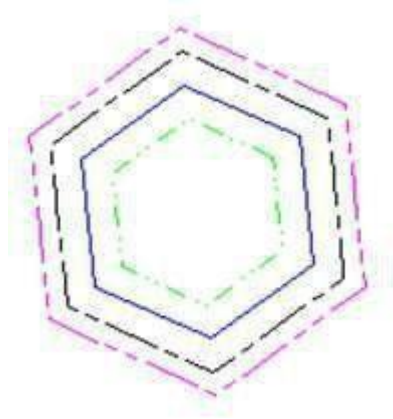


Default UI Menu: Draw/Multi Line/Polygon

Ribbon UI Menu:

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Creates a regular (equal-length sides) multi line polygon. For the interaction, see [Polygon](#)



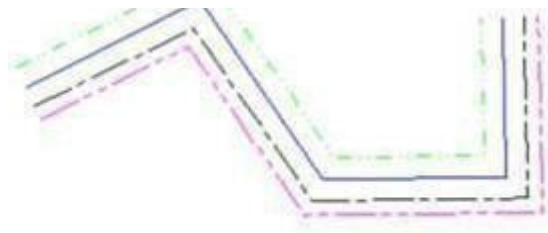
Multi Line-Polyline



Default UI Menu: Draw/Multi Line/Polyline

Ribbon UI Menu:

Creates a series of connected straight multi line segments (no arc segments) that comprise one object. For the interaction, see [Polyline](#)



Multi Line Properties

Five pages of the **Properties** window contain options relevant to Multi Lines.

Multiline Properties

General properties for multi lines:

Offset Scale: Controls the overall width of the multi line.

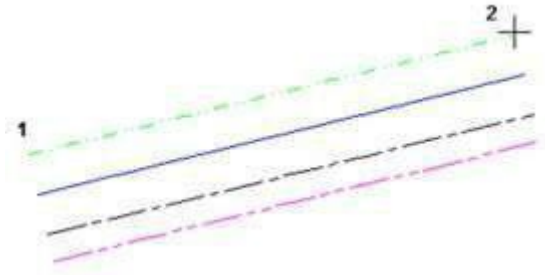
Justification: Sets the alignment of the multi line nodes. Left and right are determined by "facing" toward the multi line start point. No justification is the default; the line is aligned to a zero offset, relative to the line offsets defined in the **Line Elements** page.

No Justification(Indicated line has zero offset)

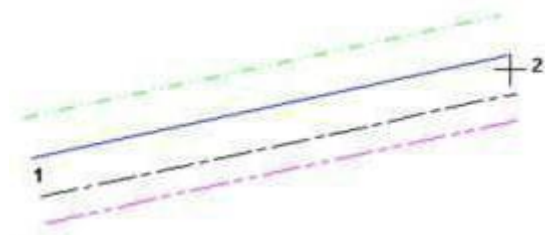


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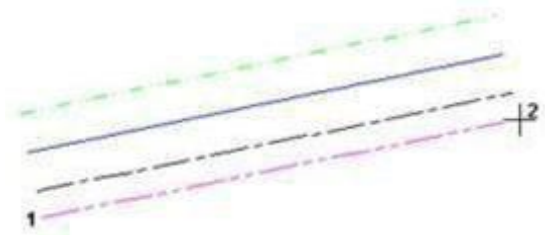
Left Justification



Center Justification



Right Justification



Dash and line width scaling: Determines whether the width of the multi lines will be scaled or will remain the same size when zoomed. This setting also applies to the size of the elements in the pen's dot-and-dash pattern.

Device: The pen's width and pattern sizes are defined in device units (the monitor or printer). If you zoom the line width and pattern size will not change on the screen.

World: The pen's width and pattern sizes are defined in by the drawing spaces units. If you zoom the line width and pattern size will change in accordance with the zoom factor.

Device Width: The pen's pattern size is defined by the drawing spaces units, and pen's width is defined in device units (the monitor or printer). If you zoom the line patterns size will change, but the pen width size will not change.

Line Elements Properties

Properties for each line comprising the multi line.

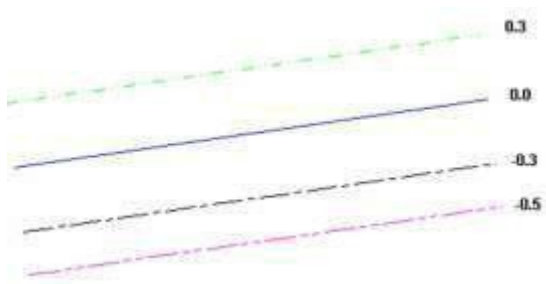
Pat. The exact look of each pattern depends on the setting of ***Dash scale** and width on the **Multiline** page.

Dash Scale: The scale of the dot-and-dash pattern.

Color: Select a color for each pen. **By Layer** and **By Block** are available.

Width: Specify the width of each line. Zero width means one screen pixel, and will print at one unit of the printer's available resolution (a 300 dots-per-inch printer will print a zero width line at 1/300".)

Offset: For each line, define its offset value. Use **Add** to create new lines, **Delete** to remove lines.



Start Cap and End Cap Properties

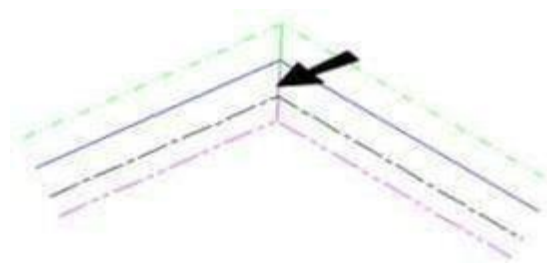
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Start and end caps are used to close the ends of the multiline. The options are the same for start caps and end caps.

Joints Properties

Use joints to show breaks at each multi line segment.

Show: Choose to show or hide the joints.



Note: Multilines can be used to trim objects, and can be trimmed themselves. However when trimming multilines you need to click near the center line of the trimmed multiline.

Multi Line-Rectangle



Default UI Menu: Draw/Multi Line/Rectangle

Ribbon UI Menu:

Creates an orthogonal multi line rectangle by defining two diagonally opposite corners.



For the interaction, see [Rectangle](#)

Multi Line-Rotated Rectangle



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Default UI Menu: Draw/Multi Line/Rotated Rectangle

Ribbon UI Menu:

Creates a non-orthogonal multi line rectangle. For the interaction, see [Rotated Rectangle](#)

Multi Line-Single Line



Default UI Menu: Draw/Single

Ribbon UI Menu:

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Creates one multi line. For the interaction, see [Single Line](#)



4.8. Inserting Objects from Other Files

(Available in all TurboCAD Variants)

Default UI Menu: Insert/Picture, Insert/File, Insert/Object

Ribbon UI Menu:

You can insert data from external files into your TurboCAD drawing. For example, you can insert a Word document or a *.gif picture file, or even data from another TurboCAD (or other CAD format) file.

You can insert files, pictures, and OLE objects. These tools can be accessed from the **Insert** menu, or from the **Insert** toolbar, displayed by right-clicking in any toolbar area and selecting **Insert**.



Inserting a File

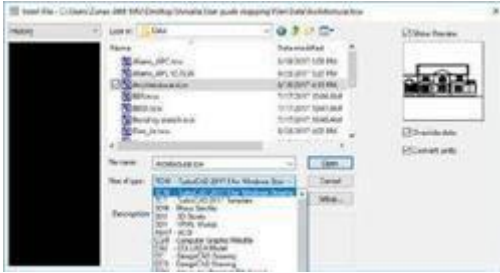


Default UI Menu: Insert/File

Ribbon UI Menu:

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Inserts a TurboCAD (or other CAD format) file into the current drawing.



Note: The contents of the inserted file are embedded, not linked. See [Inserting an OLE Object](#)

Inserting Partial Data from a File

The right side of the **Open** window contains a box in which you can select what you want to add from the selected file.

For example, you can insert all objects ("graphics") but omit their properties or line style. Or you could insert a file's layers without the objects that they contain.

You can use **File / Extract To** to save only selected components in your drawing to a *.tcw file. This is useful for creating file templates.

Override data: If any items with the same name are found in the target file, they will be replaced by the items in the source file.

Convert units: It converts the units of the file according to current units.

Inserting an OLE Object

Default UI Menu: Insert/Object

Ribbon UI Menu:

Warning

OLE is limited in functionality and no longer fully supported by Microsoft. As a result OLE objects are not recommended, results will vary. While this feature is provided for legacy reasons it is not supported and will see no future development.

Enables you to insert OLE objects from other Windows applications. You can edit an OLE object from within TurboCAD using the features and commands of the Windows application in which the object was created (the source application). For example, you can insert a Microsoft Word application into your drawing and edit the text or graphics using Microsoft Word tools.



Select **Insert / Object**, and the **Insert Object** window appears.

Create New: Inserts an OLE object from another application. Select the application from the list of all applications found on your system. The object will appear in the drawing. The object is editable in its source application

Create from File: Inserts an object from another application that has already been created in its source application. Enter the path and name of the file, or click **Browse** to find it.

Display as Icon: Places an icon representing the object into the drawing.

Tip: You can also drag the file into the drawing via Windows Explorer.

Embedded and Linked OLE Objects

OLE objects that you insert can be either linked or embedded.

Embedded objects become an integral part of your file, and take up more space in the file than a linked object. Embedded objects will move with the file if you transfer it. If you make changes to the OLE object's source file, these changes will **not** affect the embedded object.

Linked objects are references to the file on which they were based, and they retain their connection to that file. If you make changes to the OLE object's source file, the changes **will** affect the linked object.

If you plan to use multiple copies of an OLE object, using links will reduce your file size. Linked objects are referenced by their path. If you move the TurboCAD file, you will need to move all linked objects as well. If you move the source file of a linked object to

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another folder, Windows will not be able to locate the file, and you will need to edit the link.

Warning: Norton AntiVirus Script Blocking can cause problems with OLE. For example, Microsoft Word documents are inserted as a picture, not a Word document.

Paste Special

Enables you to choose how to insert (paste) into your drawing an object that was previously cut (Ctrl+X) or copied (Ctrl+C) to the clipboard. The object on the clipboard can be from TurboCAD, or it can come from another application.

The options available in the **Paste Special** window depend on the type of object in the clipboard, and the object's source application. For example, if there is a Microsoft Word document on the clipboard, you can paste it as:

Word Document: Inserts as an OLE object which you can edit within TurboCAD using the Microsoft Word application.

Picture: Inserts as an image

Text : Inserts as a TurboCAD text object.

When inserting an object as OLE, select **Paste Link** to link the object to its source file or **Paste** to embed the object (break the link with the source file). Check **Display As Icon** to insert the OLE object as an icon. You can then click **Change Icon** to select another icon.

Note: Only two types of data can be inserted into TurboCAD as an editable TurboCAD drawing object: text (inserted as a TurboCAD text object), and TurboCAD drawing objects that have been copied to the clipboard from TurboCAD. All other clipboard objects will be pasted either as OLE objects or as pictures (either bitmaps or metafiles).

Editing Inserted OLE Objects

Edits an OLE object in its source application.

Select an OLE object in your drawing. This object may appear as a document or a bitmap, or it may appear as an icon, depending on the options you chose when you inserted the OLE object.

Double-click on the object to open it for editing.

Editing Links to OLE Objects

You can set each linked object so that the update happens automatically, or so that the update must be done manually.

Links box: contains all the links in the active drawing. You can highlight multiple links by pressing Shift or Ctrl.

Update Option: Sets whether the selected links are updated automatically or manually.

Update Now: Updates the current link. This is how to update sink that are set to update manually.

Open Source: Opens the file in its source application. This also enables you to compare the linked object to the source file.

Change Source: Enables you to choose a new source for the object. Once the object is linked to the new file, its contents will change to match the new file.

Inserting a Picture

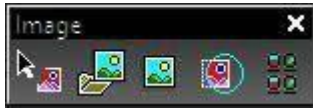


You can insert a picture from a file, from a list of images, or as an OLE object. Inserting pictures is useful, for example, if you have a picture of a site plan and want to create buildings directly on it.

Note: To create a tracing of an inserted picture, see [Tracing](#)

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As with inserting any object, the picture will be inserted "on top" of existing objects. To adjust the object stack, you can use the **Bring to Front** and **Send to Back** tools. See [Stacking Objects](#). The **Image** toolbar can be opened by right-clicking on any toolbar area and selecting **Image**.



Note: Nearly all graphic formats are raster images, meaning they are composed of dots. Even when reading vector formats, such as *.wmf, the image is converted into a raster format for TurboCAD purposes.

Image Manager

Default UI Menu: Insert/Picture/From Image List

Ribbon UI Menu:

Enables you to manipulate all the raster images in the current drawing. You can insert files directly from this window using **From Image List**.

The toolbar at the top of the window can be used to change the image list as thumbnails, list, or detailed list. **Image List:** Lists all images inserted into the drawing. The list can be sorted by name, status, or path. To edit a name, select it and then click it again. You can then enter the new name. **New:** Adds a new image file to the list. **Delete:** Deletes an image file from the list. **Embed:** Saves the image file in the drawing as an embedded object, and not as a reference to an external file. **Unload:** Unloads image data from work memory. **Reload:** Reloads image into work memory. **Properties:** Opens the Image Visualization Properties* window, which is useful for setting the image quality for large images.

OpenGL: Limits image output to the maximum OpenGL texture size (1024x1024 is supported by most hardware), and resizes image to a square view. For example, a 400

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x 300 image will be resized to 512 x 512, thereby losing accuracy. As you increase the zoom, the image may become blurry. **GDI+:** Supports large images without resizing, maintaining high accuracy. Not all hardware can render this way efficiently, so this method may be slower than OpenGL. **Filtering quality:** Controls scaled or rotated images. **Nearest point:** The pixel with coordinates nearest to the desired pixel value is used, resulting in a set of large squares when viewed closely. **Bilinear:** A weighted average of a 2 × 2 area of pixels surrounding the desired pixel is used. This is the most common filtering algorithm, adding smoothness in large scales. **Trilinear:** A more exact extension of **Bilinear** and is practically indiscernible. But it is unsupported with OpenGL.

Note: These parameters are not file-specific; they affect visualization of all images in all TurboCAD files.

Inserting a Picture from a File



Default UI Menu: Insert/Picture/From File

Ribbon UI Menu:

Using this tool you can insert any type of graphic - .gif, *.jpg, etc. **The file is embedded and the picture itself cannot be edited. You can, however, select it and perform any of the *Select Edit commands (see [Select Edit](#)).** All inserted files are listed in the **Image Manager**.

Change the workplane, if necessary. The picture will be inserted onto the current workplane (though it can be moved later).

Select **Insert / Picture / From File** and browse to select the desired picture.

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Select two points to determine the size of the inserted picture. Use the local menu option **Keep Aspect Ratio** if you want to maintain the picture's proportional size.



Note: The first point you select corresponds to the upper-left corner of the picture. You can move, rotate, or scale the picture using any of the **Select Edit** tools.

Inserting a Picture from a List of Images



Default UI Menu: Insert/Picture/As Object

Ribbon UI Menu:

Invokes the **Image Manager**, from which you can select a picture to insert.

Inserting a Picture as an OLE Object

Inserts a picture as a linked OLE object. This means that the picture can be edited in its source application and all changes will be reflected in the TurboCAD drawing as well.

File types that are allowed for OLE insertion are *.wmf (Windows metafile) and *.dib (device-independent bitmap).

Select **Insert / Picture / as Object** and browse to select the desired picture.

Tip: You can also drag the file into the drawing from Windows explorer.

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The picture is inserted in its original size. You can resize, move, or rotate it by using the **Select Edittools**.

To edit the picture, you can double-click on it to open it in its source application.

To select a previously inserted picture, click along an edge instead of inside the picture.

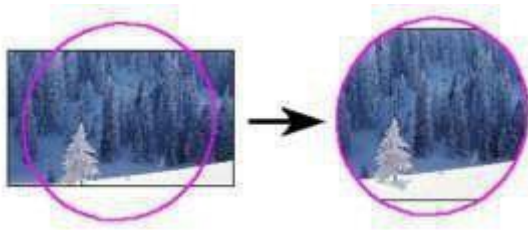
Clipping Images



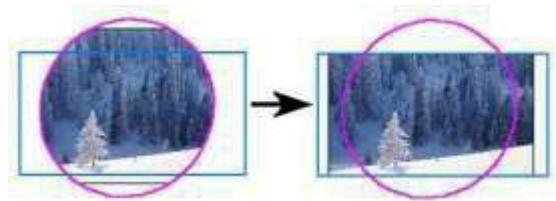
Enables you to use a border to clip a raster image. Borders must be either a closed polyline, or a closed circle or ellipse.

Select the image you want to clip.

Select the new image outline. The image will be clipped to the outline.



The image still retains its original size, even if it is not fully displayed. Therefore,



you can clip it again using larger borders.

Image Properties

You can view certain properties of all currently inserted picture by selecting an image and opening its **Properties** (see [Object Properties](#)) to the **Image** page.

The toolbar at the top of the window can be used to change the image list as thumbnails, list, or detailed list. **Show Image:** Shows or hides the images. **Show Clipped:** Uncheck to hide the clipping contour and show the original image. Check the box to show a clipped image.

From the Selection Info Palette you can access the additional properties of:

Brightness

Contrast

Saturation

Transparency

These can change the appearance of the image. Here are some examples.

4.9. Organizational Chart

(Available in Platinum)

Default UI Menu: Tools/Wizards/Org Chart Wizard

Ribbon UI Menu:

The Organizational Chart wizard is a separate module of TurboCAD that allows you to create organizational charts simply by entering text into an outline. The chart is created as a *.csv (comma-separated value) file, which can be read as a spreadsheet.

In the first page of the wizard, choose whether to open an existing *.csv file or to create a new one.

The second page of the wizard is where you enter or edit the chart data. The title and each item in the chart can have up to three lines of text (**Text 2**, and **Text 3** fields and tabs), and for each entry in the chart you can specify text alignment, box shape, etc.

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Tip: The format of each item is taken by default from the format of the title (but can be changed). If you want the entire chart to have a particular format, change the settings while the **Title** field is still activated. Similarly, the format of a child item takes the format of its parent.

Use these icons to add a new item directly below (**Create Sibling**) or in one level (**Create Child**) from the selected item. The "trash can" icon deletes a selected item.



Use these arrows to adjust the hierarchical levels of existing items. You can also drag an item over another item to create a parent-child relationship.



Use these arrows to change the order of items.



In the last page of the wizard, set the chart orientation and separator line properties.

Finally, choose whether to insert the chart into the drawing, and whether to save it.

Warning: If you do not save your chart, you will not be able to edit it later in the wizard.

The inserted chart is not one single object, rather it is composed of standard TurboCAD drawing objects. Each item is a group that can be exploded into its component parts. You can edit and redesign your chart using any of the geometric editing tools.

4.10. Hyperlinks

Hyperlinks

(Available in all TurboCAD Variants)

Hotkey: Ctrl+H



Default UI Menu: Insert/Hyperlink

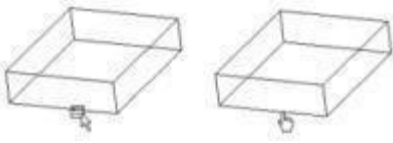
Ribbon UI Menu:

A hyperlink is attached to an object in your drawing, and enables you to jump to a specified file (for example, a drawing or text document) on your hard disk or company network, or to open a URL. Another example of hyperlink usage is to display pictures of certain items, such as a bracket or fastener. You can create a hyperlink to a **.jpg or *.gif file that will appear in the Internet Palette when clicked with the *Pick Hyperlink tool**. You can also display text or spreadsheet files to provide descriptions of the clicked object.

Note: You can also assign and edit a hyperlink via the General tab of an object's Properties (see [Properties Window](#)).

Place the cursor over the desired object. The cursor will be shaped like a screen. (If a hyperlink has already been defined for the object, the cursor will be shaped like a hand.).

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The Define Hyperlink dialog appears.

Specify the following parameters for your hyperlink:

Hyperlink Name: Assigns a name (alias) to the hyperlink.

Named Location: Opens the hyperlinked file to a specific location, such as a named view. If you use a named view as the hyperlink, that view will be shown when the hyperlink is opened.

Based Path: Displays the default base path for all relative path hyperlinks in the current drawing.

File Path or Web: The location of a desired file on your disk or company network, or a URL. You can browse for either files or websites.

Go To: Invokes the target referenced by the hyperlink. If the target is a TurboCAD file, that file will open in another window. A **.doc f ile will open in Microsoft Word, and a URL will open in the web browser.**

Set Path: Relative and Absolute: are available if a file path or web address is provided. A relative path is relative to the path to the active drawing.

Name and Path or Address: Displays a list of hyperlinks used in the current drawing.

To use any of these hyperlinks for the selected object, double-click the hyperlink's **Name** field.

Local menu options:

Edit the Hyperlink: Use this option on an object that already has a defined hyperlink.

This opens the **Define Hyperlink** window, in which you can change any parameters of the link.

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Open the Hyperlink: Brings up the defined hyperlink.



4.11. Stellated Polygon

Stellated Polygon



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/Stellated Polygon

Ribbon UI Menu:

Creates a parameterized star shaped (stellated) polygon.

To create a Stellated Polygon:

Specify the number of teeth

Specify the Internal and External radii

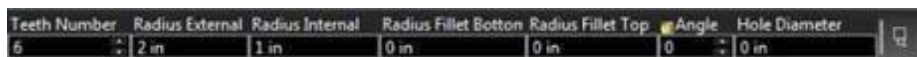
If to wish the polygon to be rounded specify the Radius Fillets. (Skip this if you want to set the polygon as a wave)

If you want there to be a hole in the polygon specify the Hole Diameter. Then:

Click to place the center point for the polygon.

Move the cursor and click to specify the rotation Angle for the polygon Or:

Specify the angle value in the Angle field and click the Enter key.



Options: Stellated as Wave - this option sets an automatic wave curvature for the polygon. When this option is on the Fillet values are ignored.

Note: You cannot extrude, revolve, loft or perform similar 3D operations on Stellated polygons. You must explode them first.

Editing Stellated Polygons

You can use the Edit tool to re-arrange and modify a Stellated Polygon.

To Edit a Stellated Polygon:

- Select the polygon.

- Right click, and select Edit Tool.

- Use the node to reshape the polygon.

Note: You cannot rotate a Stellated Polygon using the standard rotation handles of the Selector tool. You must use the rotation angle of the polygon itself.

4.12. Gear Contour



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum)

Default UI Menu: Draw/Gear Contour

Ribbon UI Menu:

Creates a parameterized 2D Gear Contour.

Definitions

Diametral pitch - equals the number of teeth of the gear per inch or per centimeter of the diameter, of its pitch circle. For example, if a gear has 32 teeth and a diameter of 8 inches the diametral pitch is four teeth per inch.

Pitch circle - is the circle that passes through the contact point between two meshing gears.

Pressure angle - is the angle at a contact point between two gears. Common values are 14.5, 20 and 25 degrees. However, teeth with smaller pressure angles are weaker. Gears must have the same pressure angle in order for the teeth to mesh properly.

To create a Gear Contour:

Specify the number of teeth

Specify the Diametral Pitch.

Specify the Pressure Angle.

Specify the Angle to set the rotation of the gear.

If you want there to be a hole in the gear specify the Hole Diameter. Then:

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6. Click to place the center point for the gear.

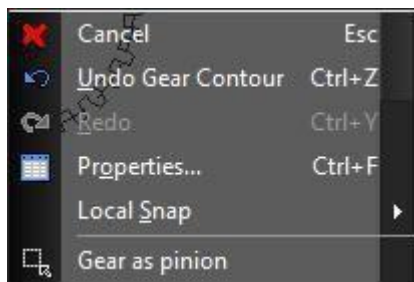
Options: Gear as Pinion - this option align a new gear with a previously created gear.

Select the Gear as Pinion option.

Click on an existing gear.

Rotate the new gear around the old gear until it is located as desired.

Click to place the gear.



Note: You cannot extrude, revolve, loft or perform similar 3D operations on gear contours. You must explode them first.

Editing Gear Contours

You can use the Edit tool to re-arrange and modify a Gear Contour.

To Edit a Gear Contour:

- Select the Gear Contour.

- Right click, and select Edit Tool.

- Use the node to reshape the Gear.

Note: You cannot rotate a Gear Contour using the standard rotation handles of the Selector tool. You must use the rotation angle of the gear itself.

4.13. Arrow Tools



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Arrow

Ribbon UI Menu:

Arrow Tools

There are four Arrow tools which allow you to quickly create arrows to annotate your drawings.



Arrow Arc

Arrow Curve

Arrow Line

Arrow Polyline

Results and Properties of using Arrow Tools:

Arrow Arc

Default UI Menu: Draw/Arrow/Arrow Arc

Ribbon UI Menu:

To draw an Arrow Arc

Select the Arrow Arc tool

Click to set the beginning point of the arc.

Click to set the end point of the arc.

Click to set the middle point of the arc.

Arrow Curve

Default UI Menu: Draw/Arrow/Arrow Curve

Ribbon UI Menu:

To draw an Arrow Curve

Select the Arrow Curve tool

Click to set the first point of the curve.

Click to set the next point of the curve. Repeat until you have set all of the points of the curve.

Right click and select Finish.

Arrow Line

Default UI Menu: Draw/Arrow/Arrow Line

Ribbon UI Menu:

To draw an ArrowLine

Select the Arrow Arc tool

Click to set the beginning point of the Line.

Click to set the end point of the Line.

Arrow Polyline

Default UI Menu: Draw/Arrow/Arrow Polyline

Ribbon UI Menu:

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Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

To draw an Arrow Polyline

Select the Arrow Polyline tool

Click to set the first point of the Polyline.

Click to set the next point of the curve. Repeat until you have set all of the points of the Polyline.

Right click and select Finish.

Arrow Properties

There are three special properties for Arrows.

Head

Tail

Size

Head and Tail: are used to specify the symbol that will appear on the ends of the arrow. You can specify a custom user defined arrowhead or tail. User defined arrow heads are based by selecting User Defined and then selecting one of your existing blocks. (i.e.

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Those in the Block Palette)

Size: Length of the arrowhead or diameter of the dot

4.14. Slots

Slots

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/Slot

Ribbon UI Menu:

These three tools allow you to create 2D slot objects.

Slot



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Draw/Slot/Slot

Ribbon UI Menu:

Creates a straight slot.

1. Define the segment start point.

Define the endpoint, or specify the length and angle in the Inspector bar.

Define the width, or specify the width or half-width in the Inspector bar.

(without rounded ends)

Options:

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Rounded Ends: Specifies that the slot is to be drawing with rounded ends. This option is on by default.

Draw as Polyline: Specifies that the slot be drawn as a polyline.

Properties:

Rounded Ends: Specifies that the slot is to be drawing with rounded ends. This option is on by default.

Trim Ends: Specifies that the ends of the slots terminate at the node points. This property is only available if Rounded ends is turned off.

Node Editing a Slot

A Slot can be node edited just like a single line segment.

Slots can be combined using the 2D Boolean tools (Add, Subtract, Intersect)

Circular Slot

Default UI Menu: Draw/Slot/Circular Slot

Ribbon UI Menu:

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Creates an circular arc slot by defining the start point, the endpoint, middle point on the arc and the slot width.

Select the start point.

Select the endpoint.



Select a point through which the arc will pass.

Define the width, or specify the width or half-width in the Inspector bar.

(without rounded ends)

Options:

Rounded Ends: Specifies that the slot is to be drawing with rounded ends. This option is on by default.

Draw as Polyline: Specifies that the slot be drawn as a polyline.

Properties:

Rounded Ends: Specifies that the slot is to be drawing with rounded ends. This option is on by default.

Trim Ends: Specifies that the ends of the slots terminate at the node points. This property is only available if Rounded ends is turned off.

Node Editing a Circular Slot

A Slot can be node edited just like an arc.

Slots can be combined using the 2D Boolean tools (Add, Subtract, Intersect)

Circular Slot by Center and Radius

Default UI Menu: Draw/Slot/Circular Slot by Center And Radius

Ribbon UI Menu:

Creates an circular arc slot by defining its center, a point on its circumference, and the start and end angles and width.

Select the arc centerpoint and a point on the circumference of the circle from which the arc will be cut. You can also specify the radius, diameter, or circumference in the Inspector Bar.

A dotted line appears from the centerpoint. Move the cursor to set the arc start angle, or enter the angle in the Inspector Bar.

Move the cursor **counterclockwise** (you can change this to clockwise with the Angle options) to draw the arc, or enter the end angle or arc length in the InspectorBar.

4. Define the width, or specify the width or half-width in the Inspector bar.

(without rounded ends)

Options:

-

Rounded Ends: Specifies that the slot is to be drawing with rounded ends.
This option is on by default.

Draw as Polyline: Specifies that the slot be drawn as a polyline.

Properties:

-

Rounded Ends: Specifies that the slot is to be drawing with rounded ends.
This option is on by default.

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Trim Ends: Specifies that the ends of the slots terminate at the node points. This property is only available if Rounded ends is turned off.

Node Editing a Circular Slot

A Slot can be node edited just like an arc.

Slots can be combined using the 2D Boolean tools (Add, Subtract, Intersect)

Selecting and Transforming Objects

Selecting and Transforming Objects

(Available in all TurboCAD Variants)

This section covers selecting, moving, copying, rotating, and scaling objects, both 2D and 3D.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

5.1. Copying Objects

(Available in all TurboCAD Variants)

Default UI Menu: Modify/Copy

Ribbon UI Menu:

The **Copy Entities** tools enable you to make single or multiple copies of selected objects, while precisely controlling the placement of the copies.

You can display the **Copy** toolbar by right-clicking in any toolbar area and selecting **Copy**.

Note: You can also make simple copies of objects in **Select Edit** (see [Copying Objects in Select Edit](#)). For other tools you can use to copy objects relative to specific points, see [Transforming](#)

The **Copy Entities** tools can be used on 2D and 3D objects. For the objects you want to use, be sure the **Selector** is set correctly (**2D**, **3D**, or both). See [2D-3D Selector](#) These tools are typically invoked when objects are already selected. You can also invoke them even if no objects are currently selected, as long as the **Select** tool is active. In this case, the local menu option **Select** can be used to select objects to copy. Close the **Select** option when all objects are selected.

Local menu options:

For all of the **Copy** tools (except Copy in Place), the following local menu options are available:



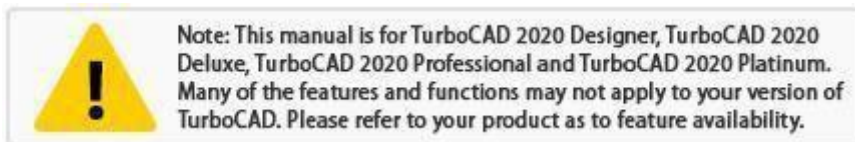
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Select: Enables you to add or remove objects from the selection set. Once the objects have been selected, deactivate this object to continue with the operation.

Repeat: Enables you to copy the selected objects again. Otherwise, the operation ends once the first operation is complete.

Make Pattern Object as Result: Creates a Pattern object as a result of using the Copy tool. In TurboCAD you can use **Auto Constraints** and to control the spacing of pattern-copied objects. See [Constraining Pattern-Copied Objects](#)

Tip: When using the **Copy Entities** tools, you will probably need to switch often between the Inspector Bar and the drawing area. You can press Tab to enter the Inspector Bar, and click once in the drawing or press Esc to return to the drawing.



Array Copy



Default UI Menu: Modify/Array/Array

Ribbon UI Menu:

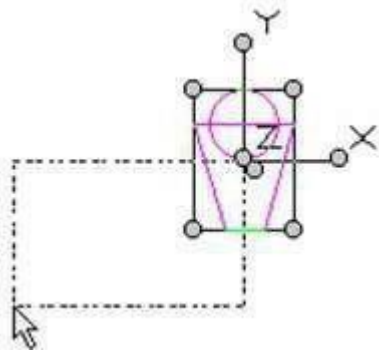
Creates copies of 2D or 3D objects into a 2D or 3D linear array, in which each copy is separated by a specified distance.

1. Select the objects you want to copy and activate **Array Copy**.

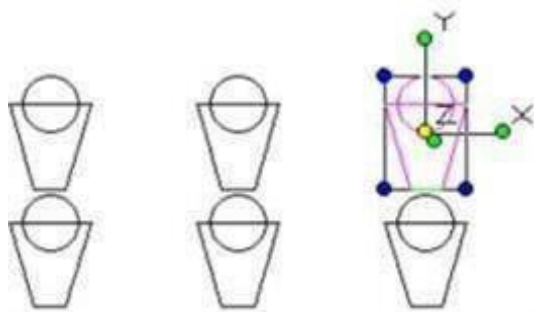
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In the Inspector Bar, enter the total number of **Rows**, **Column**, and **Levels** (number of copies in Z) that will result.

Set the distance between adjacent copies by using the mouse to define the copy vector, or enter **Step** values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.

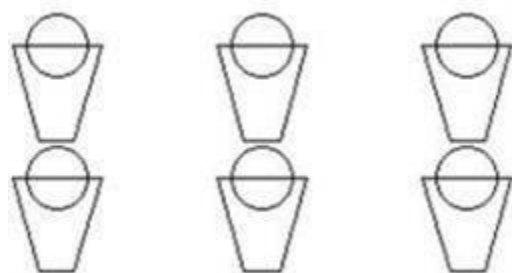


4. The copies are made after the vector has been defined.

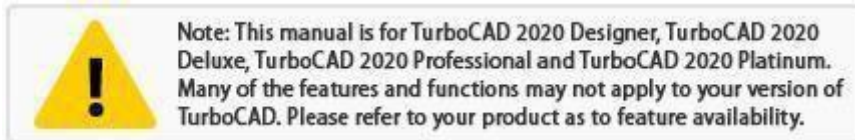


Rows = 2, Columns = 3

5. Click anywhere to exit **Select** mode, or press Esc.



Copy In Place



Default UI Menu: Edit/Copy in Place

Ribbon UI Menu:

Copy in Place creates a copy of all selected objects in exactly the same place as they are.

Select the objects you want to copy and select **Copy in Place**.

The resulting copied objects will now be the selected objects.

Fit Array Copy



Default UI Menu: Modify/Array/Fit Array

Ribbon UI Menu:

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Creates copies of 2D or 3D objects into a 2D or 3D linear array, in which a specified number of copies are evenly spaced.

Select the objects you want to copy and activate **Fit Array Copy**.

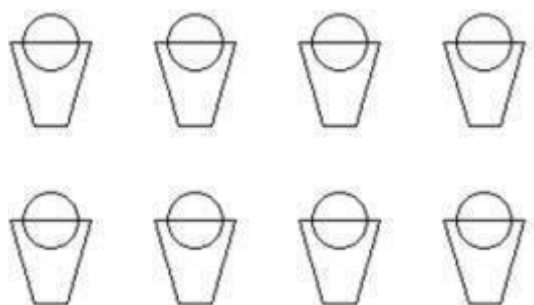
In the Inspector Bar, enter the total number of rows, columns, and levels (number of copies in Z) that will result.

Set the distance between first and last copies in each direction by using the mouse to define the entire copy array boundary, or enter **Bound** values in the Inspector Bar. The copy array is defined from the reference point of the selection set.

4. The copies are made after the array boundary has been defined.

Rows = 2, Columns = 4

5. Click anywhere to exit **Select** mode, or press Esc.



Fit Linear Copy



Default UI Menu: Modify/Array/Fit Linear

Ribbon UI Menu:

Creates copies of 2D or 3D objects along a straight line, along which a specified number of copies are evenly spaced.

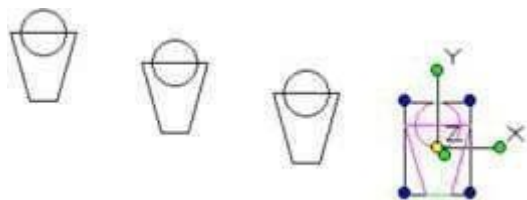
Select the objects you want to copy and activate **Fit Linear Copy**.

In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.

Set the distance between the first and last copies by using the mouse to define the copy vector, or enter size values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.

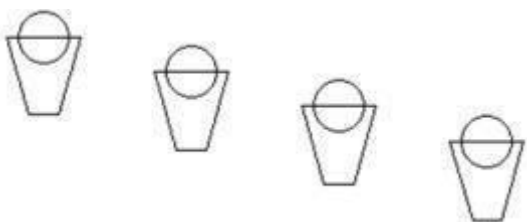


4. The copies are made after the vector has been defined.



Sets = 4

5. Click anywhere to exit **Select** mode, or press Esc.



Array Fit Linear command take into account the all the properties of the object of which linear array is being created e.g. Workplane, etc. In addition, the processing of ADT objects, i.e. File Open, House Wizard, etc. has been markedly sped up.

Example 2:

Fit Radial Copy



Default UI Menu: Modify/Array/Fit Radial

Ribbon UI Menu:

Creates copies of 2D or 3D objects along an arc, along which a specified number of copies are evenly spaced and rotated.

Select the objects you want to copy and activate **Fit Radial Copy**.

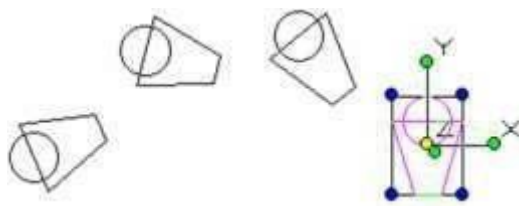
In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.

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Set the angle the first and last copies by using the mouse to define the center of the copy arc and the angle, or enter the angle in the Inspector Bar. You can enter a rotation value to set the rotation of each copy relative to the previous one. If **Rotation** = 0, the copies will not rotate at all.

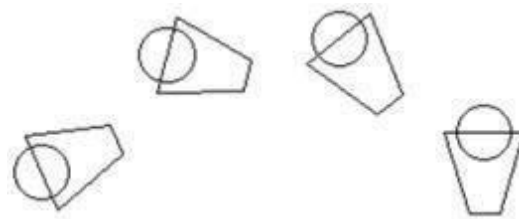
The copy arc is defined from the reference point of the selection set.

1. The copies are made after the arc has been defined.



Sets = 4

2. Click anywhere to exit **Select** mode, or press Esc.



Note: For copying in 3D, the copy arc is positioned on the current workplane. Each copy is rotated around the axis perpendicular to the workplane, passing through the reference point of the selected objects.

Mirror Copy

Default UI Menu: Modify/Copy/Mirror

Ribbon UI Menu:

Options:

Mirror along line: Creates a mirror image of an object by defining a mirror line.

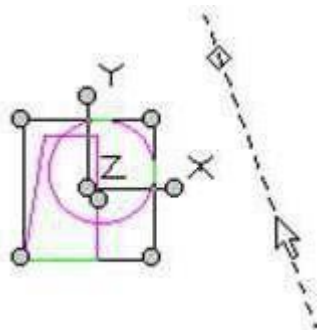
Mirror along face: Select a face plane along which Mirror operation should be performed.

Try to create single object: When selected TurboCAD will attempt to add original object and its mirrored copy

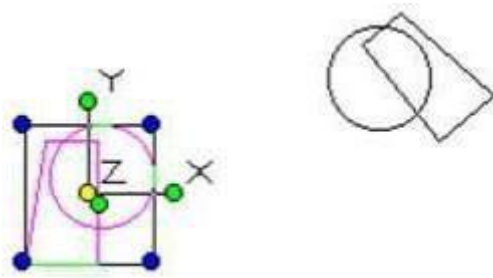
Create a mirror image of an object by defining a mirror line:

Select the objects you want to mirror and activate **Mirror Copy**.

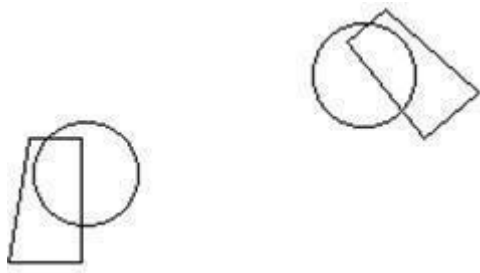
Define the mirror line by selecting two points, or by selecting one point and entering an angle in the Inspector Bar.



3. The mirror copy is made after the mirror line has been defined.



4. Click anywhere to exit **Select** mode, or press Esc.



Create a mirror image of an object by selecting a face:

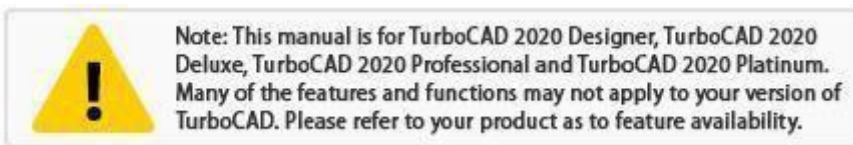
1. Select the objects you want to mirror and activate **Mirror Copy**.

2. Define the mirror line by a face.

The mirror copy is made after the mirror line has been defined. Click anywhere to exit **Select** mode, or press Esc.

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Note: For text to be mirrored, it must be **Flexible**. Otherwise the text will read the same way on both sides of the mirror line. See [Text Properties](#). When selecting objects for mirroring, especially text or dimensions, it is important that the selector be in 2D Mode. If the selector is in 3D mode the text will be mirrored as well as any geometry. In 2D mode the text remains normal (in readable orientation) while the geometry is mirrored.



Linear Copy

Default UI Menu: Modify/Array/Linear

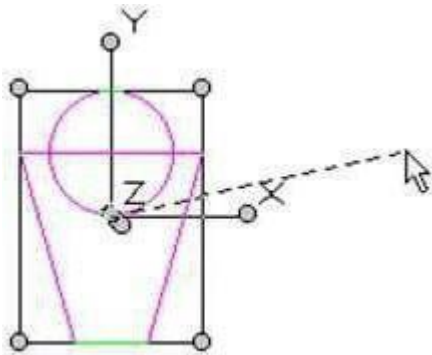
Ribbon UI Menu:

Creates copies of 2D or 3D objects along a straight line, in which each copy is separated by a specified distance.

Select the objects you want to copy and activate **Linear Copy**.

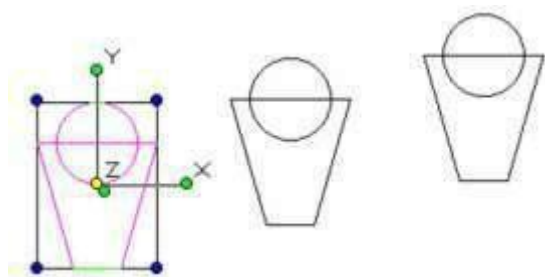
In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.

Set the distance between adjacent copies by using the mouse to define the copy vector, or enter **Step** values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.



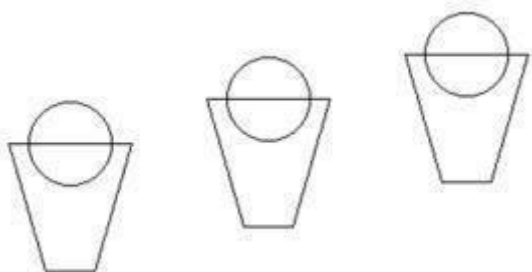
Note: To make a single copy using a distance vector that does **not** start at the reference point of the selected objects, see [VectorCopy](#).

1. The copies are made after the vector has been defined.



Sets = 3

2. Click anywhere to exit **Select** mode, or press Esc.



A good use for Fit linear copy is for architectural smart object like windows.

With new improvement Array Fit Linear command take into account the all the properties of the object of which linear array is being created e.g. Workplane, etc.

Fit Radial Copy



Default UI Menu: Modify/Array/Fit Radial

Ribbon UI Menu:

Creates copies of 2D or 3D objects along an arc, along which a specified number of copies are evenly spaced and rotated.

Select the objects you want to copy and activate **Fit Radial Copy**.

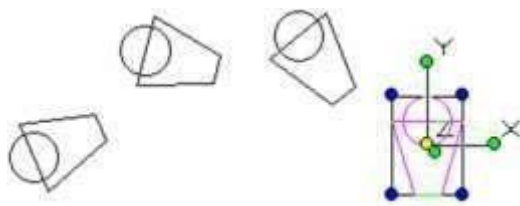
In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.

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Set the angle the first and last copies by using the mouse to define the center of the copy arc and the angle, or enter the angle in the Inspector Bar. You can enter a rotation value to set the rotation of each copy relative to the previous one. If **Rotation** = 0, the copies will not rotate at all.

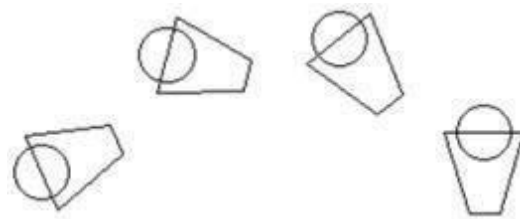
The copy arc is defined from the reference point of the selection set.

1. The copies are made after the arc has been defined.



Sets = 4

2. Click anywhere to exit **Select** mode, or press Esc.



Note: For copying in 3D, the copy arc is positioned on the current workplane. Each copy is rotated around the axis perpendicular to the workplane, passing through the reference point of the selected objects.

Mirror Copy

Default UI Menu: Modify/Copy/Mirror

Ribbon UI Menu:

Options:

Mirror along line: Creates a mirror image of an object by defining a mirror line.

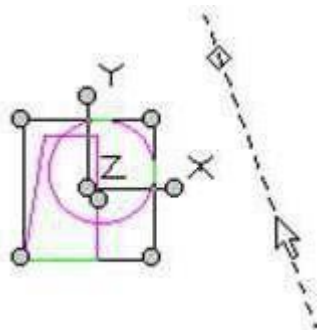
Mirror along face: Select a face plane along which Mirror operation should be performed.

Try to create single object: When selected TurboCAD will attempt to add original object and its mirrored copy

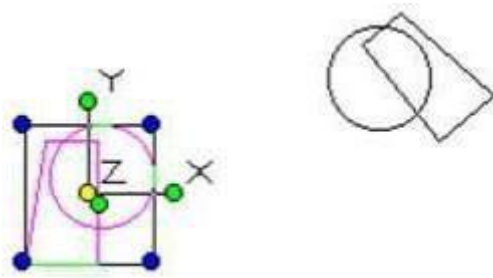
Create a mirror image of an object by defining a mirror line:

Select the objects you want to mirror and activate **Mirror Copy**.

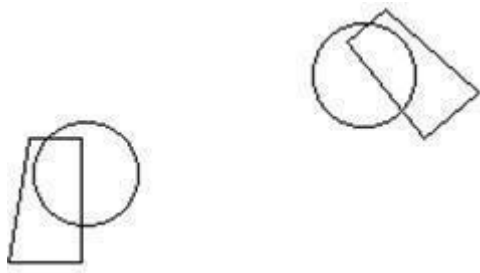
Define the mirror line by selecting two points, or by selecting one point and entering an angle in the Inspector Bar.



3. The mirror copy is made after the mirror line has been defined.



4. Click anywhere to exit **Select** mode, or press Esc.



Create a mirror image of an object by selecting a face:

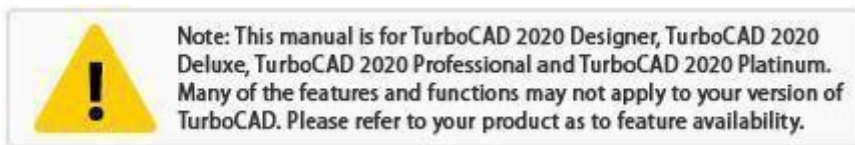
1. Select the objects you want to mirror and activate **Mirror Copy**.

2. Define the mirror line by a face.

The mirror copy is made after the mirror line has been defined. Click anywhere to exit **Select** mode, or press Esc.

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Note: For text to be mirrored, it must be **Flexible**. Otherwise the text will read the same way on both sides of the mirror line. See [Text Properties](#). When selecting objects for mirroring, especially text or dimensions, it is important that the selector be in 2D Mode. If the selector is in 3D mode the text will be mirrored as well as any geometry. In 2D mode the text remains normal (in readable orientation) while the geometry is mirrored.



Linear Copy

Default UI Menu: Modify/Array/Linear

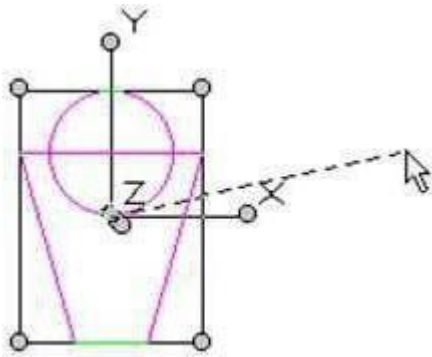
Ribbon UI Menu:

Creates copies of 2D or 3D objects along a straight line, in which each copy is separated by a specified distance.

Select the objects you want to copy and activate **Linear Copy**.

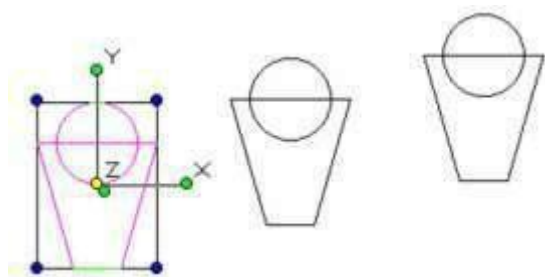
In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.

Set the distance between adjacent copies by using the mouse to define the copy vector, or enter **Step** values in the Inspector Bar. The copy vector is defined from the reference point of the selection set.



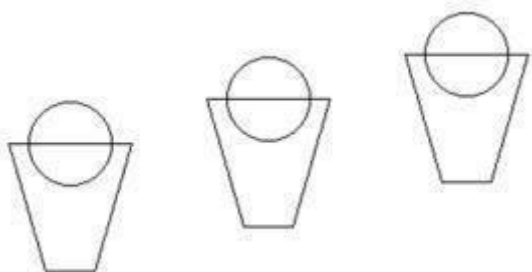
Note: To make a single copy using a distance vector that does **not** start at the reference point of the selected objects, see [Vector Copy](#).

1. The copies are made after the vector has been defined.



Sets = 3

2. Click anywhere to exit **Select** mode, or press Esc.



A good use for linear copy is for architectural smart object like windows. This feature became available in the TurboCAD 2017.

Offset

Default UI Menu: Modify/Offset/Offset

Ribbon UI Menu:

Creates one or more offset copies of a 2D object. The object can be open or closed. For closed objects, the offsets will also be closed (in most cases). You can offset these objects:

- Lines

- Arcs

- Circles

- Ellipses and elliptical arcs; creates a spline

- 2D Polylines

- 3D Polylines whose points remain on one plane; creates a 2D polyline.

- 2D Curves; Bezier and Spline

- 3D Curves whose points remain on one plane; creates a 2D spline.

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The offsets are created so that linear segments remain parallel. When possible, arc start and end angles remain the same as well. Various options become available depending if the **Skip Define Distance step** option is on. This is normally active by default.

A **Multiple** option may be used after selecting the source object instead of specifying a **Sets** amount.

Skip Define Distance step:

active

inactive.

Select the Offset tool.

Set the offset Distance either by entering a value in the Inspector Bar or by selecting two points. The angle of this line does not matter; the value of its length

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will be used.

Enter the number of Sets in the Inspector Bar. This is the total number of copies, plus the original.

4. Select the object.

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Select the side of the object where you want to place the copies.

The offset copies are made on the selected side. In this case, the left object was offset to outside and the right object to the inside. Where lines overlap, they are automatically trimmed.

If you enter a plus or minus **Z Off set** value, each offset copy will be separated from the adjacent one by this height; thus, on a new workplane.

You can use the same parameters to offset other objects, you can change the parameters, or select Finish from the Local Menu or Inspector Bar to exit the tool.

Options:

Finish: Closes the Offset tool and returns you to the Line tool.

Multiple: When an object is initially selected, the Multiple option becomes selectable. If selected, every click of the mouse (LMB) will offset a single copy from the last copy. If more than 2 sets are specified, only one copy is made. **Undo:** Undoes the offset in a "Multiple" operation.

Next Object: Exits the current selected object in a "Multiple" operation and allows you to choose another object.

Rounded Corners: The exterior corners (relative to the offset direction) will be rounded.

Use Compound Profile: When active, all the objects whose endpoints meet, are selected as if they were a single polyline.

Skip Define Distance step: When active (default), the Offset tool uses the last Distance value. When inactive, three choices become available and a Distance value needs to be specified in the Inspector Bar or by selecting two points.

Use Law: Uses a law (formula) to modify and deform the offset copies.

Edit Law: Edit and select the laws to be used

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The following options are only available when the Skip Define Distance step option is turned off:

Through: Offset an object through a point that you click/snap to.

Erase: The origin object is deleted and just the offset remains.

Layer: Specify a Layer.

If the Layer option is turned off the following two options appear:

Current: The current active drawing layer.

Source: Uses the layer of the source object.

Offset with Law

Laws are mathematical functions in the form of a formula. They are used to define how resulting objects are modified.

They are composed and selected in the Laws Editor which is specific to each tool that uses laws and can be opened from that tool.

To open Laws Editor:

Select Offset tool from Modify menu.

Select Use law from local menu or inspector bar.

3. Now select Edit law.

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Laws Editor will open.

To use a law:

Open the Laws Editor.

Select the desired law in the Laws table by double-clicking and press OK. The law that is applied is what is in the formula field.

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To create a law:

Type the desired formula into the formula field.



Press the + (plus) button.

3. Type in a title for the law and press OK. The new law will appear in the Laws table

Law title	Law expression
u	u
new	u

To Delete a law:

1. Open the Laws Editor.

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Click on the Law in the Laws table to select it.

Right click and select Delete, or press the - (minus) button.

Laws cannot be edited but double click a law will load it into the Formula field where it can be altered and then a new altered law added.

$f(u)$

In the Offset tool the function are defined as a function of U. Where U is the distance along the object being offset. Therefore the function is dependant upon the direction that the object is drawn. You can determine the direction by looking at the vertex values in the Selection Info palette.

The effect of the formula is added to the Offset Distance.

Examples:

All of these examples are set with the Offset Distance set to Zero (0).



Law title	Law expression
u	u
Double	2
Double U	2*u
cos	cos(u)

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Setting $f(u)=u$

Setting $f(u)=2$

Setting $f(u) = \cos(u)$

Setting $f(u) = u$

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Notice that the distance between the start and end of the resulting spiral is $\sim \pi$ since the diameter of the original circle is 1.0.

Import and Export laws:

The laws created can also be saved by using the export button in the laws editor. After creating a law, press export and enter the name you want give to the file then press save.

You can also import laws. Open laws editor, and select import then choose the desired file and press open. The laws will be loaded from file and displayed in the laws editor.

Radial Copy



Default UI Menu: Modify/Array/Radial

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Ribbon UI Menu:

Creates copies of 2D or 3D objects along an arc, in which each copy is separated by a specified angle and rotation (optional).

Select the objects you want to copy and activate **Radial Copy**.

In the Inspector Bar, enter the number of **Sets** - the total number of objects that will result.

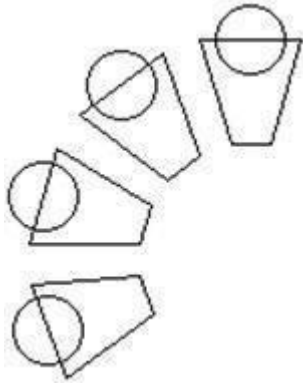
Set the angle between adjacent copies by using the mouse to define the center of the copy arc and the angle, or enter the angle in the Inspector Bar. You can enter a rotation value to set the rotation of each copy relative to the previous one. If **Rotation** = 0, the copies will not rotate at all. By default, **Rotation** = **Angle**.

The copy arc is defined from the reference point of the selection set.

1. The copies are made after the arc has been defined.

Sets = 4

2. Click anywhere to exit **Select** mode, or press Esc.



Note: For copying in 3D, the copy arc is positioned on the current workplane. Each copy is rotated around the axis perpendicular to the workplane, passing through the reference point of the selected objects.

Vector Copy



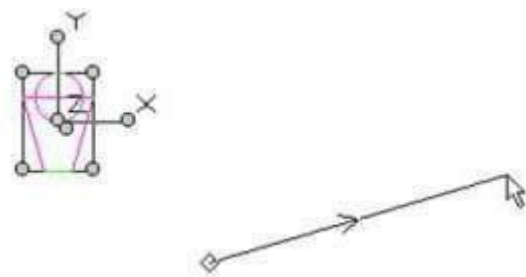
Default UI Menu: Modify/Copy/Vector

Ribbon UI Menu:

Creates a single copy of an object at a specified angle and distance from the reference point of the original.

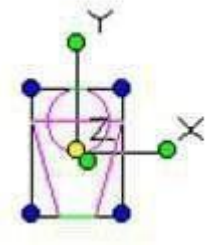
Select the objects you want to copy and activate **Vector Copy**.

Define the copy vector by selecting two points, or by selecting one point and entering the length and angle in the Inspector Bar. The copy vector can be anywhere, but it is applied to the reference point of the selection set.

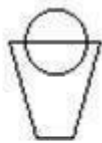


Note: To make one or more copies using a distance vector that starts at the reference point of the selected objects, see [Fit Linear Copy](#)

3. The copy is made after the vector has been defined.



4. Click anywhere to exit **Select** mode, or press Esc.



5.2. Deleting Objects

(Available in all TurboCAD Variants)

Default UI Menu: Edit/Clear

Ribbon UI Menu:

The simplest way to delete objects is to select them and press the Delete key.

The **Clear** menu also contains options for clearing (deleting) objects:

Clear Selection: Deletes all selected objects.

Clear All: Deletes all objects in the active Model Space or Paper Space. Other spaces are not affected, but using **Clear All** in Model Space will leave viewports in Paper Space empty.

Warning: Clear All deletes objects even on layers that are locked and not visible.

To delete only visible, unlocked objects, use **Select All** and the Delete key.

Clear Constructions: Deletes all constructions or a selected construction from the drawing. In Paper Space, only constructions added in Paper Space will be deleted.

See [Construction Geometry](#)

5.3. Isolate and Hide Objects

(Available in all TurboCAD Variants)



It is now possible to select any number of entities in the drawing and then isolate them from other objects. In this way, you can perform editing on these specifically selected or 'isolated' objects without seeing or affecting other un-selected entities in the drawing.

The Isolate tool is particularly useful in a complicated drawing when we are trying to work with a subset of the drawing. When you're done, editing the isolated objects, simply click the 'End Isolation' tool and the drawing re-displays those objects which had not been previously isolated.

There are three options for isolation:

1. Isolate Objects:



Default UI Menu: Edit/Isolate Objects

Ribbon UI Menu:

Use the Isolate Objects tool to temporarily hide all objects in your drawing other than those you select with this tool.

2. End Isolation :



Default UI Menu: Edit/End Isolation

Ribbon UI Menu:

Click End isolation tool to show all the objects that were hidden temporarily.

3. Hide Object:



Default UI Menu: Edit/Hide Objects

Ribbon UI Menu:

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The Hide Objects tool temporarily hides selected object in the drawing. To use this tool, select the object(s) in your drawing you wish to hide and then click Hide Objects. To have hidden objects reappear, click the End Isolation tool.

Select object and click on Hide object.

Demo Video:

<https://youtu.be/y0A-OZ4M3tY>

5.4. Selecting Objects

Selecting Objects

(Available in all TurboCAD Variants)

Default UI Menu: Edit/Select

Ribbon UI Menu:

Tip: In most cases, you can also access **Select** mode by pressing the Space Bar (if another tool is active).

To select a single object, click on it.

To select multiple objects, keep the Shift key pressed while selecting.

To deselect an object from a selected group, Hold Shift-select it again.

To select all objects, use **Select All** or use Ctrl+A.

When a group of objects is selected, press Ctrl to select one of the objects in the group. You can edit just this one object, and when you are finished, click outside the group. The previously selected group is once again selected.

Four hotkeys are also available for selecting:

F6 selects the first object created. Pressing F6 repeatedly will select subsequent objects in the order they were created.

F7 selects the last object created. Pressing F7 repeatedly will select subsequent objects in reverse creation order.

Shift+F6 selects multiple objects. Starting with one object, subsequently created objects are added to the selection set.

Shift+F7 selects multiple objects. Starting with one object, previously created objects are added to the selection set.

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For any selected object or group of objects, you can view and access some properties in the Selection Info Palette. See [Selection Info Palette](#)

Tip: You can use the Design Director to select all objects that reside on a particular layer or sit on a particular workplane, select lights, etc. See [The Design Director](#)

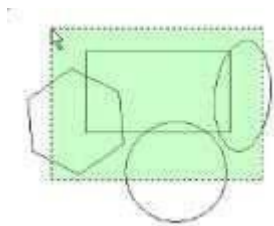
Use Previous Selection

This option in the local menu allows you to re-select the last previously modified selection.

Using a Selection Window

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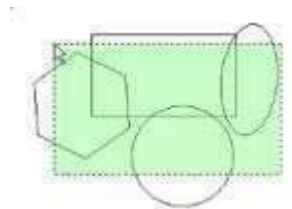
You can select a group of objects by dragging a rectangle around them. Click the first corner of the rectangle, keeping the mouse button pressed, and drag the rectangle to



the opposite corner.

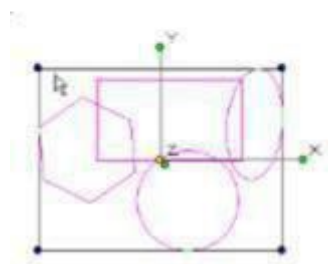
If you need to select objects using a shape other than a rectangle, see [Select by Fence](#)

If you draw the selection rectangle from right to left, all objects completely or partially inside the fence will be selected.



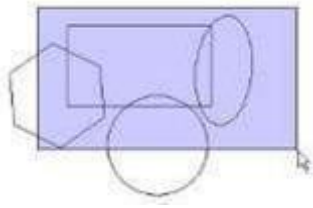
Alternatively, if you need a shape other than a rectangle this can be performed by using [Select by Fence](#)

Draw any shape for selecting the objects. Turn the **Open Window Mode 'On'** from local menu if it is '**Off**'. Then click on '**Finish**'.



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If you draw the selection rectangle from left to right, only objects completely inside the fence will be selected.



Alternatively, if you need a shape other than a rectangle this can be performed by using [Select by Fence](#)

Draw any shape for selecting the objects. Turn the **Open Window Mode 'Off'** from local menu if it is 'On'. Then click on 'Finish'

Super Selector Modes

You can specify how selection will work with the Super Selection modes: Rectangular mode, Window Polygon mode, Crossing Polygon mode, and Fence mode.

Rectangular mode - the default standard bi-directional rectangle selector.

Window Polygon mode - objects completely inside closed polygon will be selected.

Click and drag to set the first two points of the selection window.

Click each subsequent point to define the selection window.

Double click the last point to close the window and finish the selection.

Crossing Polygon mode - objects crossed by or within a closed polygon will be selected.

Click and drag to set the first two points of the selection window.

Click each subsequent point to define the selection window.

Double click the last point to close the window and finish the selection.

Fence mode - objects crossed by an open polygon (fence) will be selected.

Click and drag to set the first two points of the selection window.

Click each subsequent point to define the selection window.

Double click the last point to close the window and finish the selection.

Selection Info Palette

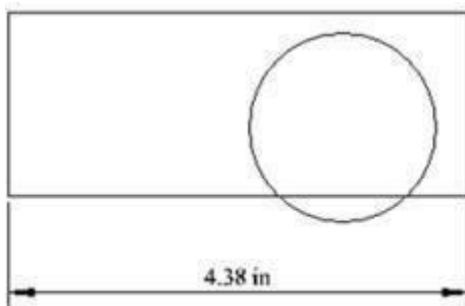
Default UI Menu: Tools/Palettes/Selection Info

Ribbon UI Menu:

Note: For details on using the Selection Info palette for 3D objects, and to see part history, see [Editing 3D Objects using Selection Info](#)

Selection Info - Properties

This example consists of a rectangle, a circle, and a linear dimension.



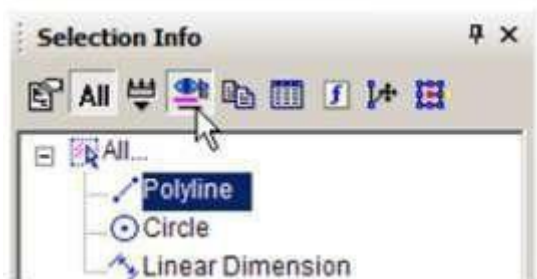
Open the Selection Info palette to see the three objects listed.



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Tip: The number of selected objects must be less than or equal to **Maximum Selection Info Entities**, which is set in the **Program Defaults** page of the TC Explorer Palette.

In the Selection Info toolbar, click **Highlighting**, and make sure Polyline is selected.



This highlights the rectangle in the drawing window.

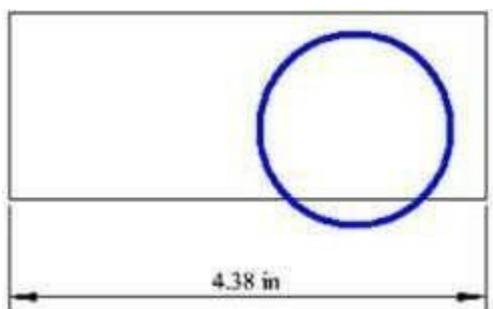
Conversely, if you select objects in the drawing area, they will be highlighted in the Selection Info palette.

Note: If a group is selected, its objects will be listed under the group name in a similar tree format.

You can use this palette to change an object's properties. Highlight the circle, and the lower section of the palette contains the categories of properties for the circle. In this example, the color and line weight was changed in the **Pen** category.

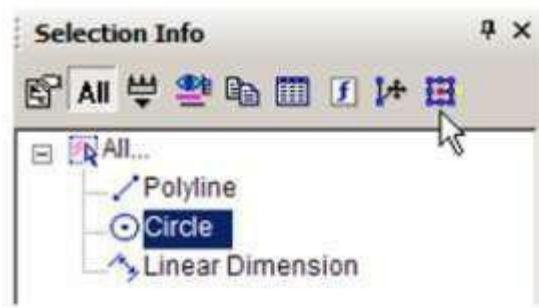
Note: For 3D ACIS objects, physical properties are not automatically listed. Click the **Physical Metrics** icon in the palette toolbar if you want to see engineering properties (volume, moments of inertia, etc.).

The new properties appear in the drawing area. (You may have to turn off the **Highlight** icon in the toolbar to see the changes.)



To make some changes to the circle, click **Select**.

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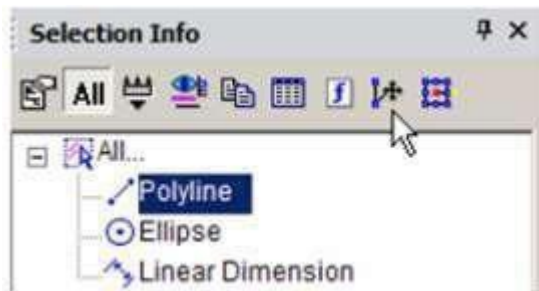


This invokes **Select Edit** mode, and all objects other than the circle are faded. Perform a change like moving and scaling the circle. (For details on editing this way, see [Select Edit](#)). Note that this changes "Circle" to "Ellipse" in the palette.

Select **Cancel** to exit **Select Edit** mode.

You can also invoke the Edit Tool. Highlight the Polyline and click **Edit Tool**. This enables you to reshape an object by moving its nodes. See [Edit Tool](#).

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Move one or more nodes to change the shape.

To exit the **Edit Tool**, select **Cancel** twice.

To change the dimension, highlight and make changes in the **Text** category.

The changed dimension appears in the drawing area.

Note: You can also edit an object's properties by clicking the **Properties** icon in the palette toolbar. This opens the **Properties** window for the object (see [Object Properties](#).)

The **Copy** icon is used to view an object's properties in HTML format. They will appear in your default browser. The HTML file is stored in the ...Profiles/Built-in folder.

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To set options for the Selection Info palette, click **Options**.



The options at the top control highlighting and fading for objects that are not being edited. To set a filter for what objects are displayed in the palette, click **New**. This opens the **Selection by Query** window, in which you can specify types of objects you want to display.

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Note: For details on selecting by query, see [Select by Query](#)

1. Each filter must be assigned a name.



2. To apply this filter, click **Filter** in the toolbar.



3. Select the filter you just created.

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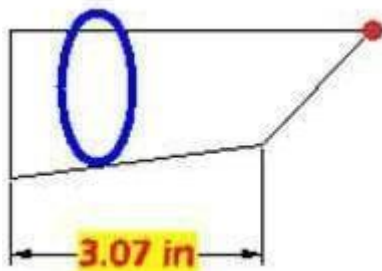


Now only the polyline and dimension appear in the palette.

You can change geometry by modifying geometry in the **Metrics** category. Open **Metrics** for the Polyline - the coordinates for each vertex are listed.

Open the branch for a vertex to see its X and Y coordinates. These values can be changed.

The vertex highlighted in the palette is also highlighted in the drawing area.

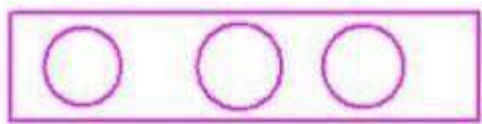


Some values cannot be edited, such as Area and Perimeter. These values depend on the coordinates of the vertices and are grayed out

.

To see options for another type of object, create several objects and group them (select them, then select **Format / Create Group**).

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The Extents Size parameter has been added to the selection Info palette properties. Extents Size parameter also works with groups, blocks, and insertions. You can use the 'Extents Size' property to select entity according to size valuation. Select objects that you think are big or small, or object in any range you want.

The Extents Size update provides you with additional flexibility so that you can use this parameter for various needs and for analyzing wrong entities.

Demo Video:

<https://youtu.be/6y4uCMfXFi0>

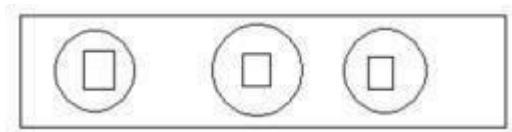
The Extents Size update provides you with additional flexibility so that you can use this parameter for various needs and for analyzing wrong entities.

You can also select objects on the basis of extents size parameter using Select by Query (see [Select by Query](#)).

Note: For details on groups, see [Groups](#)

This object is a "Group of Graphics," and you can open it to see each object that comprises the group. To edit the group, click **Edit Content**.

The group's contents fill the screen. You can make some change, such as adding or removing objects from it.



3. When finished, click **Finish Edit Content**.

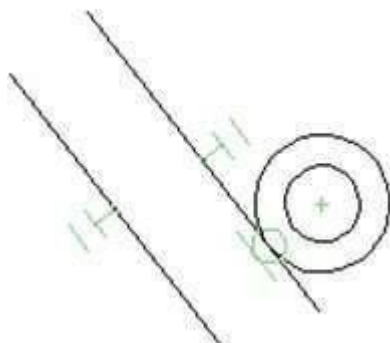


Selection Info- Constraints

The **Constraints** tab of the Selection Info palette shows what constraints have been applied, and to which objects they are applied.

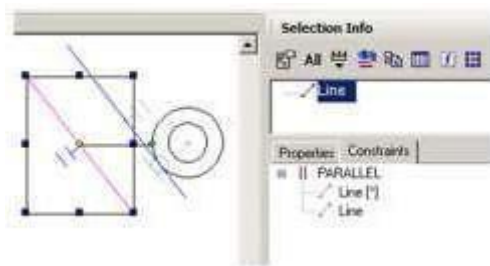
For details on constraints, see [Constraining Geometry](#)

This example consists of two lines and two circles. Three constraints were applied: the two lines are constrained to be parallel, the circles are constrained to be concentric, and a tangent constraint was applied to the outer circle and adjacent line.

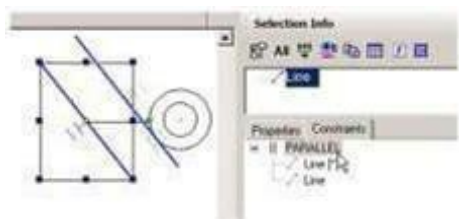


Make sure the **All** icon is disabled at the top of the Selection Info palette. Select the outer line, which only has one constraint (parallel). In the **Constraints** tab, the PARALLEL constraint is listed, below which are listed the two lines to which the constraint applies.

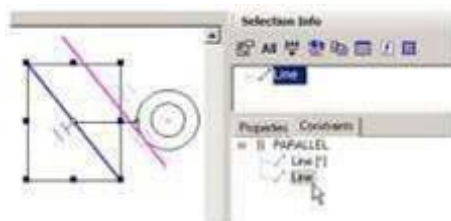
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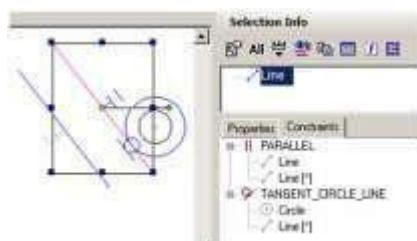
Move the cursor over the PARALLEL constraint in the **Constraints** tab, and the two lines that have this constraint are highlighted.



3. Move the cursor over either line below PARALLEL. The relevant line is highlighted.



Now select the inner line, which has two constraints applied to it. These two constraints are listed: PARALLEL and TANGENT.



Now click **All**. The four objects (two lines, two circles) are listed, along with several blockmarks. These are the constraint markers themselves, which can be selected and deleted if you want to remove the constraint. In the **Constraints** tab, two items are listed for each constraint - one for each of the constrained objects.

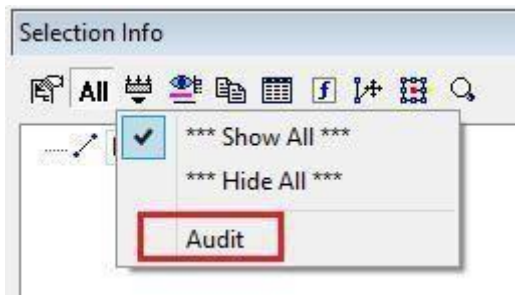
Select any blockmark in the palette to see it highlighted onscreen (**Highlighting** must be enabled).



Audit System:

“Audit” is added to menu “Filters”. Audit System is used to detect a ‘bad’ object in drawing. A ‘bad’ object triggers at least one audit conditions.

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Font-weight: normal;

Font-style: normal;"New item "Audit" was added to menu "Filters".

The possibility to highlight 'bad' objects in Selection Info palette

The possibility to turn on/off audit conditions and changing their parameters

The possibility to generate an audit report for one or more objects

The possibility to filter 'bad' objects in Selection Info palette

The possibility to send notify (about a 'bad' object) to Event Viewer palette

The bad objects are highlighted in the selection info palette with severity mark with audit icons.

Audit Report:

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We can also generate an audit report by clicking on the audit report icon on the menu bar of selection info palette.

"Audit" report (HTML page).

Audit Options:

Audit icons which are shown in Selection Info Palette with objects can be enabled or disabled by Selection Info Palette Options dialogue box. To open Selection Info Palette Options dialogue box select options icon from selection info menu bar.

Select "Options..." from Selection Info Palette Options.

"Audit" section

New audit condition 'ACIS check entity' (Level = 70) was added. Switched 'Off' by default.

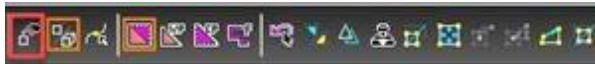
Demo Video:

<https://youtu.be/DUOAOwo0CpM>

2D-3D Selector

The properties of the **2D Selector** can be set in the **Selector Properties** window. This can be opened by right-clicking anywhere while in **Select** mode, and selecting **Selector 2D Properties** from the local menu. (If the 3D Selector is active, the local menu item will be **Selector 3D Properties**.)

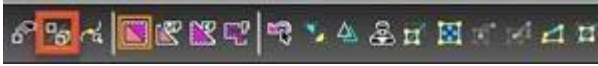
You can also click the icon on the Inspector Bar to open the **selector Properties**.



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While working, you can easily switch between **2D** and **3D Selector Mode**, without having to open the **Selector Properties** window.

Simply click the Toggle 2D/3D* icon on the Inspector Bar.



General

Controls how 2D and 3D objects are selected, and what transformation fields appear in the Inspector Bar.

2D Mode for Model and Paper Space: Treats all objects as if they are 2D, and only uses the 2D selection rectangle. The Inspector Bar contains transformation fields in X and Y only.

3D Mode for Model Space: Treats all objects as if they are 3D, and only uses the 3D selection box. The Inspector Bar contains transformation fields in X, Y, and Z. **2D/3D**

Depending on Selected Object and Space Mode: Treats objects differently; 2D objects are treated as 2D objects and 3D objects are treated as 3D objects.

Edit Properties: If this option is set, double-clicking on an object with the selector will open its Properties dialog.

Edit Content (Geometry): If this option is set, double-clicking on any object with the selector will set the object for editing. For most objects, this will be the equivalent of activating the edit tool. For text double clicking will allow you to edit the text content.

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Customize: Allows you to specify how double clicking will act for each class of objects.

Selector 2D

Options for 2D selection and fields in the Inspector Bar.

Shell Shape: Select **None** to make the selector shell (bounding rectangle) invisible.

The options are:

Box

None

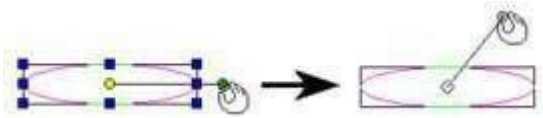
Manipulation Options

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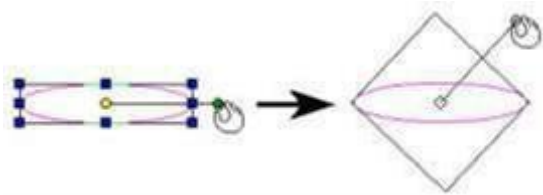
Show Rotation Bars: If unchecked, the rotation bar is not shown and is not accessible.

Link Selector Shell with Rotation Bar: If checked, when you press Ctrl and move the rotation handle, the selector shape will change accordingly.

Link Selector Shell not checked



Link Selector Shell checked



Show Object while Dragging: Displays the selected objects dynamically as they are transformed. If not checked, only the selection shell is visible.

Inspector Bar: Controls the 2D transformation fields that appear on the Inspector Bar.

See [Inspector Bar](#)

Note: The 2D Selector always moves the UCS (workplane) to the selection.

Selector 3D

Selector 3D contains options for 3D selection such as additional options for selection boxes and Inspector Bar fields. Many of these parameters are the same as for the Selector 2D page.

Configuration

Use Conceptual: Activates the Conceptual Selector (3D).

Coordinate System

CS Stands for COORDINATE SYSTEM

Ext ents CS: Sets the coordinate system (workplane) used by the selector shell. Because these settings work in tandem with other settings, their performance can become complex. Careful study of the **Ext ents CS** and **Coordinate system** parameters will help you understand this powerful tool.

Entity CS: Selector shell CS equals the CS of the selected objects.

User CS: Selector shell CS equals the CS of the current workplane.

World CS: Selector shell CS equals the world CS.

View CS: Selector shell CS equals the CS of the plane of the current view.

Inspector Bar: Sets the coordinate system (workplane) used by the Inspector bar. Because these settings work in tandem with other settings, their performance can become complex.

Entity CS: Inspector bar CS equals the CS of the selected objects.

User CS: Inspector bar CS equals the CS of the current workplane (UCS).

World CS: Inspector bar CS equals the world CS.

Dragging Options

Show Object: Displays the selected objects dynamically as they are transformed. If not checked, only the selection shell is visible. Two options are available which allow you to

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activate showing object for Wireframe, or Render, or both.

Rotation Bar (dragging with Ctrl key): If unchecked, the rotation bar is not shown and is not accessible.

Link Selector Shell with Rotation Bar: If checked, when you press Ctrl and move the rotation handle, the selector shape will change accordingly.

Inspector Bar

Sets the fields that appear in the Inspector Bar (see [Inspector Bar](#))

Scale and Size: X, Y, and Z fields.

Position: Current location.

Shift: Change in position.

Rotation: Rotation from a position relative to the Extents CS.

Delta Rotation: Change in rotation from the current position.

Move by direction and Distance: Delta Distance and Delta Angle work in conjunction.

Delta Distance: the distance an object is moved along the specified Delta Angle

Delta Angle: the angle along which an object is moved the specified Delta Distance There are only four possible scenarios where the user changes the value in either one or both of these fields and then presses Enter. This is similar, in some ways, to draw a line segment by length and angle.

Like the other Delta fields, once the user presses Enter and the changes have been made the fields will revert to a value of 0 (zero).

The Use Cases are as follows:

#1

<u>Delta Angle Field</u>	Changed,	<u>Delta Distance</u>
<u>Field</u>	Changed	

Result

Everything that is selected moves the specified Distance at the specified Angle.

Example: Delta Distance (16mm) at 36.8 degrees. The selected objects will move 16mm at 36.8 degrees.

#2

<u>Delta Angle Field</u>	Unchanged	<u>Delta Distance</u>
<u>Field</u>	Changed	

Result

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Everything that is selected moves the specified Distance at 0 degrees. Example: Delta Distance (16mm) at 0 degrees. The selected object/s will move 16mm at 0 degrees.

#3

<u>Delta Angle Field</u>	Changed	<u>Delta Distance</u>
<u>Field</u>	Unchanged	

Result

Nothing happens. Moving a zero distance is a Null.

#4

<u>Delta Angle Field</u>	Unchanged	<u>Delta Distance</u>
<u>Field</u>	Unchanged	

Result

Nothing happens. Moving a zero distance is at a zero angle is a Null.

Manipulation

Safe UCS Mode: Prevents changes to the current workplane when you manipulate objects with the 3D Selector. If unchecked, you will get a warning message when using 3D Selector for 2D object replacement.

Warning: Use this option with caution. Simply moving an object along the same workplane will not change the workplane, but rotating an object will change the workplane.

Set by normal of facet: New mode is meant to facilitate the assembling of 3D objects. If Set by normal of facet = On and snap to facet is turned on (Nearest on facet = On), then Z axis of the selector is placed on the normal to the facet.

Classic

Shell Shape: **Box** is the default. The **Sphere** shell is useful for visualizing rotation, but does not allow drag and drop scaling. Select **None** to make the selector shell invisible.

Conceptual

Show Handles

These options set which parts of the Conceptual Selector are visible. The options are: Move, Scale, Rotate, Boundary Box. and the Selection CS.

Screen Size

The options go from 50 pixels to 1000 pixels.

Minimum: Sets the minimum size for the display of the selector.

Maximum: Sets the maximum size for the display of the selector.

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Mouse Cursor

Set cursor by handle's transform: When on, this causes the cursor to alter in order to indicate the available operation (scale, rotate etc.)

Lighting

The Conceptual Selector is drawn in a separate render scene, which has its own lighting, independent of the drawing.

Type: Use this to specify what the Conceptual Selector will use for lighting: Ambient, Headlight, Compound headlight.

Ambient and Headlight light sets are the regular ambient and headlight light sources.

Compound headlight set (by default) consists of three headlight light sources, which are not located at the camera center. Instead, they are positioned on different sides of the view direction. This provides a better 3D effect when lighting the Universal Selector.

Profile

The profile defines the appearance and functionality of the Conceptual Selector. There are two default options Normal and Simple, and a third custom option, User. The Simple option has fewer controls than the Normal. Users can create their own profiles. To select a user created profile select User in the drop-down list. Then browse to and select the appropriate TCW file containing the profiles specification.

More details and specifications for the Conceptual Selector can be found under: [Conceptual Selector Details](#)

Inspector Bar Options

When using the 3D Selector, the local menu and Inspector Bar provide three options that do not appear when working in 2D.

Set UCS by Selector: Moves the UCS origin to the selection reference point.



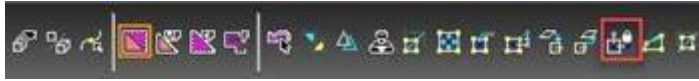
Set Selector by UCS: Moves the selection to the UCS origin (similar to the **Format / Place on WorkPlane** option for 2D objects).



Note: The 2D Selector always moves the UCS (Workplane) to the selection.

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Lock/Unlock Axis: Locks or unlocks a rotation bar. Move the cursor to the end of the rotation bar you wish to lock or unlock, and select **Lock** or **Unlock Axis** from the local menu.



Auto select compound profile: When selecting curves for a geometry tool, this option will enable you to select smoothly connected chains of curves.



Click Test

If this option is turned on, then the Rectangular Mode fence will be used for the visualization of a screen area where the 'Click Test' will be performed. This provides a method for verifying and identifying those areas on the screen which are valid for the selection of objects.

For each pixel of the specified area (by Rectangular Mode fence) the virtual 'mouse click' will be performed. If the user clicks on a pixel that will allow a graphic to be selected, then those pixels are colored by the graphic's color. If there are selected graphics, then other graphics are ignored. The test takes into account the 'Aperture size', Render Mode and Layer's styles.

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Example of Click Test:

Conceptual Selector Details

The Conceptual Selector is enormously powerful and is extremely adaptable. The following sections detail the usage and management of this tool.

Note: The Conceptual Selector, since version 2018, is only available to use if the RedSDK plug-in is installed.

Introduction

In contrast to the classic mode, the Conceptual Selector uses the full-blown 3D model, materials, lighting, adjustable scale and editing capability. The user is able to edit: the selector geometry, materials, the set of handles, response to mouse movements, etc. All data about the Selector is stored in [TCW Profiles](#), which are accessible for editing. The **GCtrlProfiles** folder contains selector variants (profiles), and the **Samples** sub-folder contains additional Samples. A user can create and use his own profiles. At the moment Conceptual Selector is available only in Model Space and only for the 3D mode of selector. Using RedSDK imposes certain restrictions. the Conceptual Selector can be displayed only in RedSDK windows. When objects in non-RedSDK windows are edited, it will automatically force the classic selector mode to be used. As is, the primary functions of the Conceptual Selector are:

- Move only back and forth along this axis (X).

- (Cube) Scale bi-directionally but only along this axis (X).

- (Rectangular cuboid) Scale uni-directionally on this side (equivalent to dragging one face of the selection bounding box).

- (Rectangular cuboid) Scale uni-directionally on this side (equivalent to dragging one face of the selection bounding box (opposite face from #4)).

- (Rectangular cuboid) Scale along both axes (X, Y).

- (Colored arc and cylinder) Drag, but only on two axes (X, Y).

- (Yellow sphere) the reference point.

(Colored ball) Acts like the rotation handle for accurate snapping (Y).

(Colored ball) Rotate around associated axis (X).

(Colored ball) Rotate around associated axis (Z).

(Colored arc) Rotate around associated axis.

Boundary box

To use any of the handle you may click-hold-and-drag or click-drag-click.

Colors and Materials

The Normal profile uses the following color scheme: XYZ -> RGB, that is, colors Red, Green, Blue correspond to axes X, Y, Z. Handles that are located on the axes or that are tied to the axes by transformation (rotation around the axis) have the color matching that specific axis. Shades of Red, Green, Blue are used either for auxiliary highlight geometry (which appears when the mouse cursor hovers over a handle):

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Using light red color for highlight geometry

Or to differentiate an 'accurate' (i.e. with snap) handle from non-accurate:

Using light blue color (on the right) to mark imprecise rotation

The Yellow, Magenta and Cyan colors are used for the handles that work in XY, XZ and YZ planes, respectively.

Another possibility for planes is to paint the handles using the colors of corresponding axes, for example, paint one half of the handle red, the other - green . But as the handles themselves are small, this type of coloring is often not visible enough. Light Gray is used for the Boundary Box.

Colors used in the Normal profile.

Conceptual Selector Boundary Box

In addition to the handle controls, you can scale using the Boundary Box (if it is turned on in the Selector Properties)

Scale By Side

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Hover over the side of the boundary box until it highlights (Note: This will only happen if you are in a RedSDK rendering mode, not GDI or LightWorks).

Click and Drag to scale the object along that sides axis.

Scale By Edge

Hover over the edge of the boundary box until it highlights (Note: This will only happen if you are in a RedSDK rendering mode, not GDI or LightWorks).

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Click and Drag to scale the object along that the two axes that are perpendicular to that edge.

Scale By Corner

Hover over the corner of the boundary box until its three associated edges highlight (Note: This will only happen if you are in a RedSDK rendering mode, not GDI of LightWorks).

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Click and Drag to scale the object along all three axes.

Interactive Handles

General Structure and Function

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As mentioned above, profiles can have an arbitrary set of handles. Below is shown a list of handles for the default Normal profile (UniversalCtrl.tcw).

Move only back and forth along this axis (X).

(Cube) Scale bi-directionally BUT only along this axis (X).

(Rectangular cuboid) Scale uni-directionally on this side (equivalent to dragging one face of the selection bounding box).

(Rectangular cuboid) Scale uni-directionally on this side (equivalent to dragging one face of the selection bounding box (opposite face from #4)).

(Rectangular cuboid) Scale along both axes (X, Y).

(Colored arc and cylinder) Drag, but only on these two axes (X, Y).

(Yellow sphere) the reference point.

(Colored ball) Acts like the rotation handle for accurate snapping (Y).

(Colored ball) Rotate around associated axis (X).

(Colored ball) Rotate around associated axis (Z).

(Colored arc) Rotate around associated axis.

Boundary box

Snapping, accurate /nonaccurate handles

Handles that provide accurate transformation (with mouse snap) are marked with outlined callouts on the illustration (handles 7, 8). For the handle to perform an accurate

transformation, it is necessary to specify yes for the `handle_snap_to_center` variable in the profile. In this case, the mouse cursor will be pulled up to the reference point or to the center of the visible part of this handle.

'`handle_snap_to_center`' variable

In reality, even if the above-mentioned variable is not set, a handle can perform accurate transformations, for example, using the edit bars.

Priority

A user can assign priority to handles. If several handles with varying priority get into the cursor aperture, the handle with the higher priority will be chosen. The following priorities are set for the default profile: Reference Point has the highest priority (number 0), Boundary box has the lowest (-1). Priority is controlled by the `handle_priority` variable:

'`handle_priority`' variable

Small Handles

The ability to zoom in/out makes it reasonable to create handles of different sizes. A combination of large and small handles provides a way to fit interactive handles in the window tightly, without obstructing the view.

Seven handles at the spike of the X axis

Grouping Handles by type of Transformation

Tighter packing enabled by the grouping of handles (portals) by type of transformation. Such portals are slightly larger than a handle, but they allow for all types of transformations. One of the problems with the classic selector was that it was impossible to change the rotation axis. Typically, a user would realize that he selected a wrong axis only after rotation was started. In Conceptual Selector, a portal that contains handles 8, 9, 10, 11 (see the handles layout) allows rotating around an axis. The Normal profile includes the following portals:

Portals: movement along axes, movement in planes, rotation around axes.

Movement along axes, shown dynamically. Animated GIF (animation only in HTML version)

Introduction

In contrast to the classic mode, the Conceptual Selector uses the full-blown 3D model, materials, lighting, adjustable scale and editing capability. User will be able to edit selector geometry, materials, the set of handles, response to mouse movements, etc. All data about the manipulator is stored in [TCW Profiles](#), which is accessible for editing. GCtrlProfiles folder contains selector variants (profiles), and Samples subfolder - additional **Samples**. User can create and use his own profiles.

Lighting

As mentioned, the Conceptual Selector is drawn in a separate render scene, which has its own lighting, independent of the drawing.

There are three preset light sets available to the user:

Ambient, Headlight, Compound headlight.

The Ambient and Headlight light sets are the regular ambient and headlight light sources. The Compound headlight set (by default) consists of three headlight light sources, which are not located at the camera center but are positioned on different sides of the view direction. This provides a better 3D effect when lighting Conceptual Selector.

The profile parameters affect diffuse and Affect specular provide a way to adjust intensity of corresponding lighting components.

Lighting: Compound headlight, Headlight, Ambient

Repositioning Control Handles of the Selector

Changing the Reference Point

To move the reference point, you must first select (or pick) it. There are several ways to do this:

- Press D

- Press Ctrl and click the reference point

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Select **Edit Reference Point** from the local menu or Inspector bar.

The cursor changes to a hand symbol.

Click on the Reference point to select it. (Note: if you used D this step happens automatically.)

Move the reference point to its new location

4. Click to set the new location.

Now any moving, scaling, or rotating will be done relative to this new point.

To return the rotation bar to its default position, select **Default Reference Point** from the local menu or the Inspector Bar.

Changing the Rotation Bar

For both ends of the rotation bar, you can select the points by pressing Ctrl and clicking. (The yellow point is the reference point and has additional ways to be selected.)

1. The cursor changes to a hand symbol. Click on the Rotation handle.

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2. Move the handle to a new location.

3. Click to set the location.

To return the rotation bar to its default position, select **Default Rotation Bars** from the local menu, or click the icon on the Inspector Bar.

Now any rotating will be done relative to the new rotation handle and reference point.

Response to User Actions

Using the Cascading Style Sheets (CSS) language creates the possibility to have a formal description of visual style for the conceptual Selector which is dependent on user actions. For example, a button can have different styles depending on whether the mouse cursor is hovering over it or not. A similar approach is used in the Conceptual Selector. For each handle in the selector, three types of geometry can be added to the TCW profile: when the mouse cursor is outside the handle, when it is over it, and while dragging. This allows for the active handle to be highlighted. It also provides feedback (tips) to the user about the transformation that is associated with the handle in addition to the nature of handle movement while dragging (axis, plane). Generally, an experienced user does not need tips, so it is desirable to have a way to turn them off. Currently, this can only be done by deleting the corresponding graphics from the profile. As a default, when the cursor is placed on the handle, a transparent, 10% larger copy is drawn. This provides a highlighting effect:

Response of MoveX handle to mouse movement.

Representation of ScaleXY handle for 'mouse cursor outside', 'mouse cursor over', 'while dragging'. Light red and light green are used to denote the possibility to move along X, Y axes, and the result for movement of the arc along an axis.

TCW Profiles

Profiles can have an arbitrary (non zero) number of handles, which can access any available transformations, including non-interactive (for example, coordinate-axes). Filtering of the handles in the current profile can be done by using filters by type of transformation from the selector settings dialog.

Filtering handles of the current profile by type of transformation

Users can create their own profiles or modify existing ones. There are, however, certain specific characteristics of profile loading that need to be observed. To save time on reading the TCW file, profiles are unloaded only when the application is closed.

Otherwise, they would have to be opened/closed upon each activation of the corresponding tool. This can slow down the work significantly. So if a profile is already uploaded to memory, reloading it (for instance, after editing) is possible only after restarting the application. A new profile can be set up without an application restart.

Samples

The Samples folder contains several examples of different profiles. It makes sense to start all new templates by using the template.tcw, which contains the minimum set of handles and also includes all required variables (graphics custom properties).

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Template.TCW profile

Defined custom properties in the Database Define Fields dialog

TheMoveRotate2D.TCW sample demonstrates the possibility of creating a selector in a "protractor" style. It contains 12 handles for movement along axis, positioned at every 30 degrees, and a handle for rotating around the z-axis.

MoveRotate2D.TCW profile

The FlatGrid3x3.TCW example demonstrates the possibility to create a flat 5x5 grid from handles for accurate movement in the XY plane (reference point). Grid spacing is 1-inch. A specific feature of this grid is that the distance between handles (1 in) does not change when zooming in/out. Ability to set fixed values of coordinates (in a selector coordinate system) allows for placing selector handles at critical points for drawing objects (for example, focal plane of the lens or focuses of an ellipse).

FlatGrid3x3.TCW profile

Layers

All profiles that are distributed with TurboCAD have a certain system of layers. Each handle has its own layer. This provides a way to hide unnecessary handles. Layers

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whose names have the "handle" prefix are used for handles. For example, the "_handle_MoveX" layer contains the handle for movement along the x-axis. Layers whose names have the "part_" prefix are used for parts of handles. For example, the "part_MouseIn" layer contains the geometry that becomes visible when the cursor hovers over the handle.

Fragment of layer table from a profile.

It should be noted that this system of layers exists only as an added convenience for profile development. When a profile is uploaded into the application, layer 0 is assigned to all handles.

Handles

When a profile is uploaded into the application, the loader analyzes graphics and uses certain criteria to identify handles. A handle is a 'group' graphic that has a certain set of subgroups and a set of custom properties. Graphics that do not meet these criteria are ignored.

A handle has to have several mandatory custom properties, such as 'text', type that define its name, position, transformation type, etc.

In addition to mandatory custom properties, there are optional properties. These properties are used by the program to determine the purpose, behavior, appearance of the handle, and its response to mouse movement. A user can set other admissible values and change the handling behavior that way.

Set of variables for the Boundary Box handle.

Set of variables for MoveX handle.

The current set of variables is not final as of this writing, it may be changed and expanded. A handle has to include a certain set of subgroups.

Each of the subgroups is responsible either for the drawing or for the behavior of handle.

The type of the subgroup is set by the `handle_part` variable:

'handle_part variable'

This variable can have one of the following values:

mouseout- visible geometry when the cursor is outside the handle

mouseover - visible geometry when the cursor is over the handle

active - visible geometry while dragging (optional)

static- visible geometry (optional)

coordsystem-invisible geometry, defining the coordinate system of the handle

highlight_area - invisible geometry, turns on mouseover when the cursor is over it. If this part is absent, mouseout (optional) is used instead.

visibility_area - reserved (optional)

Parts of MoveX handle

Visual Representation

Foreground Camera

The Conceptual Selector is created in a separate render scene that has its own geometry, lighting, and list of materials. The scene is connected with the foreground camera, so it is drawn in front of drawing objects. Foreground Camera is set depending on render mode

Drawing in front of the main scene provides visibility and accessibility of all the selector handles to the user, regardless of selected objects' positions.

Selector is in front of drawing objects.

Scaling

The user can set the minimum and maximum size of the Conceptual Selector on the screen in pixels. The benefit of an adjustable scale range over a fixed scale is that it gives users the capability to use a single movement (rotating the mouse wheel) to zoom in and get access to additional control handles that are grouped into sets by type of transformation (move, scale, rotate).

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A part of the selector with the size of 200 pixels and 500 pixels

In fact, saying "selector size" is not quite correct. The Selector model contains several sets of graphics that are drawn depending on the selector state and can have different sizes. In addition, there are auxiliary graphics that are not drawn at all and whose size does not matter. So there is the following assumption:

Screen size (min/max) parameters defines the screen size of a unit cube (1 inch) from the TCW profile of selector.

Size of the visible part of Normal profile (UniversalCtrl.tcw)

The idea is that a developer of a TCW profile will fit the size of visible selector geometry into the unit cube. Dependence of selector size on window zoom within the defined limits is practically linear, outside of the limits - a constant (has a resemblance to hysteresis).

Dependence of selector size on window zoom

Zoom in dynamic. Animated GIF. (Animation only in HTML version) Only users can tell how convenient this scaling algorithm is in any instance. It is possible that scaling parameters, default values and the range will need to be adjusted, for example, for desktop and laptop, and touch screen.

Multi-Window Mode, Active Window

The Conceptual Selector is drawn in its entirety in the active window. In other windows (if they are open) - only parts of it are drawn. This was done to speed up drawing as well as single out the active window. Active control handles (that respond to mouse movements) are drawn only in the active window. Those that are not interactive (e.g. coordinate axes) are drawn in all windows.

Drawing in the active (left) and inactive windows.

The User can control the drawing mode in multi-window mode, by specifying 'yes or 'no' for a `handle_multi_view` variable for the specific handle in the TCW profile.

'handle_multi_view' variable

Select by Color

Default UI Menu: Edit/Select by/Color

Ribbon UI Menu:

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Hotkey: Ctrl+K

Selects objects of specified colors.

By default, the color list contains all colors used by visible objects in the drawing. The default list does not include colors used by objects that are contained within groups and blocks. See [Color Palette](#)

Press Shift to select a range of colors from the list. Press Ctrl select or deselect individual colors.

Recursive: Colors of objects contained within groups and blocks are added to the color list. Because the color list needs to be rebuilt, select this option before selecting colors.

Search with:

Or: Objects that contain at least one of the highlighted colors will be selected. **Only:** Objects that contain all of the highlighted colors, and only those colors, will be selected.

And: Objects that contain all of the highlighted colors will be selected.

Add to current selection: All objects selected by the color match will be added to any current selection in the drawing.

Select by Attribute

Default UI Menu: Edit/Select by/Attribute

Ribbon UI Menu:

Hotkey: Ctrl+I

Selects objects that share the same text in their **Attribute** fields. This field is on the **General** page of their **Properties** window.

Type text in the **Attribute** field that exactly matches the contents of the **Attribute** field of a set of objects, then click OK to select the objects.

The attribute text can contain the wildcard symbol. **For example, the attribute "Wall"** will select "Wall1," "Wall+Window" and WallBrick." All characters that follow the * symbol will be ignored.

Case sensitive: The search will distinguish between uppercase and lowercase letters.

All Entities: Selects all objects that share the specified text attribute.

Group names only: Selects only groups that share the specified text attribute.

Block names only: Selects only blocks that share the specified block name - the name that is assigned to a block while creating. The name of an inserted block is listed in the

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Refers to the field of the **Properties** window, **Block Insertion** page.

Text entities only: Select only text objects that share the specified text attribute.

Note: The attribute of a text object is the text string itself.

Exact match: Selects all attributes that exactly contain the text in the **Attribute** field.

By string: Selects all attributes that contain the string in the **Attribute** field.

Add to current selection: All objects selected by the attribute match will be added to any current selection in the drawing.

Select by Entity Type

Default UI Menu: Edit/Select by/Entity Type

Ribbon UI Menu:

Selects objects of specified types.

Press Shift to select a range of types from the list. Press Ctrl to select or deselect individual types.

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Add to current selection: All objects selected by the match will be added to any current selection in the drawing.

In this version, Circle/Ellipse item of previous TC versions is separated into two cases: Circle or Ellipse. The same is for Circular and Elliptical arcs. Previous versions could not select Circles or Arcs separately.

Select by Fence

Default UI Menu: Edit/Select by/Fence

Ribbon UI Menu:

Selects a set of objects by drawing a closed polygonal "fence" around them.

See [Irregular Polygon](#) for details on how to create the fence.

If **Open Window Mode** is activated, all objects completely or partially inside the fence will be selected.

If **Open Window Mode** is not activated, only objects completely inside the fence will be selected.

Select by Layer

Default UI Menu: Edit/Select by/Layer

Ribbon UI Menu:

Hotkey: Ctrl+L

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Selects objects that lie on specific layers. See [Layers](#)

Press Shift to select a range of layers from the list. Press Ctrl select or deselect individual layers.

Add to current selection: All objects selected by the layer match will be added to any current selection in the drawing.

Select by Block

Default UI Menu: Edit/Select by/Block

Ribbon UI Menu:

Select by Block displays the list of blocks in the drawing and allows user to select block(s) via one click.

User can check "Add to the current selection", if there's a selection made already before clicking this tool.

Select by Query

Default UI Menu: Edit/Select by/Query

Ribbon UI Menu:

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Hotkey: Ctrl+Q

Selects objects meeting a complex set of criteria. For example, you can select all blue arcs or yellow dashed curves on Layers 3 and 5. Query selection can be useful in large or complicated drawings in which you need to filter through numerous colors, layers, entity types, etc.

You can write a query manually in the edit box if you know the syntax rules but it is recommended to use the tools in the **Selection by Query** window.

Note: **Query** will not select individual objects that are members of groups or blocks.

Query window: Contains the expression used to filter the objects for selection. You can assemble the expressions via the **Criteria** button, or you can enter them manually.

Note: The query text does not automatically wrap. Press Enter to separate lines. Also, new criteria are inserted at the end of the expression, regardless of the cursor position. You can cut and paste text to correct the order.

And: Selects objects meeting all criteria.

Or: Selects objects meeting any criteria.

Not: Selects objects that do not meet the specified criteria.

Parenthesis: Used for grouping elements of the expression.

Clear: Erases the current expression.

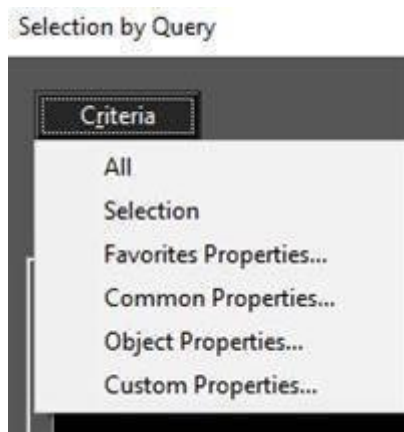
Parse Query: Tests the syntax of the query expression. If there is an error in logic or format, an error message will appear. You will also be notified if the query is sound.

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Note: **Parse Query** only tests the logic of the query expression. It does not test whether objects will be selected, nor does it implement the query.

Query Criteria

The main elements of a query are names of object properties, values of properties, and logical operations. Click the **Criteria** button to start your query.



All: Runs the query on all objects in the file.

Selection: Runs the query only on selected objects.

Common Properties: Enables you to select from a list of commonly-used properties. For example, you can select objects that have certain brush colors, line widths, layers, etc.

For example, If you click **Brush Color**, the **Choose Values** window will have the list of all brush colors. Check all the colors you want to include in the query. At the bottom of the window, you can choose **Any** to select all objects with these colors. If you choose **None**, objects that have other colors will be selected.

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This is how the brush color query looks in the **Selection by Query** window.

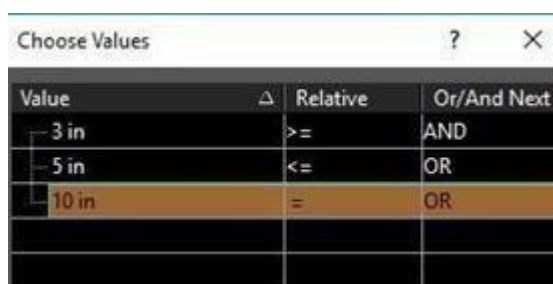


Note: Some properties do not have specific values or a list of choices.

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Object Properties: Enables you to select objects that have certain physical properties such as length or radius. For example, arcs and circles can be selected by arc length, circumference, radius, etc. Click **Radius**.

Enter a **Value** and its **Relative** (equal, less than, greater or equal, etc.). In this example, the query will select arcs and circles with radii between 3 and 5 inches, or equal to 10 inches.



Value	Relative	Or/And Next
3 in	>=	AND
5 in	<=	OR
10 in	=	OR

This is how the radius value query looks in the **Selection by Query** window.

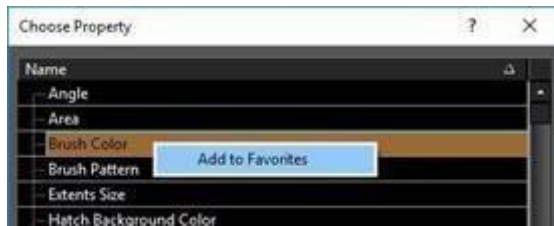


Custom Properties: See [Custom Properties, Database, and Reports](#)

Favorite Properties: Enables you to choose from a list of queries you use often. You can add common properties and object properties to the list of favorites.

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For example, open the **Common Properties**, and right-click on **Brush Color**. Select **Add to Favorites**.



Then open the **Common Properties**, and right-click on **Height** under **Box**. Select **Add to Favorites**.

To see where these favorites are listed, open **Favorite Properties**.

The brush color and height properties are listed here.



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To remove a property from the favorites list, right-click on it and select **Remove**.

Saving and Loading a Query

If you want to use a query later, you can save it. Click Save Query.



Enter a name for the query and click **Save**.

Then in the **Selection by Query** window, find the query name in the List.

To change saved queries, click **Edit List**.

Select by Similar

Available in TurboCAD Platinum and Pro only

Default UI Menu: Edit / Select by Similar

Ribbon UI Menu:

A powerful new tool that allows you to select any 2D or 3D object type or a Block and then find all occurrences of that in the drawing. Very detailed selections can be made that include parameters such as pen color and style, brush color and style, layer, even the selected object's coordinate system. A great tool to use in conjunction with making further global changes to that selected object.

Select Entity By Similar command is available after you've selected an object.

Select Entity By Similar Options command is available if there is at least one open drawing.

Example:

Select an object in the drawing

Then go to the **Edit** menu and choose **Select by Similar**:

After choosing Similar, a new Dialog box will appear which will let you define which of the selected object's parameters will be selected:

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Result: All the objects that are on the same layer as the selected object will be selected automatically.

Following are the available Properties for **Similar By** command:

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2D/3D	Graphic dimension 2D or 3D. Disabled for paper spaces.
Solid Mode	ACIS/Surface/Smesh. Disabled for paper spaces. Disabled for 2D graphics.
Plane	XY plane of Local CS. Disabled for paper spaces. Local CS matrix.Disabled for paper spaces. This is the working title of graphic's coordinate system,which is stored as a matrix of 3D coordinate system.
Matrix	There are three types of coordinates system available. Entity Coordinate System (ECS) Object Coordinate System (OCS) Graphic Coordinate System (GCS)

Demo Video:

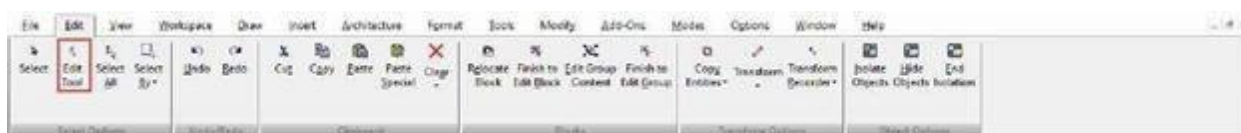
<https://youtu.be/aenXqqM4qjQ>

5.5. Select Edit

(Available in all TurboCAD Variants)

Default UI Menu: Edit/Select

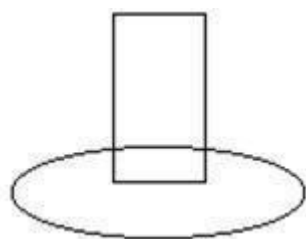
Ribbon UI Menu:



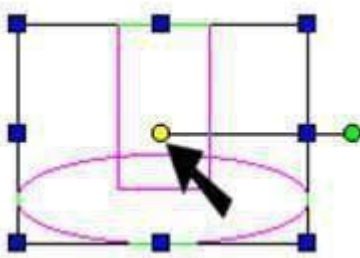
This section covers ways you can edit objects in the current selection set.

Components of Select Edit Mode

For this section, consider the following example of two 2D objects.

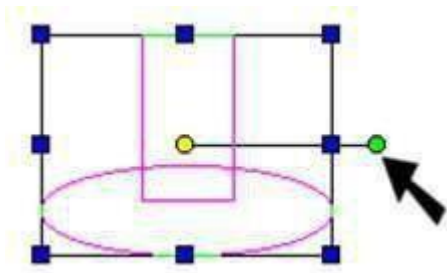


When they are both selected in **Select** mode, the selection set appears in magenta, with several small circles and squares. The yellow circle is the **reference point**. By default, it is located at the center of extents of all selected objects, but it can be moved. It is used for moving, and as a reference for rotating and scaling.

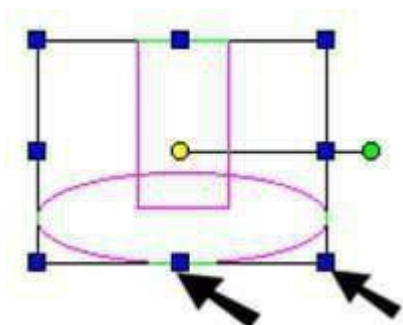


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The green circle is the **rotation handle**. You can click and drag this point to rotate the objects around the reference point.



The blue squares are **drag handles**. You can click and drag these handles to scale the objects. Click an interior handle to resize one dimension of the rectangle. Click on a corner handle to resize both dimensions.



Changing the Reference Point

To move the reference point, you must first select (or pick) it. There are several ways to do this:

- Press D

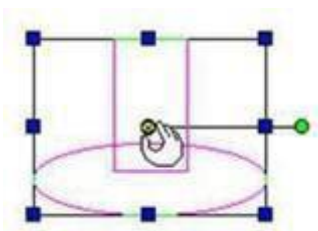
- Press Ctrl and click the reference point

- Select **Edit Reference Point** from the local menu

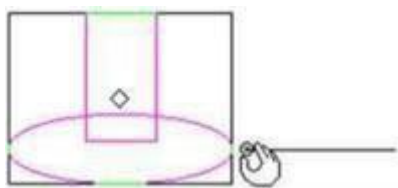
- Click the icon in the Inspector Bar



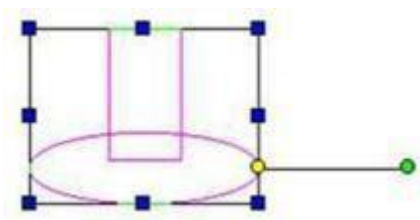
The cursor changes to a hand symbol.



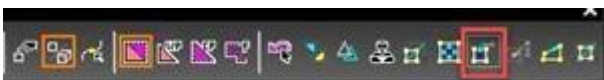
Move the reference point to its new location, in this case the quadrant point of the ellipse.



Now any moving, scaling, or rotating will be done relative to this new point.



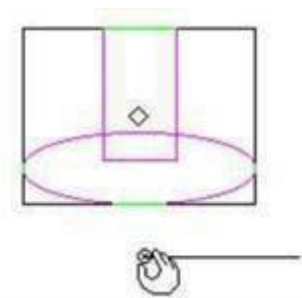
To return the rotation bar to its default position, select **Default Reference Point** from the local menu, or click the icon on the Inspector Bar.



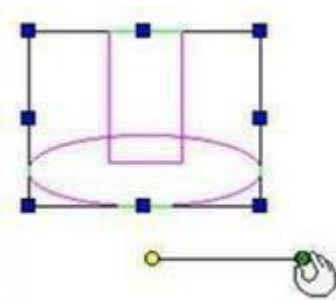
Changing the Rotation Bar

For both ends of the rotation bar, you can select the points by pressing Ctrl and clicking. (The yellow point is the reference point and has additional ways to be selected.) The cursor changes to a hand symbol. In this example, the reference point is moved first.

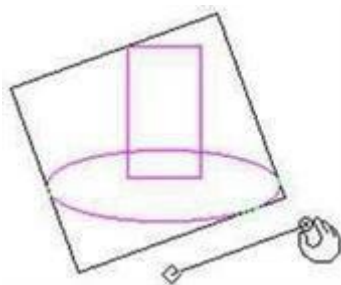
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The green rotation handle is selected and moved the same way.

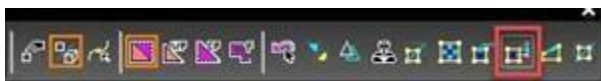


When you move the rotation bar, whether the selection rectangle changes depends on the **Link Selector Shell** option in the **Selector 2D** page of the **Selector** properties. See

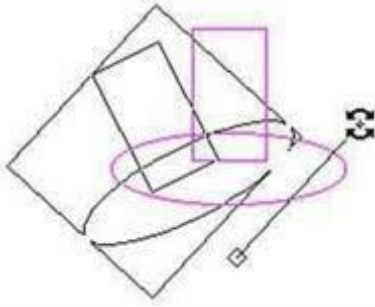


[2D-3D Selector](#).

To return the rotation bar to its default position, select **Default Rotation Bars** from the local menu, or click the icon on the Inspector Bar.



Now any rotating will be done relative to the new rotation handle and reference point.



Copying Objects in Select Edit

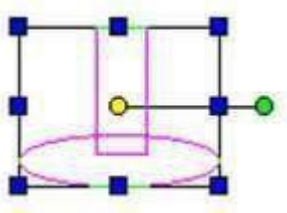
This section covers ways you can copy objects while in **Select** mode.

While these methods are generally the most convenient for making quick copies, there are other tools you can use for more complex and detailed copying. See [Copying Objects](#). For other tools you can use to copy objects relative to specific points, see [Transforming](#)

Make Copy

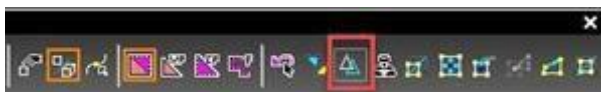
Creates one copy of the selected objects.

1. Select the objects you want to copy.



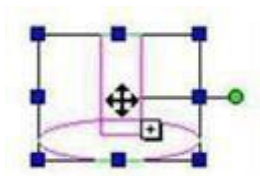
Warning: If you want to change the reference point, do not do so while **Make Copy** is active, or you will create a copy on top of the original. Change the reference point first, then activate **Make Copy**. See [Components of Select Edit Mode](#)

2. Select **Make Copy** from the local menu, or click the icon on the Inspector Bar.

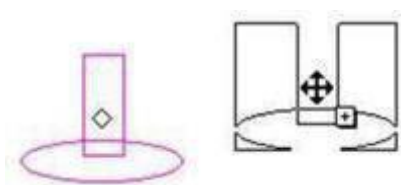


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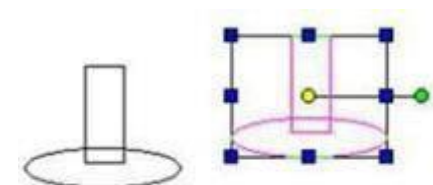
Click the reference point, and the cursor changes to a four-directional arrow with a plus sign.



Move the reference point to the new location, or enter the new position in the Inspector Bar.



The copied objects are now the ones selected. If you want to make another copy, you must click the reference point again.



6. When the objects are copied, click anywhere or press Esc to exit **Select** mode.



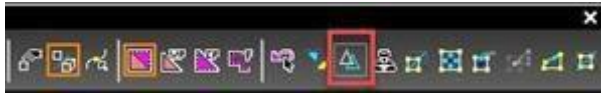
7. Turn off **Make Copy** unless it is still needed.

Simple Dragging to Copy

If you place the cursor anywhere inside a selection other than on the reference point, you can simply drag the selection (keeping the mouse pressed) to another location within the drawing. If **Make Copy** is active, the objects will be copied instead of moved.

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To activate **Make Copy**, select it from the local menu, or click the icon on the Inspector Bar.

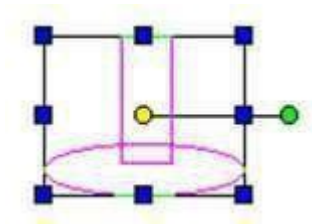


With simple dragging, you cannot use snaps or the Coordinate Fields, but it is convenient if you do not need accurate placement. Turn off **Make Copy** when it is no longer needed.

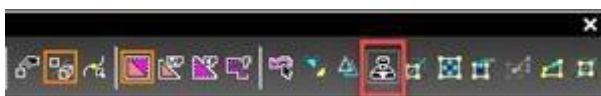
Rubber Stamp

Creates multiple copies of the selected objects.

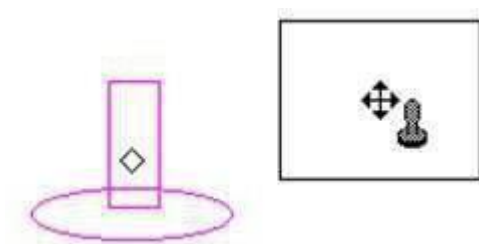
1. Select the objects you want to copy.



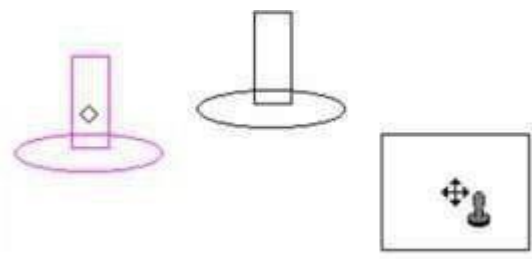
Select **Rubber Stamp** from the local menu, or click the icon on the Inspector Bar.



A rubber stamp icon appears, and you can click to locate the first copy.

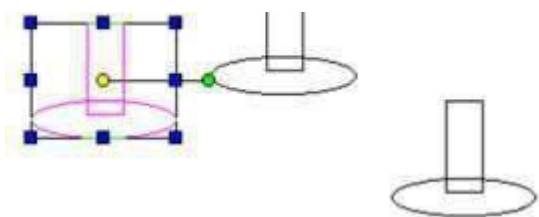


4. Create as many copies as you need.

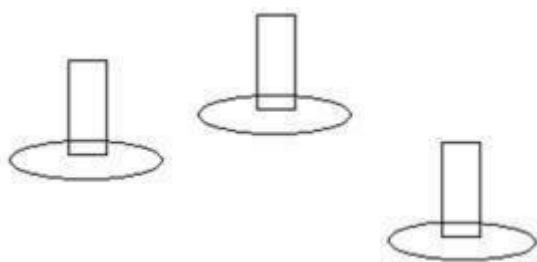


To finish press Esc, or select **Cancel** from the local menu or Inspector Bar.

The original objects are still selected.



6. Click anywhere or press Esc to exit **Select** mode.



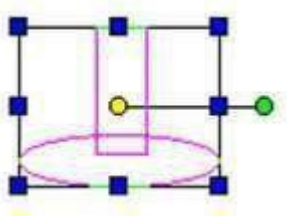
7. Turn off **Make Copy** unless it is still needed.

Making a Rotated Copy

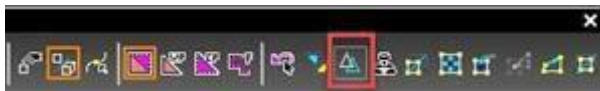
Creates one rotated copy of the selected objects.

Select the objects you want to copy. (See [Components of Select Edit Mode](#) if you want the rotation bar in a different place.)

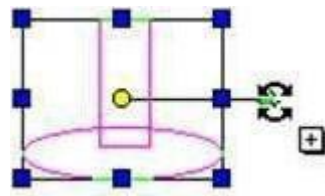
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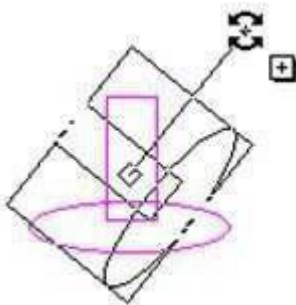
Select **Make Copy** from the local menu, or click the icon on the Inspector Bar.



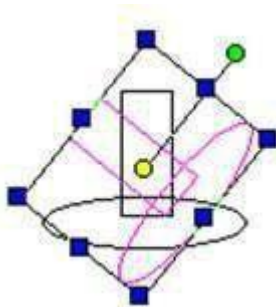
Click the rotation handle, and the cursor changes to a rotation arrow with a plus sign.



4. Move the rotation handle to the new location, or enter the angle in the Inspector Bar.

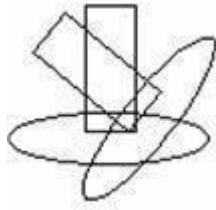


The copied objects are now the ones selected. If you want to make another copy, you must click the reference point again.



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6. When the objects are copied, click anywhere or press Esc to exit **Select** mode.

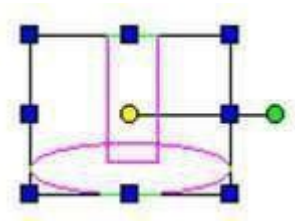


7. Turn off **Make Copy** unless it is still needed.

Making a Scaled Copy

Creates one scaled copy of the selected objects.

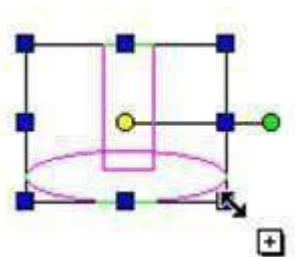
1. Select the objects you want to copy.



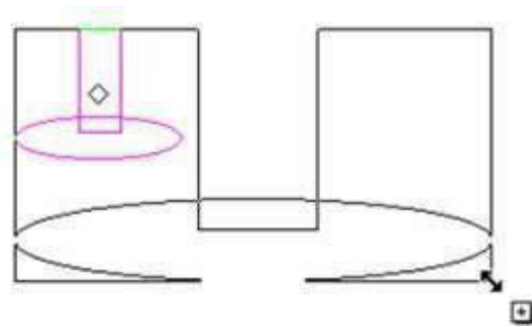
Select **Make Copy** from the local menu, or click the icon on the Inspector Bar.



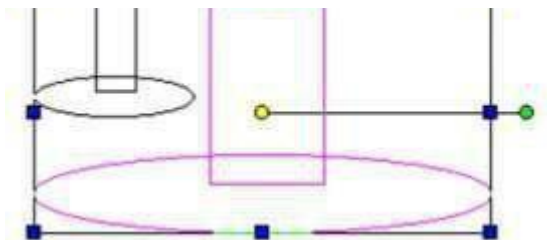
Click one of the drag handles, and the cursor changes to a resize symbol with a plus sign.



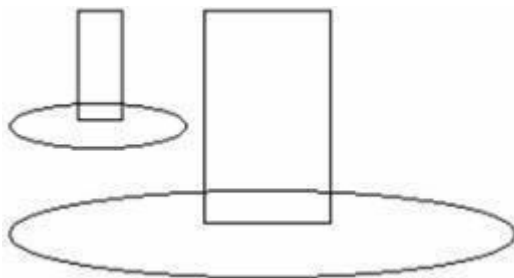
Drag the handle to the new location, or enter the new scale or size in the Inspector Bar.



The copied objects are now the ones selected. If you want to make another copy, you must click the reference point again.



6. When the objects are copied, click anywhere or press Esc to exit **Select** mode.



7. Turn off **Make Copy** unless it is still needed.

Using Ctrl to Copy

An easy way of creating one copy of selected objects is to use the Ctrl key. The procedure is basically the same for moving, rotating, or scaling objects.

To create a copy, select the objects and start moving by clicking the yellow reference point. Press Ctrl, and complete the move.

To create a rotated copy, select the objects and start rotating by clicking the green rotation handle. Press Ctrl, and complete the rotation.

To create a scaled copy, select the objects and start dragging one of the blue rotation handles. Press Ctrl, and complete the scaling.

Geometric and Cosmetic Select Modes

Geometric extents are based on the distances between object vertices (the three corners points of a triangle or the two endpoints of a line). **Cosmetic** extents are based on distances between the outermost points on the object that will be drawn with the minimum (zero) pen width. Therefore, the difference between the two modes is relevant for objects such as double lines, multi lines, and objects that have a nonzero pen width. The select mode will come into play when using snaps and assigning dimensions. For most applications, and for greater accuracy, use **Geometric** select mode. There are two main advantages to using **Geometric** select mode:

When you scale using geometric extents, it is easy to see exactly what is going to be scaled. For example, if you scale a double-line object, the double-line separation distance and pen width are not scaled.

Geometric select mode makes this visually obvious.

When you scale an orthogonal linear object using geometric extents, you get accurate results. For example, if you scale a 1-inch horizontal line segment 200 percent along the X axis, the line segment will always be 2 inches long. If you were to use cosmetic extents, the results would vary depending on the pen width.

Tip: If you want to change the width (separation distance) or pen width of a double line, you can enter new values in the **Properties** window.

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The disadvantage of geometric selection is that if you snap the endpoint of a wide line to another object, the cosmetic end of the line will overlap the object. You can work around this problem by selecting a line using cosmetic extents and using a snap mode to position the cosmetic end of the line.

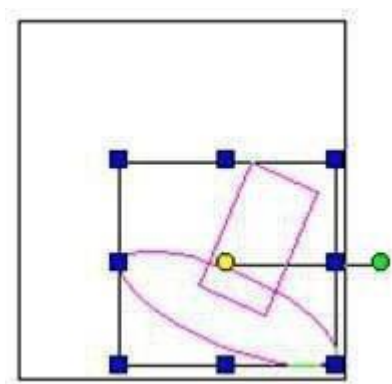
Warning: **Geometric** select mode is **not** the installation default. If you choose to use **Cosmetic** select mode, zoom in closely and be careful where you snap. If you set **Geometric** select mode, it will remain active the next time you open TurboCAD.

Moving Objects in Select Edit

The easiest way to move objects is to select the objects, then click and move their reference point. You can also specify the move distances in the coordinate fields.

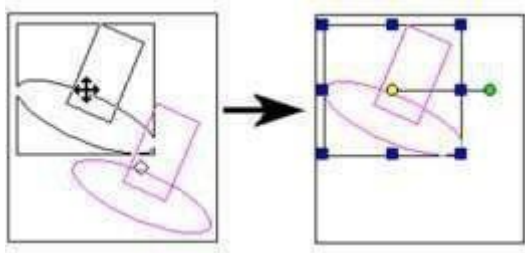
Note: To move (or copy) objects relative to specific points or objects, see [Assembling](#)

Select the objects you want to move. (See [Components of Select Edit Mode](#) if you want the reference point in a different place.)



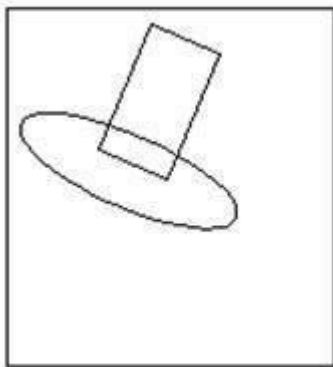
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Click the reference point, and the cursor changes to a four-directional arrow. Move the reference point to the new location, or enter the new position in the Inspector Bar.



Note: You can make a copy by clicking the reference point, pressing Ctrl, and then locating the new reference point.

3. When the objects are moved, click anywhere or press Esc to exit **Select** mode.



Moving along One Axis

You can move or drag an object along the X or Y axis (or Z for a 3D object) by clicking and moving the axis handle itself.

Simple Dragging

If you place the cursor anywhere inside a selection other than on the reference point, you can simply drag the selection (keeping the mouse pressed) to another location within the drawing. With simple dragging, you cannot use snaps or the Coordinate Fields, but it is convenient if you do not need accurate placement.

OLE Drag and Drop

This method is essentially the same as moving objects in **Select** mode (clicking and moving the reference point.) However, for OLE drag and drop, the object is dragged - the mouse is kept pressed while moving.

The main advantage of OLE drag and drop is you can copy the selection from one drawing into another, or even from a drawing into another application. You may also use this method to create symbols or blocks by dragging into the Library palette or Block palette. See [Blocks](#) and [Library](#)

Warning: The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. It is therefore not a recommended method for moving objects that require precise placement.

Nudge

Nudging moves objects by small increments.

If you pick up an object you can nudge its position using the numeric keypad. The NumLock key must be on for this to work.

You can pick up a selected object by clicking its reference point, but it is better to use Ctrl + Num 5. Ctrl + Num 5 will pick up the object without any possibility of you accidentally dragging the object.

Use 4 and 6 to move the object along the X Axis.

Use 8 and 2 to move the object along the Y Axis.

Each time you tap a key the object will move a small amount in the associated direction.

Holding down the Shift key will cause the distance moved to be significantly greater.

You can drop a selected object by clicking its reference point, but it is better to use Ctrl + Num 5. Ctrl +Num 5 will drop the object without any possibility of you accidentally

dragging the object.

The Nudge distance is specified in Options | Program Setup | Advanced Preferences

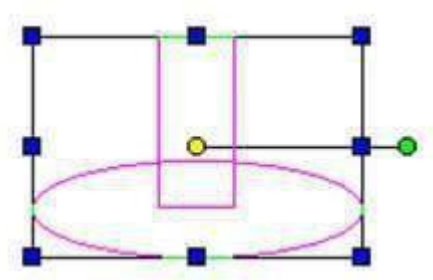
Rotating Objects in Select Edit

The easiest way to rotate objects is to select the objects, then click and move their rotation handle. You can also enter rotation values in the Inspector Bar.

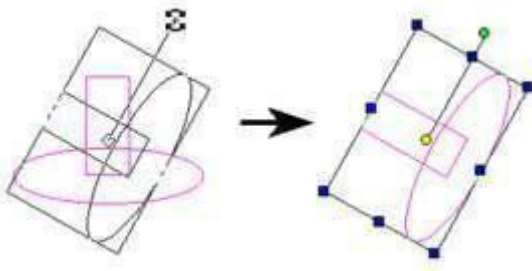
Note: To rotate (or copy) objects relative to specific points or objects, see [Assembling](#)

Tip: If you are using the 3D Selector, you can lock a rotation axis. Right-click on the rotation handle and select **Lock Axis** from the local menu. See [2D-3D Selector](#)

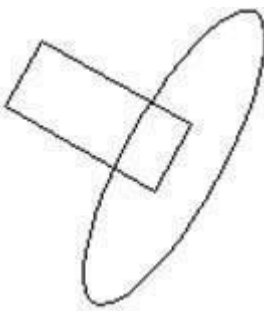
Select the objects you want to rotate. (See [Components of Select Edit Mode](#) if you want the rotation bar in a different place.)



Click the rotation handle, and the cursor changes to a rotation symbol. Rotate to the new position, or enter the angle in the Inspector Bar.



3. When the objects are rotated, click anywhere or press Esc to exit **Select** mode.



Note: You can make a copy by clicking the rotation handle, pressing Ctrl, and then rotating.

Scaling Objects in Select Edit

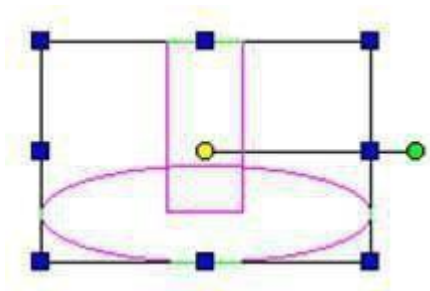
The easiest way to scale objects is to select the objects, then click and move drag handles. You can also enter scale values for relevant axes in the Inspector Bar.

Note: To scale (or copy) objects relative to specific points or objects, see [Transforming](#)

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Tip: If you are scaling an ACIS solid object, you can specify uniform scaling or enable scaling in all axes - see [ACIS](#).)

1. Select the objects you want to scale.



If you want to maintain the aspect ratio (the ratio between X, Y, and Z), select **Keep Aspect Ratio** from the local menu or click the icon on the Inspector Bar.



You can also maintain the aspect ratio by clicking a drag handle and pressing Shift while resizing.

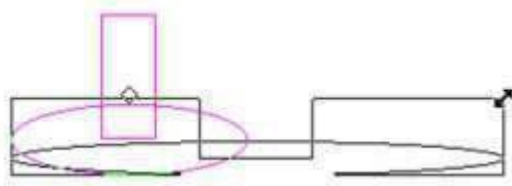
Click one of the drag handles to resize, or enter the scale and size in the Inspector Bar.

You can use **Keep Center of Extents** when scaling. This option forces all scaling to occur relative to the center of the object. If this option is off all scaling will occur relative to the reference point.

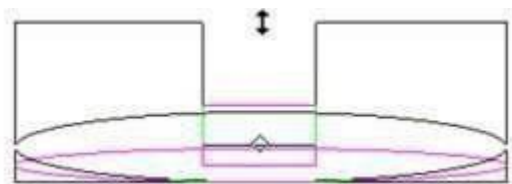


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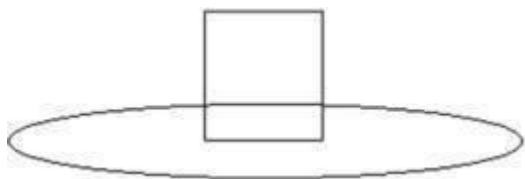
Note: You can make a copy by clicking the drag handle, pressing Ctrl, and then resizing. Click on a corner handle to scale in more than one dimension.



Click on an interior handle to scale in one dimension only.



When the objects are resized, click anywhere or press Esc to exit **Select** mode.



Note: If the object you are scaling contains double lines or lines drawn with a pen width greater than 0, the results of scaling a selection will vary, depending on whether you are using **Geometric** or **Cosmetic** selection. For accurate results, use the default **Geometric** select mode. See [Geometric and Cosmetic Select Modes](#)

Select Edit in 3D

The **Select Edit** tools (see [Select Edit](#)) work the same way in 3D as for 2D.

As with **Select Edit** for 2D objects, you can move, rotate, scale, and copy objects when they are selected. The only difference is that with 3D space there are additional options for movement. Whereas in 2D, you can only rotate in the XY plane, 3D objects can be rotated in the XZ and YZ planes as well. Movement can be in the Z direction, in addition to X and Y.

You need to activate the **3D Selector** in order to have all editing tools available. See [2D-3D Selector](#)

5.6. Transforming

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Transform

Ribbon UI Menu:



The **Transform** tools enable you to quickly move, rotate, or scale objects. You can optionally leave the original object in place, creating a copy. In addition, you can record the transform operations for use on other objects.

You can display the **Transform** toolbar by right-clicking in any toolbar area and selecting **Transform**.



The **Transform** tools can be used on 2D and 3D objects. For the objects you want to use, be sure the Selector is set correctly (2D, 3D, or both). See [2D-3D Selector](#)

Note: The **Assemble** tool also enable you to move and rotate objects, relative to other objects or locations. **Assemble** tools require more accurate input than **Transform** tools, however, and do not create copies. See [Assembling](#)

The **Transform** tools are typically invoked when objects are already selected. You can also invoke them even if no objects are currently selected, as long as the **Select** tool is active. In this case, the local menu option **Select** can be used to select objects to copy. Close the **Select** option when all objects are selected.

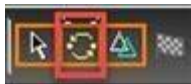
For each of the transform tool (**Move**, **Rotate**, **Scale**, and **Generic**), the following local menu options are available:

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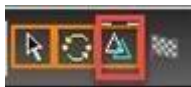
Select: Enables you to add or remove objects from the selection set. When the correct objects have been selected, deactivate this object to continue with the operation.



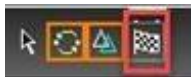
Repeat: Enables you to transform or copy the selected objects again. Otherwise, the function ends once the first operation is complete.



Keep Original Object: Activated by default, makes a copy of the selected objects. Otherwise the object is simply transformed.



Finish: Complete the transform operation.



Generic



Default UI Menu: Modify/Transform/Generic

Ribbon UI Menu:

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Transforms a 3D object by selecting six points.

The first two points define the movement distance, similar to using **Move**.

The third and fourth points enable you to rotate the object in the current workplane.

The fifth and sixth points enable you to flip the object.

Note: The Generic Transform tool is also available from the local menu of the selector tool.

Local menu options:

Scaling: Enables you to change the object's scale while transforming.



Keep Aspect Ratio: If **Scaling** is used, choose whether to maintain the object's aspect ratio.



Skew: If **Scaling** is used and **Keep Aspect Ratio** is not used, you can scale the object while transforming.



Move



Default UI Menu: Modify/Transform/Move

Ribbon UI Menu:

Moves or copies objects relative to two defined points.

1. Select the objects you want to move or copy, and activate **Move**.

5

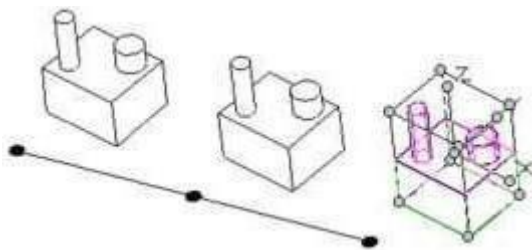
2. Select the source point - the first of two points that defines the transform vector.

Select the destination point. The objects are moved along the transform vector from their original location.

Note: By default, the original objects remain in place. If you do not want to create a copy, deselect the local menu option **Keep Original Object**.

4. The new or moved object is now the one selected. Click outside to deselect it.

If you use the local menu option **Repeat**, you can transform the selected objects again. Otherwise the operation ends.



Rotate



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Default UI Menu: Modify/Transform/Rotate

Ribbon UI Menu:

Rotates and/or copies objects relative to three defined points.

Select the objects you want to rotate or copy, and activate **Rotate**.

From the local menu or Inspector Bar, select the axis of rotation (X, Y, or Z).



Select the center of rotation.

Select the source and destination points - these establish the transform angle relative to the rotation center.

5. The objects are moved along the transform angle from their original location.

Note: By default, the original objects remain in place. If you do not want to create a copy, deselect the local menu option **Keep Original Object**.

6. The new or moved object is now the one selected. Click outside to deselect it.

If you use the local menu option **Repeat**, you can transform the selected objects again. Otherwise the operation ends.

Scale



Default UI Menu: Modify/Transform/Scale

Ribbon UI Menu:

Scales and/or copies objects relative to three defined points.

1. Select the objects you want to scale or copy, and activate **Scale**.
2. Select the scaling start point and end point.

When selecting the destination point, the closer to the scaling end point, the closer the scale approaches 1:1.

The closer to the scaling start point, the scale approaches zero.

3. Select the scaling destination point.

4. The objects are scaled, relative to their original location.

Note: By default, the original objects remain in place. If you do not want to create a copy, deselect the local menu option **Keep Original Object ---**

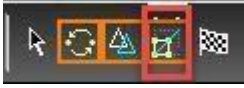
5. The new or moved object is now the one selected. Click outside to deselect it.

If you use the local menu option **Repeat**, you can transform the selected objects again. Otherwise the operation ends.

Additional local menu option:

Keep Aspect Ratio: By default, the objects are scaled so that all dimensions remain

proportional to the original.



Deactivate this option if you want to be able to change the aspect ratio.

Transform Recorder

Default UI Menu: Modify/Transform Recorder

Ribbon UI Menu:

Enables you to record a transformation and apply it to other objects.

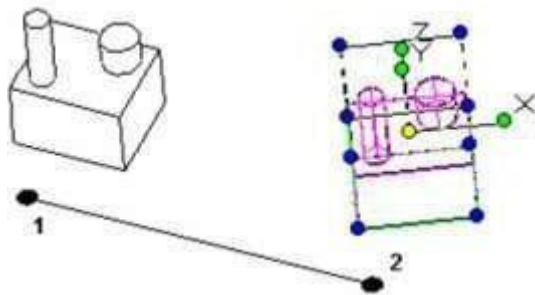
1. Select the objects whose transformation you want to record.

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Click Start Transform Recorder.



Perform the transformation, using the **Move**, **Rotate**, **Scale**, and **Generic** tools as needed. In this example, **Move** with **Keep Original Object** was used to copy the object from Point 1 to Point 2. Then **Rotate** was used on the copied object, with **Keep Original Object** turned off.



When the transformation is complete, click **Stop Transform Recorder**.



Add a new object or group of objects to which to apply the recorded transformation.

Select this object and click Apply Stored Transformation.



The **Move** and **Rotate** commands are applied to the selected object.

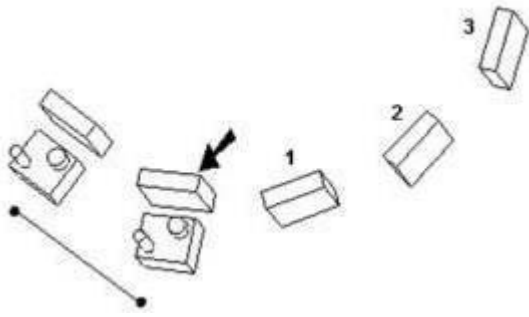
If you want to apply the transformation with different settings, click **Player**.



In the Inspector Bar you can specify the number of sets (the number of additional copies) to create. Set any additional parameters in the local menu or Inspector Bar, and click **Play**.



In this example, the transformation was applied to the previous copy, and three additional copies were made.



It is important to note that **Play (Apply Stored Transformation)** uses settings from the **Player (Customize and Apply Stored Transformation)**. **Player** options remain in effect until changed, and will affect subsequent transformations, even if they are re-recorded.

Note: If you specify multiple **Sets** and **Keep Original Objects** is turned off, the selected objects will be transformed by the distance times the number of sets. No copies will be made.

Local menu options:

Only one option is unique to **Player**.

Toggle CS - If not selected, the object will be rotated around the center of the original object. If selected, the reference point of the new object will be used as the center of rotation.

Scale by Two Points



Default UI Menu: Modify/Transform/Scale by two Points

Ribbon UI Menu:

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This tool allows you to select two points and resize the distance between those two points.

Specify the location of the anchor point (1)

Specify the location of the source point (2) Enter a value in the Size field and press enter, Or

Specify the location of the destination point (3) by:

Drag then select Finish (from the Inspector bar or Local menu)

Drag and then click.

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Note: Using the destination point will use the distance between the source point and the destination point to specify the size.

Options:

Keep Original Object: Create a copy of the original. On by Default.

Keep Aspect Ratio: The objects are scaled so that all dimensions remain proportional to the original. On by default. If off scaling will occur only along the axis defined by the anchor point and source point.

Editing and Modifying 2D Objects

(Available in all TurboCAD Variants)

This chapter focuses on tools used to change geometry and formats of objects, as well as measuring tools. You can also perform editing commands in **Select** mode, including moving, rotating, scaling, and copying. See Select Edit.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

6.1. 2D Boolean Operations

(Available in all TurboCAD Variants)

Default UI Menu: Modify/2D Boolean Operations

Ribbon UI Menu:

Boolean operations use two or more existing 2D objects to create a new object. Objects can be combined, subtracted, and intersected.

You can display the **Boolean and Facet** toolbar by right-clicking in any toolbar area and selecting **Boolean and Facet**.



Note: 3D objects can also be combined using Boolean operations. See [3D Boolean Operations](#)

The object created as a result of a 2D Boolean operation is considered a **region**. You can use the Selection Info Palette to determine the type of any object or objects you select. See [Selection Info Palette](#).

Selection order can be important, because the properties (color, layer, etc.) of the resulting region are taken from the **f**irst object selected.

2D Add



Default UI Menu: Modify/2D Boolean Operations/2D Add

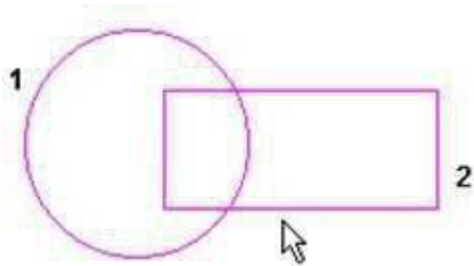
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Ribbon UI Menu:

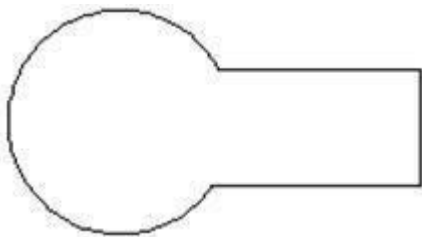
Combines 2D closed objects to form one region. Any overlap between selected objects is removed.

Note: The results of **2D Add** are identical to using **Format / Create Region**.

1. Select two or more closed objects to combine.



2. Select **Finish Selecting** from the local menu or Inspector Bar.



The resulting object is one region. You can verify this by selecting the region.

The selected objects do not have to overlap. If you combine non-overlapping objects, they will still be combined into one region. You can verify this by selecting the region.

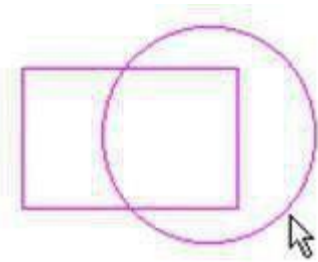
2D Intersect



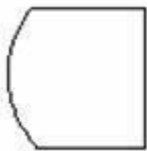
Default UI Menu: Modify/2D Boolean Operations/2D Intersect

Ribbon UI Menu:

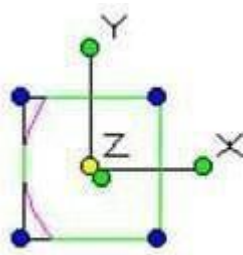
Produces the intersection of two or more closed 2D objects, to form a region.
Select two or more objects whose intersection you want to obtain.



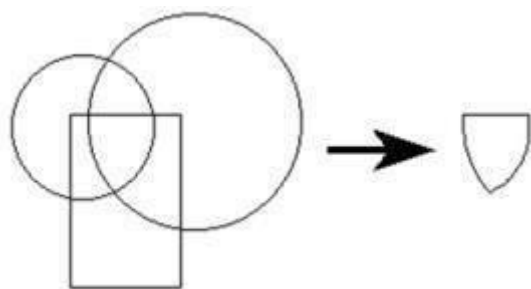
Select **Finish Selecting** from the local menu or Inspector Bar.



The resulting object is one region. You can verify this by selecting the region.



You can select more than two objects, but they must all intersect one another.



Note: If you select objects that do not overlap, the result will be no objects (original objects deleted).

2D Subtract

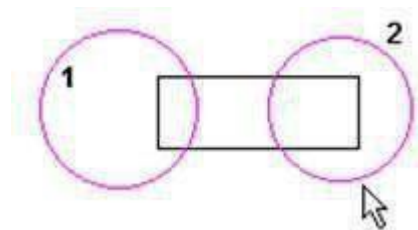


Default UI Menu: Modify/2D Boolean Operations/2D Subtract

Ribbon UI Menu:

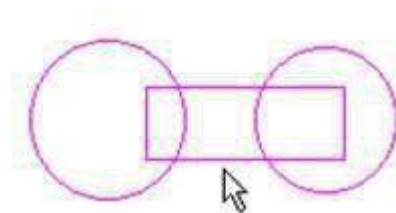
Subtracts one or more objects from one or more objects, to form one region.

1. Select one or more objects to subtract area from.



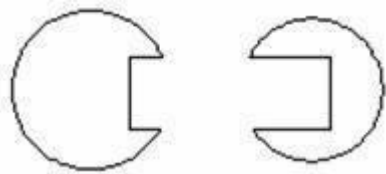
Select **FinishSelecting** from the local menu or Inspector Bar.

Select one or more objects to subtract from the previously selected objects.

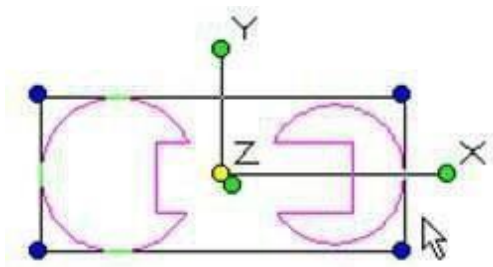


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Select **Finish** from the local menu or Inspector Bar. The second group of objects is removed from the first group.



5. The resulting object is one region. You can verify this by selecting the region.



6.2. Constraining Geometry

(Available in Platinum and Professional)

Default UI Menu: Constraints

Ribbon UI Menu:

Geometric constraints create positional relationships between 2D sketch objects. When used in conjunction with dimensional constraints (see [Constraining Dimensions](#)), you can easily control and update objects and dimensions.

Note: These tools are used for constraining geometry **after** it has been created. If you want to automatically constrain geometry **while** it is created, make sure Auto Add Constraints* is active in the Inspector Bar.



You can display the **Constraints** toolbar by right-clicking in any toolbar area and selecting **Constraints**.



Options for constraints can be found in **Program Setup**; see [Constraints](#). When you activate a constraint, the active layer switches to "CONSTRAINTS." (For details on layers, see [Snap Modes](#).)

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In the Inspector Bar, the **Illuminate Suitable Entity*** option is active by default. This means that only those objects that can be selected for the constraint type will be highlighted when the cursor passes over them. For example, if you are using the **Concentric** constraint, only arcs, circles, and ellipses will be highlighted and selectable.



When constraints are created, a constraint marker is attached to the constrained objects. This marker is on the CONSTRAINTS layer, in the layer's color. This example shows two lines that are constrained to be parallel.



Constraint markers can be selected like other objects. To remove a constraint, simply select and delete its marker.

Note: Constraints of selected objects can be viewed in the Selection Info palette.

See [Selection Info Palette](#)

Auto Constraint

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Default UI Menu: Constraints/Auto Constraint

Ribbon UI Menu:



Automatically applies multiple constraints to a selected set of objects. You can use this tool on compound objects like polygons and polylines; these objects are exploded before constraint are applied. This is a useful tool for applying a fast set of constraints on files created in previous versions of TurboCAD.

Note: If you want to constrain objects while they are created, and not after they are created, make sure **Auto Add Constraints** is active in the Inspector Bar.

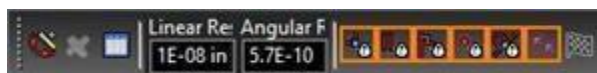


Start with a polyline (see [Double Line-Polyline](#)). Segment 1 is vertical, Segment 2 is horizontal. After Segment 3, switch to arc segments. Arc 4 is tangent to Segment 3, then close the polyline.

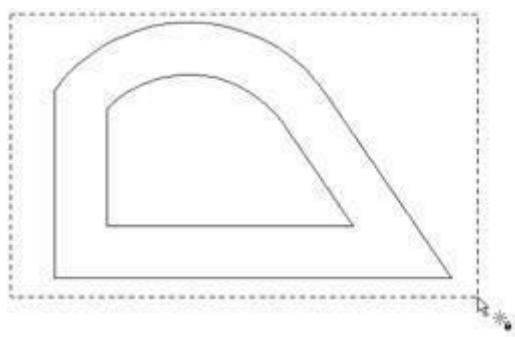
2. Use **Off set** (see [Offset](#)) to make an outward copy of the polyline.

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Activate **Auto Constraint**, and in the Inspector Bar you can select which constraints you want to apply. The available constraints are **Points Coincident**, **Parallel**, **Perpendicular**, **Concentric**, **Tangent**, and **Equal Radius**. In this example, all are selected.



Drag a selection box around all objects you want to constrain.



5. Click **Finish**, or select it from the local menu.



Constraint symbols are displayed for every identified condition.

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You can test the constraints by editing one of the arcs. Use the **Edit Tool** to increase the radius. All constraints are maintained, but the offset is no longer uniform.

Undo, and apply constraints to maintain the offset distance. Use **Equal Distance** (see [Equal Distance Constraint](#)) to make the distance from Segment 3-4 equal to Segment 1-2. Do the same for Segment 5-6.

8. Now if you edit either arc, the uniform offset is maintained.

9. For another update, add a **Connect** constraint at the corner shown.

This changes not only the specified corner, but its offset corner as well.

Auto Dimension

Default UI Menu: Constraints/Auto Dimension

Ribbon UI Menu:

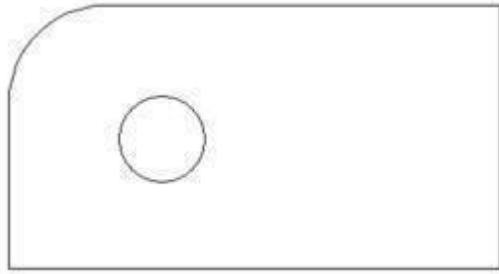


Automatically applies dimensions to a sketch.

Note: For detailed explanations of dimension tools, see [Dimensions](#). **Auto Dimension** is included in this section because it works best in conjunction with geometric constraints.

1. Start with a rectangle and draw a small circle inside it. Fillet one corner (see [Fillet](#)).

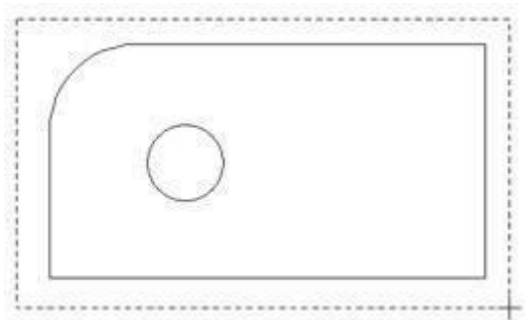
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Activate **Auto Dimension**, and in the Inspector Bar you can select which dimensions you want to identify and label. The available dimensions are **Radius**, **Distance**, **Angular**, and **Parallel**. In this example, all are selected.



Drag a selection box around all objects you want to dimension.



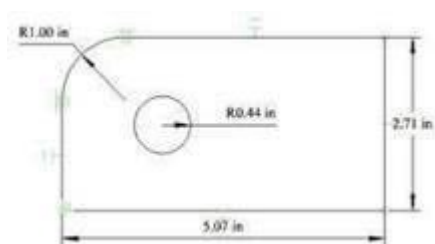
Click **Finish** or select it from the local menu.

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Undo the dimensions. The easiest way to constrain the sketch is to use **Auto Constraint** (see [Auto Constraint](#)). The result in this case includes **Tangent**, **Parallel**, and **Points Coincident** constraints.

Note: You could also get the same constraints if you keep **Auto Add Constraints** active in the toolbar while you are creating geometry (rather than applying constraints after the geometry has been created).

7. Use the same steps as before to apply auto dimensions. This is the result.



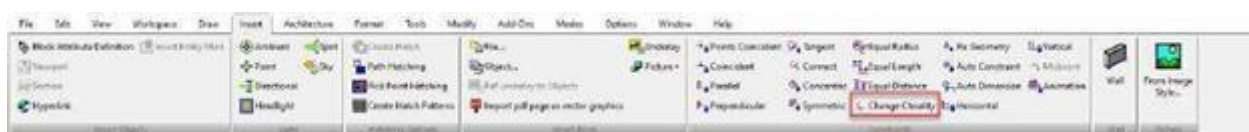
Dimensions created this way are assigned variables, which can be seen and edited in the dimensions's **Properties** (**Format** page), and in the Calculator Palette (see [Calculator Palette - Variables Palette](#)).

Change Chirality Constraint

Default UI Menu: Constraints/Change Chirality

Ribbon UI Menu:

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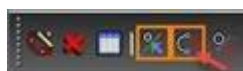


This constraint tool provides two options for correcting tangency of arcs. This tool is useful if arc tangency becomes skewed when applying other constraint, or when editing causes an arc to flip.

1. Start with two lines and an arc like these.

Apply a **Connect** constraint (see [Connect Constraint](#)) at both ends of the arc so that it is tangent to, and trimmed to, both lines. If you want a filleted corner, this result does not have the desired arc tangency.

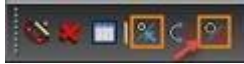
The first option is **Flip Arc**, which changes the arc to its complement. Select this option ...



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4..... and select the arc. The tangency along the vertical line is fixed, but is still incorrect along the horizontal line.

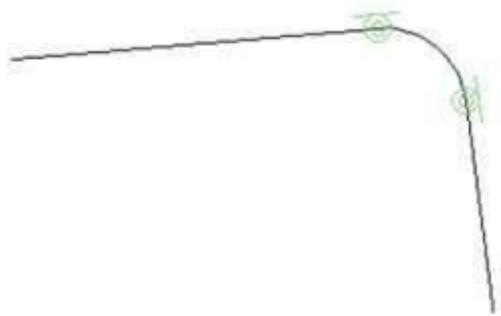
The other option is **Change Chirality**, which changes the arc orientation. Select this option ...



6..... and select the arc.

7. Then select the line whose tangency you want to keep the vertical line.

This is the result - a filleted corner.



Coincident Constraint

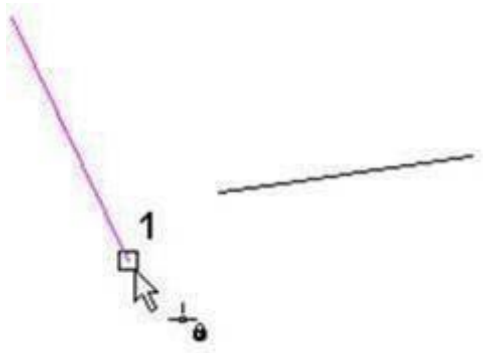
Default UI Menu: Constraints/Coincident

Ribbon UI Menu:

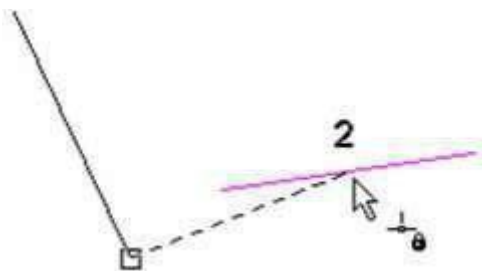


Makes a vertex lie on another vertex, or on another object.

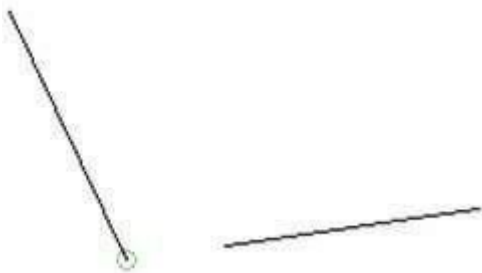
Start with two lines. Activate the **Coincident** constraint and select the vertex you want to constrain.



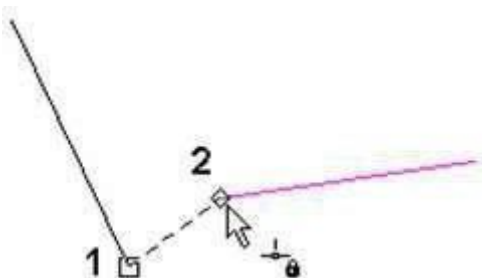
2. Select the line which you want the vertex to lie on.



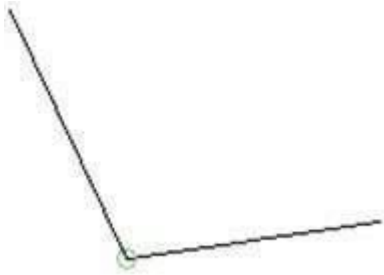
The vertex moves to the line, or to a point on the line's theoretical extension.



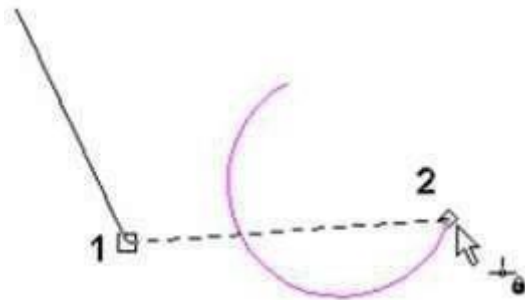
If you constrain the vertex to another vertex ...



... the two vertices will meet.



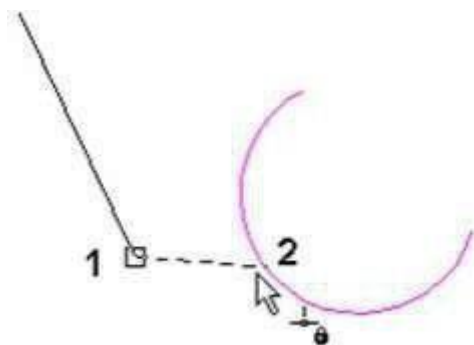
This constraint is similar when using arcs and circles. Start with a line and a circle, and constrain the line endpoint to the arc endpoint.



The endpoints now meet.



4. If you constrain the vertex to the arc itself



... the line joins the arc at the tangent point.



Concentric Constraint

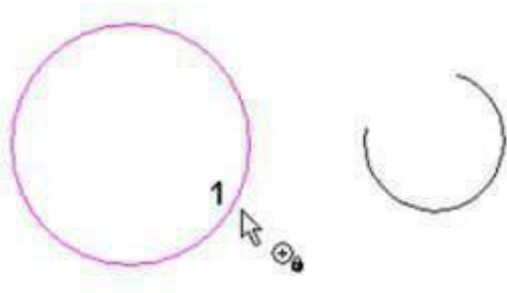
Default UI Menu: Constraints/Concentric

Ribbon UI Menu:

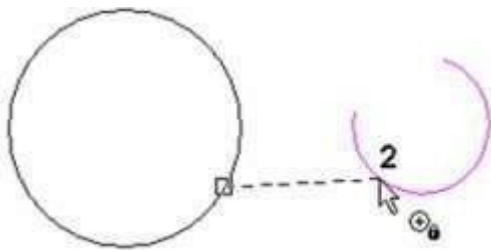


Makes two arcs/circles concentric - sharing the same center point. You can also use this tool to place a point or endpoint at the center of a circle or arc.

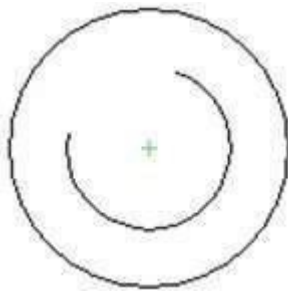
Start with two arcs/circles. Activate the **Concentric** constraint and select the arc/circle you want to make concentric. This object will move to the second.



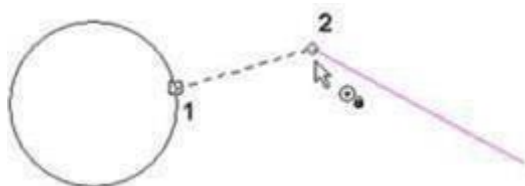
2. Select the second arc/circle.



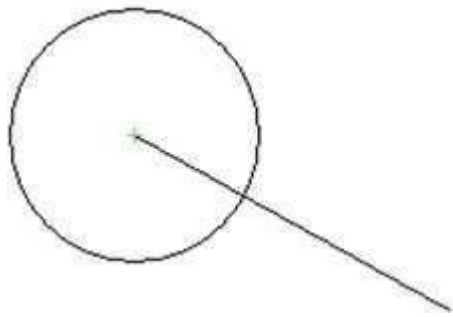
The first arc/circle moves, keeping its orientation, so that it is concentric with the second. The concentric constraint symbol (a plus sign) is added at the point of tangency.



You can also use this tool on a point or line endpoint. Select a circle/arc and the point or endpoint ..



... and the circle/arc moves so that its center meets the endpoint or point.



Connect Constraint

Default UI Menu: Constraints/Connect

Ribbon UI Menu:



Moves, rotates, and trims arcs or lines so that the two selected endpoints will be coincident, and the objects will be tangent. If two lines are selected, they are connected with no regard to tangency, equivalent to the **Points Coincident** constraint.

Start with a line and an arc/circle. Activate the **Connect** constraint and select the endpoint of the line you want to connect to the arc.



2. Select the endpoint of the arc to which the line will be connected.

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The line is moved, in an offset direction, so that it is tangent to the arc/circle. The arc is trimmed as needed. The connect constraint symbol is added at the point of tangency.



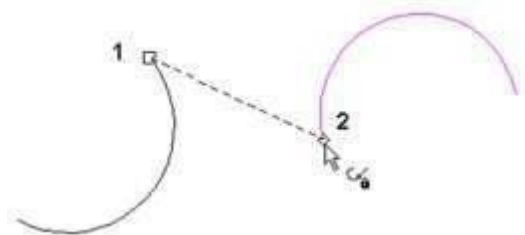
If you select the arc/circle first, and then the line



... the arc moves and the unselected line endpoint remains in place.



You can also select two circle/arcs. Select the arc you wish to move first

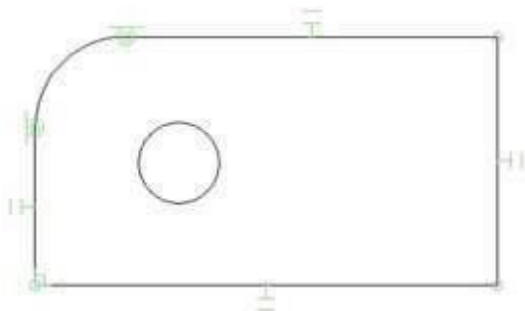


... and the first arc moves to connect to the second arc.

Constraining Dimensions

You can use the Calculator Palette to assign constraints to dimensions, making them dependent on other dimensions or values.

Start with the same shape used to demonstrate **Auto Dimension** (see [Auto Dimension](#)), and apply **Auto Constraints**.



2. Add one more constraint: make the fillet and the circle **Concentric**.

In the Inspector Bar, make sure **Auto Add Constraints** is active. With this active, all dimensions you create will be placed as variables in the Calculator Palette. Otherwise,

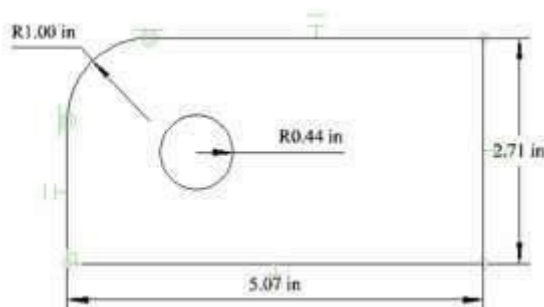
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dimensions will be created but they cannot be constrained, or used as constraints for other dimensions.



The dimension types that can be constrained are **Orthogonal**, **Parallel**, **Distance**, **Angular**, **Radius**, and **Diameter**.

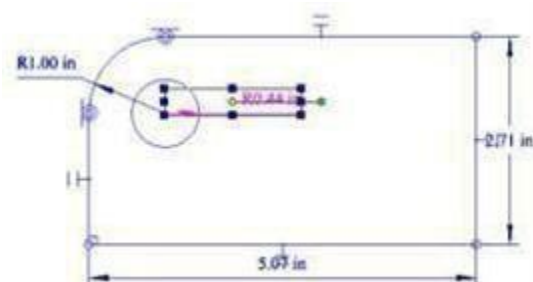
4. Apply **Auto Dimension**, and you should get the following four dimensions:



Make sure the Calculator Palette is open (see [Calculator Palette - Variables Palette](#)). The four dimensions you created are listed here, with a variable name assigned to each.

Calculator		
F[...]		
Variable	Value	Formula
Var1	1.00	
Var2	0.44	
Var3	2.71	
Var4	5.07	

6. **Select** one of the dimensions, and the corresponding item is highlighted in the list.



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Variable	Value	Formula
Var1	1.00	
Var2	0.44	
Var3	2.71	
Var4	5.07	

Note: The dimension's variable can also be seen and edited in the **Format** page of its **Properties**.

Click the variable for the hole radius dimension, and change its name to something meaningful, like "HoleRad." Avoid using spaces in variable names.

Var1	1.00	
HoleRad	0.44	
Var3	2.71	
Var4	5.07	

Change the remaining variable names.

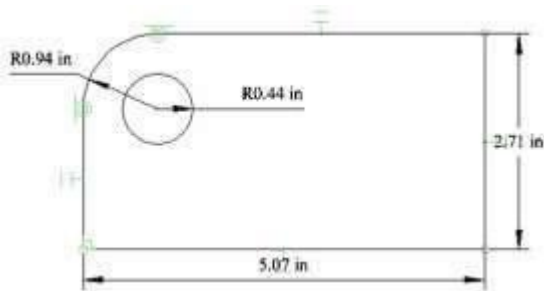
Variable	Value	Formula
FilletRad	1.00	
HoleRad	0.44	
Width	2.71	
Length	5.07	

One way to constrain a dimension is to base it on another dimension. For the fillet radius, click inside the **Formula** field and enter a formula that makes the fillet radius a set amount (0.5" in this example) larger than the hole radius.

Variable	Value	Formula
FilletRad	0.49	HoleRad+0.5
HoleRad	0.44	
Width	2.71	
Length	5.07	

Press Enter and the fillet radius updates based on the current value of the hole radius. The drawing updates as well.

Variable	Value	Formula
FilletRad	0.94	HoleRad+0.5
HoleRad	0.44	
Width	2.71	
Length	5.07	



You can also constrain dimensions to have an exact value, such as the value below for "Length." Width can also be constrained to be a constant factor (such as 1/2) of the length.

You can also define a new variable independent of any of the current dimensions. "PinRad" is the radius of the pin that fits in the hole, and is assigned a numerical value.

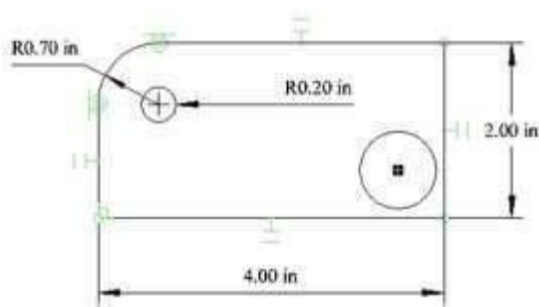
Variable	Value	Formula
FilletRad	0.94	HoleRad+0.5
HoleRad	0.44	
Width	2.00	Length/2
Length	4.00	
PinRad	0.25	

Once "PinRad" is defined, the hole radius can be made to a set amount larger than the pin.

If "PinRad" is updated, then both "HoleRad" and "FilletRad" are automatically updated.

You can also constrain new dimensions, as long as **Auto Add Constraints** is active. Create another circle, and inside this circle add a point (see [Point](#)). Make the point and the circle **Concentric**.

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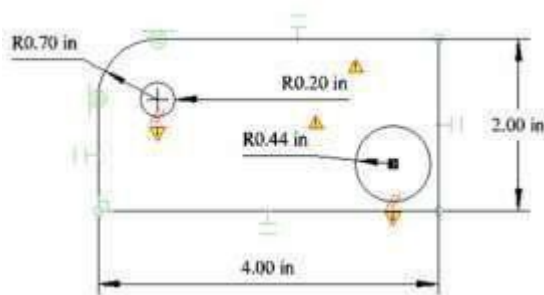
Note: The point is added because a circle's center point is not identified as an object; a physical point must be placed there.

Add a **Radius** dimension to the new circle, and its variable appears in the constraint list.

If you try placing an **Equal Radius** constraint on the two circles, you will receive an error message:



When you click OK, the problem dimensions are indicated.



This occurs because the second circle is considered to already have a set radius, once it is dimensioned. So constraining it to be equal to the first circle is contradictory. You could have assigned the **Equal Radius** constraint before assigning the dimension.

Undo, and use a dimensional constraint to set the radius, making it equal to "HoleRad."

Constraining Pattern-Copied Objects

This section refers to objects copied in patterns (lines, linear arrays, radial arrays), using the **Copy Entities** tools (see [Copying Objects](#)).

When you copy objects in a pattern, you can use **Auto Constraints** and dimension variables to control spacing, angles, etc. This example uses a circle copied into a linear array.

1. Before you begin, make sure **Auto Add Constraints** is active in the Inspector Bar.

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Create a rectangle and add a small circle. The rectangle should have parallel and coincident constraints assigned automatically.

Select the circle and use **Fit Array Copy** to copy the circle into a grid. See [Fit Array Copy](#)

4. Assign some **Orthogonal** dimensions as shown. (See [Creating Dimensions](#)).

Open the Variables palette (F2) and the dimensions should be shown, identified by a variable. In this example, the variables were assigned names (L, E1, E2).

Variable	Value	Formula
L	3.540000	
E1	0.510000	
E2	1.070000	

Note: For more information on working with variables, see [Calculator Palette - Variables Palette](#)

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Create a formula so that the two edge dimensions will be equal. In this case, the variable E1 was used as the formula for E2.

Variable	Value	Formula
L	3.540000	
E1	0.510000	
E2	0.510000	E1

The dimensions update, and the spacing between copied objects remains constant. However, in this case, the linear array does not have a 90 degree angle.

Used the Horizontal constraint on the bottom edge of the rectangle.

To fix this, activate the **Parallel** constraint and click first the array (horizontal) constraint line, then the top line of the rectangle. Repeat if necessary to make the array perpendicular vertically.

Now the array is perpendicular again.

Equal Distance Constraint

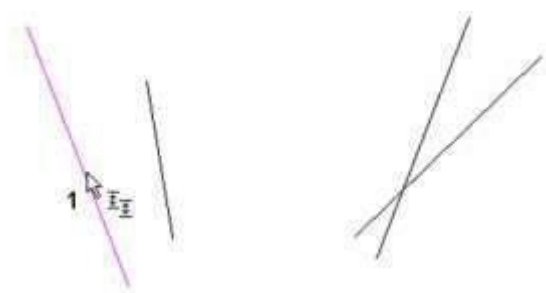
Default UI Menu: Constraints/Equal Distance

Ribbon UI Menu:

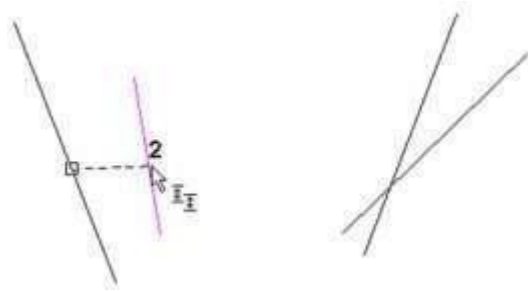


Sets the distance between two lines to the same between two other lines. Sets of lines are also made parallel.

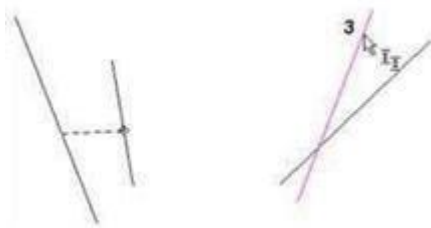
Start with four lines. Activate the **Equal Distance** constraint and select the first line in the first set.



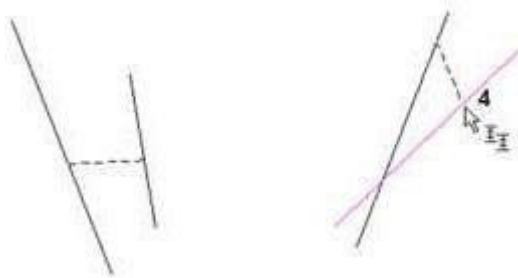
2. Select the second line in the first set. This establishes the distance for the next set.



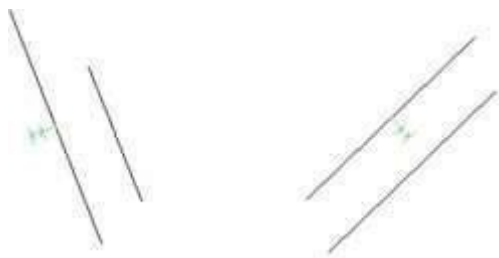
3. Select the first line in the second set.



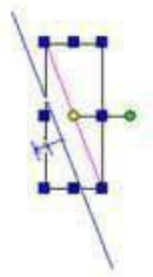
4. Select the second line in the second set.



The first set and second set of lines are made parallel. The distance between the lines in the second set matches the distance in the first set.



5. If you edit any line, the equal length constraint will be maintained.



You can also apply this constraint to a row of lines or points. Select the first and second lines.

7. The third line is the same as the second, then select the fourth.

The result is three lines, parallel and separated by the same distance.

Equal Length Constraint

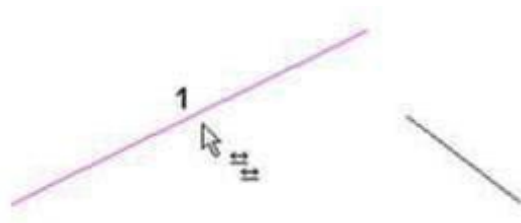
Default UI Menu: Constraints/Equal Length

Ribbon UI Menu:



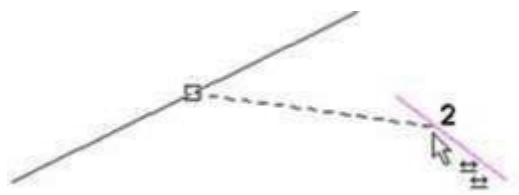
Makes the first selected line have the same length as the second line.

Start with two lines. Activate the **Equal Length** constraint and select the line whose length you want to change.



2. Select the line whose length you want to assign to the first line.

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The first line's length changes to match the second. The equal constraint (an equal sign) appears.



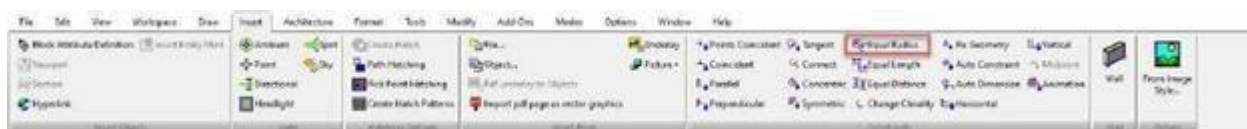
3. If you edit either line, the equal length constraint will be maintained.



Equal Radius Constraint

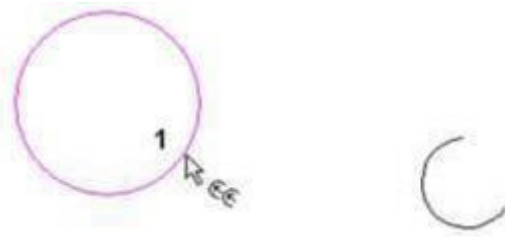
Default UI Menu: Constraints/Equal Radius

Ribbon UI Menu:

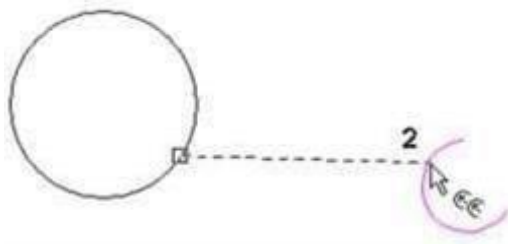


Makes the first selected circle/arc have the same radius as the second circle/arc.

Start with two circles/arcs. Activate the **Equal Radius** constraint and select the arc whose radius you want to change.



2. Select the arc whose radius you want to assign to the first arc.



The first arc's radius changes to match the second. The equal constraint (an equal sign) appears.

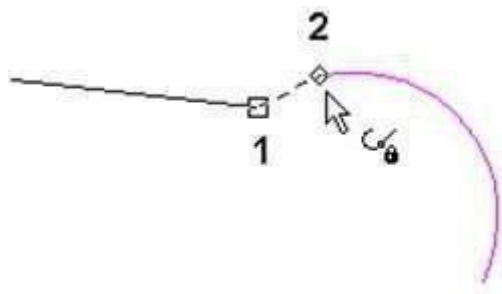


3. If you edit either arc, the equal radius constraint will be maintained.

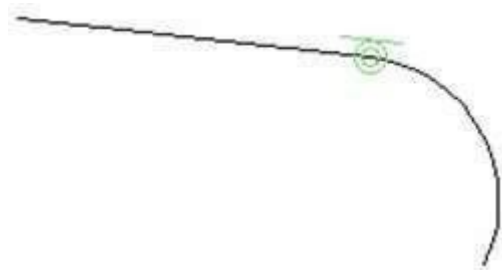
Fix Geometry Constraint



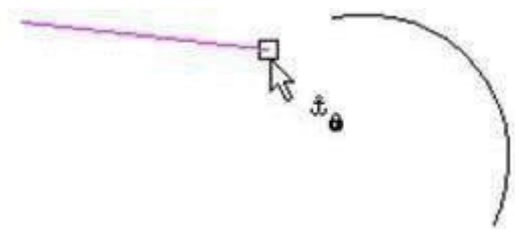
For another example of this tool, start with a line and arc and apply a **Connect** constraint (see [Connect Constraint](#)).



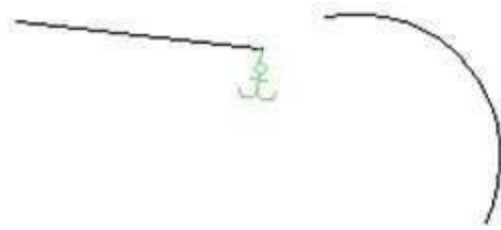
The result is that the line maintains its orientation and is tangent and trimmed to the arc.



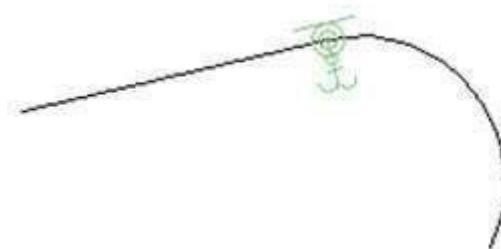
5. Undo this constraint, and apply a **Fix Geometry** constraint to the line's endpoint.



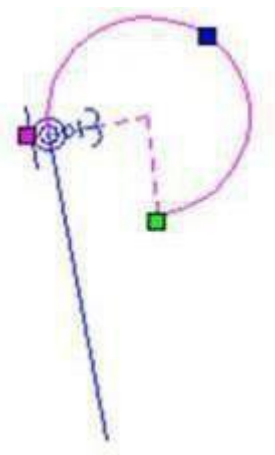
Now the anchor symbol is attached to the endpoint.



Now apply the **Connect** constraint. The endpoint stays fixed, and the orientations of the line and arc are modified.



If you edit the line or arc, such as changing start or end angles, the fixed point will not move, and the other objects will move or rotate accordingly.



Horizontal-Vertical Constraint

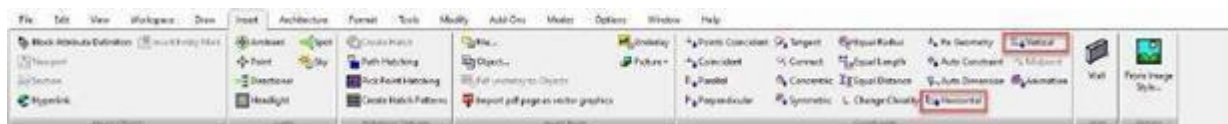


Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

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Default UI Menu: Constraints/Horizontal, Constraints/Vertical

Ribbon UI Menu:



Forces a line to be horizontal or vertical.

Start with a line.

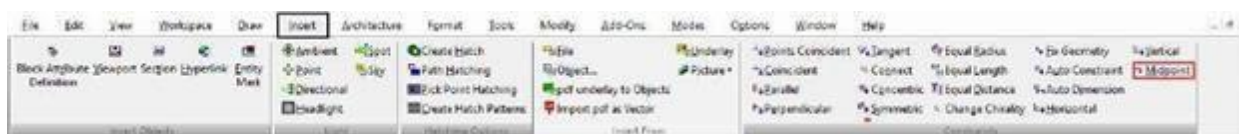
Activate the **Horizontal** or **Vertical** constraint.

Click the line, which becomes horizontal or vertical.

Midpoint Constraint

Default UI Menu: Constraints/Midpoint

Ribbon UI Menu:



Align a point or the end of a line with the midpoint of a selected line.

Start with a line and a constrained rectangle. Activate the **Midpoint** constraint and select the end of the line you want to align to the midpoint of the side of the rectangle.

2. Select the side of the rectangle to align to its midpoint.

The end of the line aligns with the midpoint of the rectangles side.

Repeat the procedure with the other end of the line and the opposite side of the rectangle.

Parallel Constraint

Default UI Menu: Constraints/Parallel

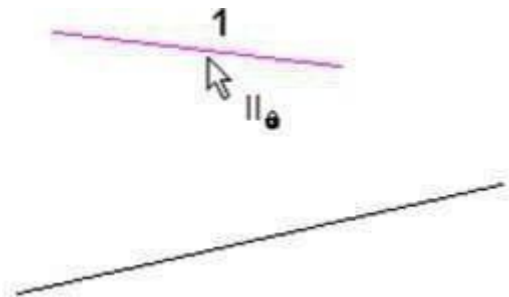
Ribbon UI Menu:

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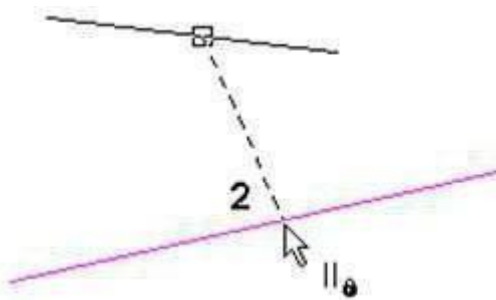


Makes the first selected line parallel to the second selected line.

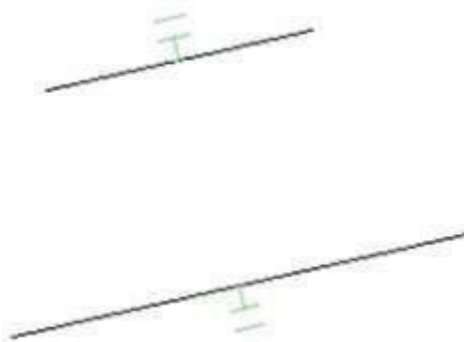
Start with two lines. Activate the **Parallel** constraint and select the line you want to be made parallel.



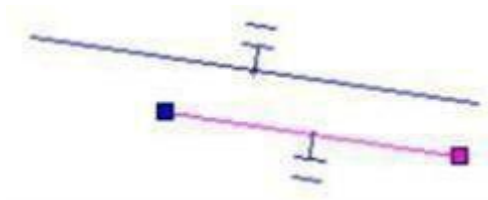
2. Select the line to which you want the first line to be parallel.



The lines are parallel, and the parallel constraint symbol is added to both lines.



3. If you edit either line, the parallel constraint will be maintained.



Perpendicular Constraint

Default UI Menu: Constraints/Perpendicular

Ribbon UI Menu:



Makes the first selected line perpendicular to the second selected line.

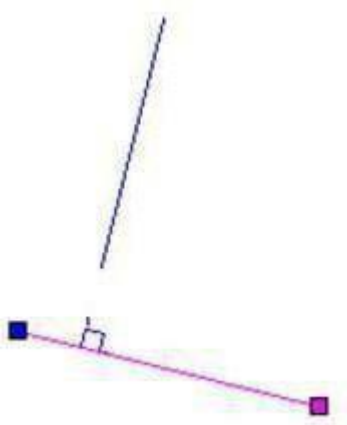
Start with two lines. Activate the **Perpendicular** constraint and select the line you want to be made perpendicular.

2. Select the line to which you want the first line to be perpendicular.

The lines are perpendicular, and the perpendicular constraint symbol is added at the point where the lines, or their extensions, meet.

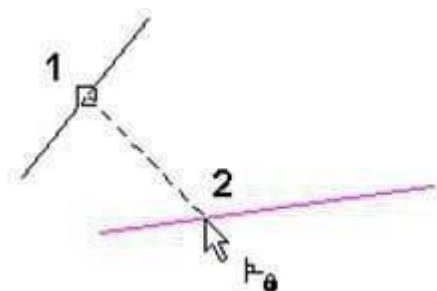


3. If you edit either line, the perpendicular constraint will be maintained.



If you apply this constraint to lines that do not intersect

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... the perpendicular constraint symbol will appear along the extension of one or both lines.



Points Coincident Constraint

Default UI Menu: Constraints/Points Coincident

Ribbon UI Menu:



Moves a point or endpoint to another point.

1. Start with two lines and an arc.

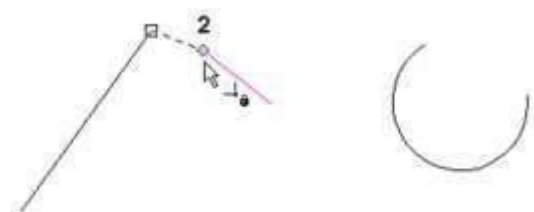


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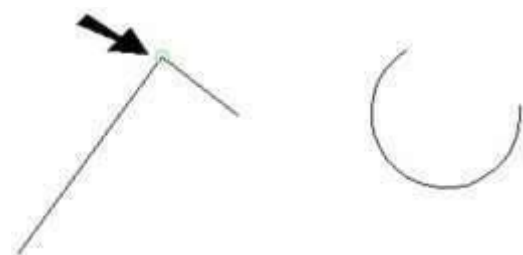
2. Activate the **Points Coincident** constraint, and click an endpoint of the first line.



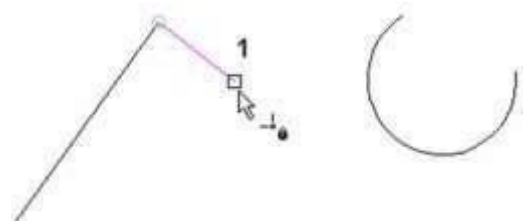
3. Click an endpoint of the second line.



The first line is moved so that the two points are coincident. The length and orientation of the line is maintained, and a constraint symbol appears at the coincident point.



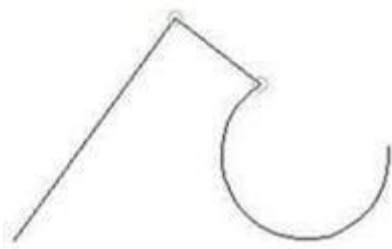
4. Create another constraint between Point 1



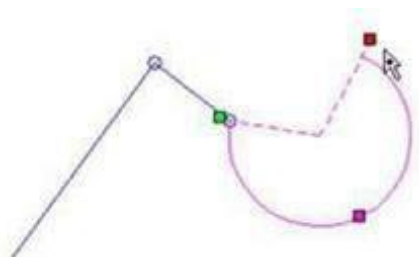
5. . . . and Point 2 an endpoint of the arc.

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Both lines move, and their lengths and orientations are maintained.



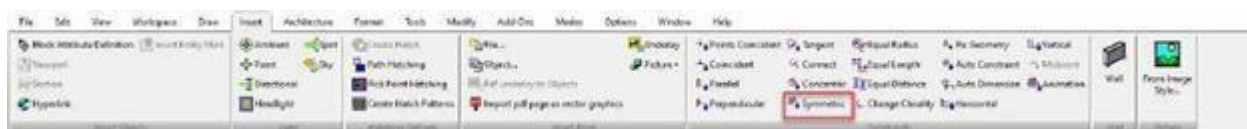
If you edit any of the objects, such as the arc in this case, the coincident constraint will be maintained, and the other objects will keep their size and orientation (when possible).



Symmetric Constraint

Default UI Menu: Constraints/Symmetric

Ribbon UI Menu:



Makes two objects, or object endpoints, symmetric with respect to a symmetry axis.

Start with three lines - one of which is the symmetry line. In some cases the symmetry line will move as a result of this constraint, so if you want it to remain in place, fix it with a constraint such as **Fix Geometry**.

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2. Activate the **Symmetric** constraint and select the first line.

3. Select the second line.

4. Finally, select the symmetry line.

The first two lines are now symmetric with respect to the third.

You can also use this constraint on endpoints. Select Vertices 1 and 2, then select the symmetry line.

The lines themselves keep their orientation, but their endpoints are symmetric.

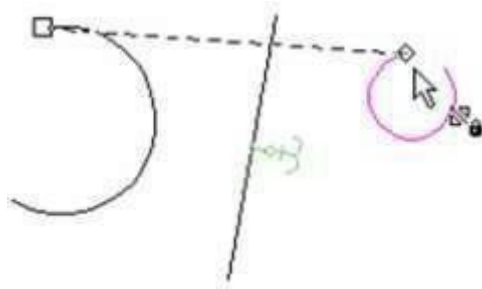
To make both lines completely symmetric, apply the **Symmetric** constraint to the other two endpoints.

This constraint can also be used on circles and arcs. If you select two arcs (not at their endpoints)

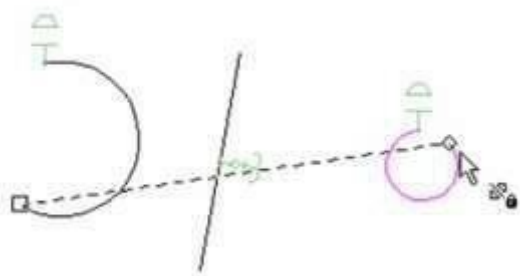
... they will become symmetric and have equal radii.

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Like with lines, you can apply the **Symmetry** constraint at both sets of endpoints. First the upper endpoints ...

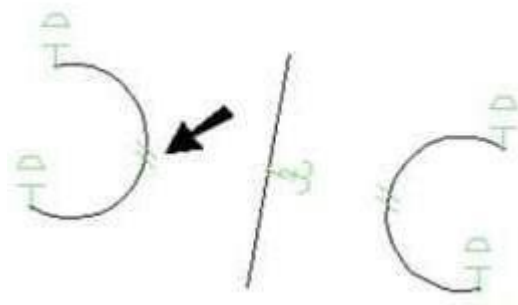


9..... then the lower endpoints.



The result is that both sets of endpoints are symmetric, but the orientations of the arcs are still different, because they have different radii.

10. You can make the arcs completely symmetric by adding an **Equal Radius** constraint.



Note: You could get the same results if you apply three **Symmetric** constraints - one to the arcs themselves, and one for both sets of endpoints.

Tangent Constraint

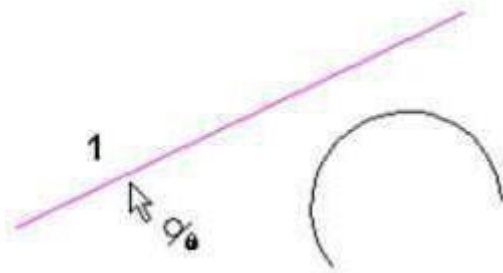
Default UI Menu: Constraints/Tangent

Ribbon UI Menu:

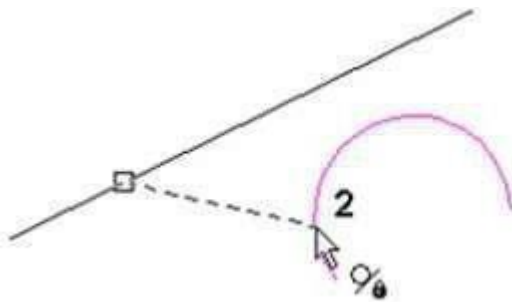


Makes an arc or circle tangent to another arc or circle, or to a line.

Start with a line and an arc/circle. Activate the **Tangent** constraint and select either object (when one object is a line, the selection order does not matter).



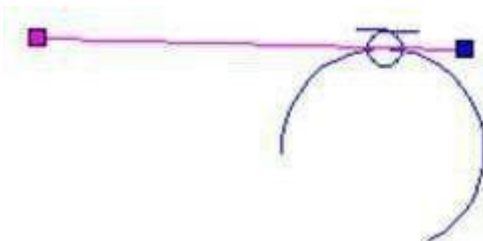
2. Select the other object.



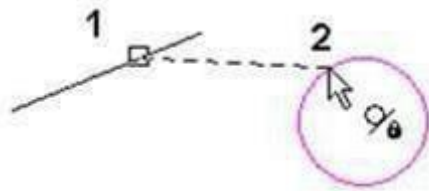
The line is moved, in an offset direction, so that it is tangent to the arc/circle. The tangent constraint symbol is added at the point of tangency.



3. If you edit either object, the tangent constraint will be maintained.



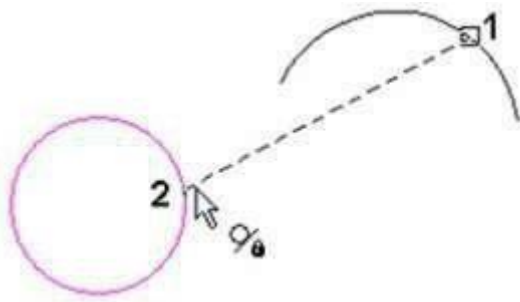
If you apply this constraint to a line whose offset will not meet the arc/circle



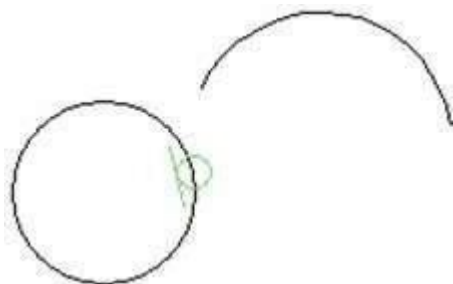
. . . the line is moved so that its extension is tangent to the circle.



You can also make two arcs/circles tangent to one another. In this case, the first selected object will move to meet the second selected object. If the arc does not meet the other arc/circle



. . . the arc is moved so that its extension is tangent to the arc/circle.



Constraint Animation

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Default UI Menu: Constraints/Animation

Ribbon UI Menu:

The Constraint Animation tool enables the user to animate dimensions in TurboCAD drawings. With the help of animation, the user can visualize the boundary of moving parts in a mechanical design. The user can play the animation in a loop, pause the animation to verify different parameters, play again from the same place, increment or decrement step by step the animation, choose the delay in animations and also select the animation steps to suit his/her needs.

Make sure that you select the dimension from the drawing. To see if the dimension is a valid one, you can see if it has been added to the Calculator palette. You can also select the variable from the Calculator palette in order to animate it

To apply Animation:

Draw any 2D object with Auto- Constraints enabled

Then, add any dimension type on that drawn 2D object

Select Animation Constraint from the menu

Click on the valid dimension added to the 2D object

The Constraint Animation Dialog then opens

Set the Constraint Parameters and then press Play to view the animation.

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Start: Define the starting dimension length, distance, size, or angle for the animation

End: Defines the ending dimension length, distance, size, or angle for the animation

Steps: Defines the number of steps/jumps required from start to end points

Delay: Defines the time delay per step

Play: Plays the animation

Stop: Stops the animation

Loop Animation: Loops the animation back to the Start when reaching the End

6.3. Detail Section



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Clip/Detail Section

Ribbon UI Menu:

The section detail tool allows you to create a 2D detail section of 2D elements for part of your drawing. The Detail Section tool ignores 3D objects entirely.

Select the Detail Section tool.

In the Inspector bar specify the scale for the detail section.

Select an existing, closed 2D shape (polyline, curve, circle) to act as the cutting contour for the section.

Drag the detail to the desired location.

Note: Dimensions and text that are not entirely within the cutting contour will be ignored/excluded from the resulting detail section. Hatches within the detail area are not scaled in the resulting detail section.

Local menu option:



Trim by circle: If you select this option you define the cutting contour drawing a cutting circle. Simply select the center point of the circle, then the radius. After that you just drag the resulting detail to its new location.

6.4. Edit Tool



(Available in all TurboCAD Variants)

Default UI Menu: Edit/Edit Tool

Ribbon UI Menu:

Nodes are hidden markers identifying elements of geometric objects. The **Edit Tool** enables you to reshape objects by editing their nodes. You can edit a single node or multiple nodes - even a group of nodes belonging to different objects.

Note: This section focuses on editing nodes of 2D objects. For 3D objects, see [Edit Tool in 3D](#). For editing tables, see [Changing Rows and Columns, Merging Cells](#). And for editing walls, see [Editing Walls](#).

There are several ways to activate the **Edit Tool**: Click **Edit Tool** on the **Select** toolbar, then select the object to edit.



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Select **dit Tool**, then select the object to edit. Select the object that you wish to edit, then select **Edit Tool** from the local menu. You can also click the icon on the Inspector Bar.



Select the object to edit, then press Ctrl and select it again. Access the **Edit Tool** from the Selection Info Palette. See [Selection Info Palette](#). In **Edit Tool** mode, the cursor becomes an arrow with a dot.



When the node editing is finished, you can return to **Select Edit** mode by selecting it from the local menu, by clicking the icon on the Inspector Bar, or by pressing Esc.



You can then exit **Select** mode by clicking outside the selection set, or by pressing Esc again.

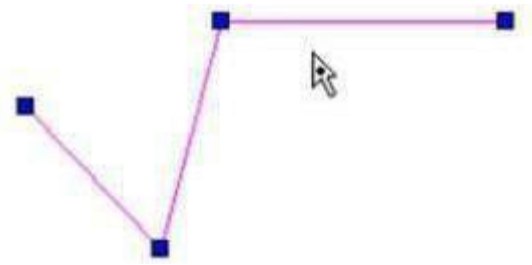
Note: If you need to node edit an object that is part of a group, explode the group first (**Modify / Explode**). For an object that is part of a block, you can node edit the object in **Edit Content** mode. See [Editing a Block](#).

Adding a Node

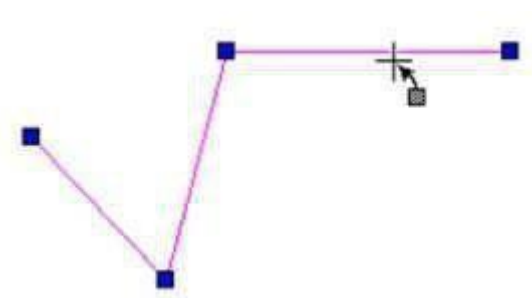
Adds nodes to lines and line segments, including objects created using double lines and multi-lines. You can also add nodes to Bezier curves, splines, and sketches.

1. Use the **Edit Tool** on the object to which you want to add a node.

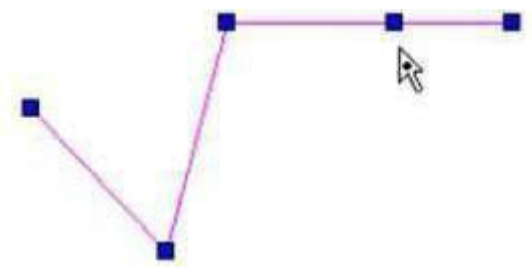
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Press Ctrl while placing the cursor where you want to add the node. The cursor has a + symbol.

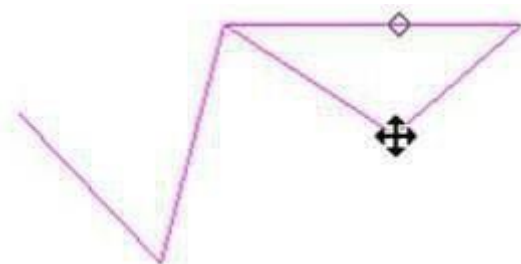


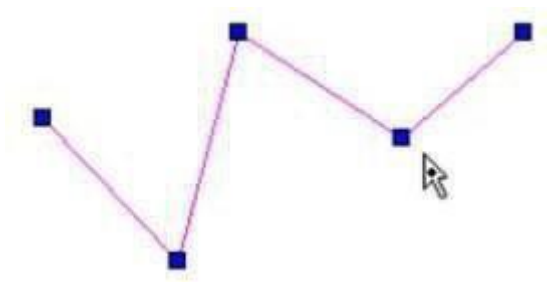
3. Click to add the node.



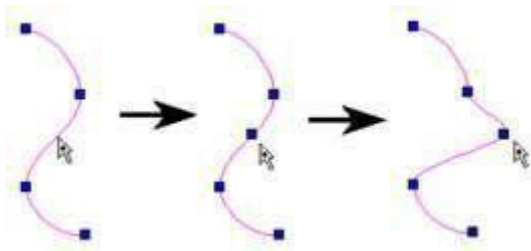
(You could also place the cursor where you want to add the node, then select **Add Node** from the local menu.)

4. You can now drag the new node to create a new line segment.





The same method can be used to add nodes to curves.



Insert Arc

The Insert Arc function of the Edit Tool allows you to insert an arc at the location of most nodes of a polyline.

1. Start the Edit Tool, then hover over a node.

2. Right-click to open the Local menu, and select Insert Arc.

3. The arc is inserted.

4. The radius of the resulting arc is defined by the Radius field of the Inspector bar.

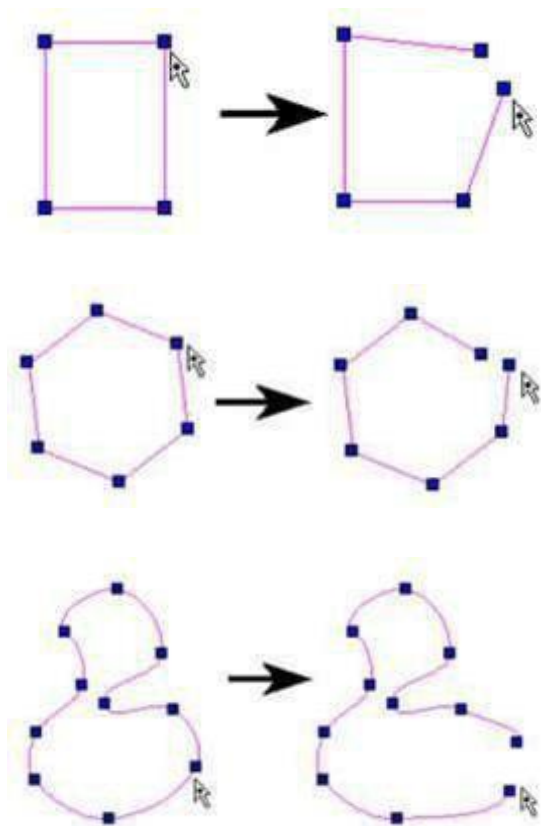
Breaking (Opening) Closed Objects

Breaks a closed object (polygon, closed polyline, rectangle, closed curve, etc.), turning it into a polyline.

Use the **Edit Tool** on the closed object you want to break.

Right-click on the node at which you want the break, and select **Break** from the local menu.

The object is now an open polyline, with two end nodes at the node where you broke the object.



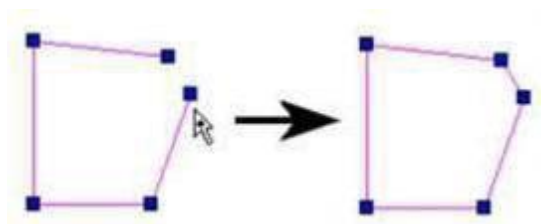
Closing Open Objects

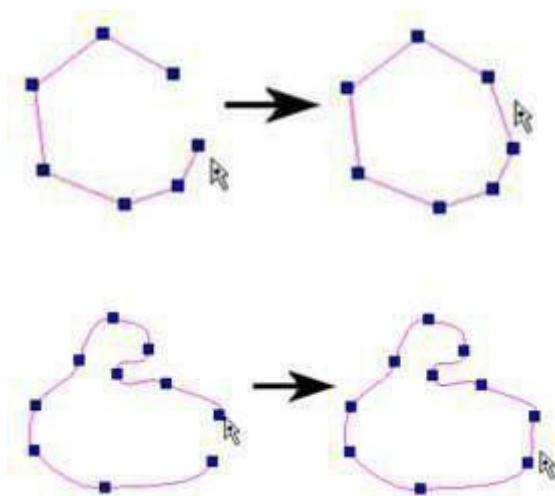
Closes an open object (polyline, open curve), turning it into a closed polyline, curve, or polygon.

Use the **Edit Tool** on the open object you want to close.

Right-click on the node at which you want to close the object, and select **Close** from the local menu.

A line segment is created between the node you selected and the other endpoint.





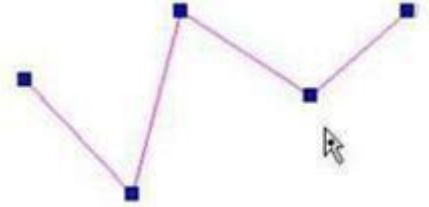
Note: If the two endpoints of the polyline share the same location, the object will simply be closed geometrically. If you simply snap the endpoints of a polyline together, the resulting object will only appear to be closed but is still considered open. This means, for example, that the object cannot be filled with a hatch pattern. In order to geometrically close the object, you must use this feature.

Deleting Nodes

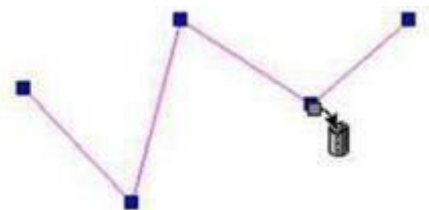
Deletes nodes from lines and line segments, including objects created with double lines and multi-lines. You can also delete nodes from Bezier curves, splines, and sketches. You cannot delete a node that is needed to define an object as a line segment or polygon. This means that you cannot delete the endpoint of a single line segment, and you cannot delete any node of a triangle. If you delete the endpoint of a polyline, you will delete the entire line segment that terminates at that point.

To delete a single node, use the **Edit Tool** on the object from which you want to delete a node.

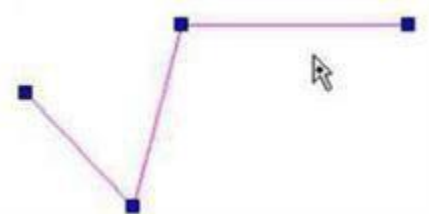
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Press Ctrl while placing the cursor where you want to delete the node. The cursor becomes a "trash can" symbol.



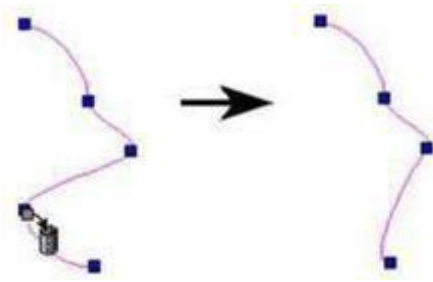
Click to delete the node. The adjacent nodes will be connected to form a new segment.



You can also use the local menu to delete one or more nodes. Within the **Edit Tool**, drag a selection window around the node or nodes you want to delete. Selected nodes turn from blue to magenta. Then select **Delete Node(s)** from the local menu.

Note: If you remove a node from a polygon, the polygon will remain closed, but will become an irregular polygon.

You can also delete nodes from Bezier curves, splines, and sketches.

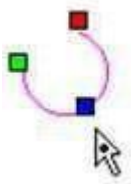


Editing Circles, Ellipses, and Arcs

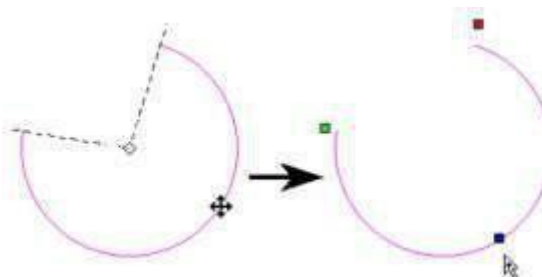
In addition to adding, deleting, or moving nodes, there are several **Edit Tool** features unique to circular and elliptical objects.

Changing the Radius of a Circle or Arc

1. Use the **Edit Tool** on the circle or arc whose radius you want to change.



Click and move the blue handle to resize the object. You can also enter the new radius in the Inspector Bar.

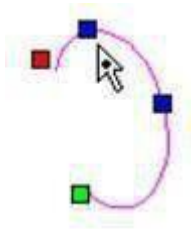


Changing the Axes of an Ellipse or Elliptical Arc

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Use the **Edit Tool** on the ellipse or elliptical arc whose axes you want to change.

There are two blue handles, one for the major axis and one for the minor.



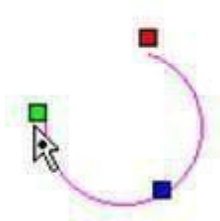
Click and move the relevant blue handle to resize the axis. You can also enter the new axis length in the Inspector Bar.

To scale the object while maintaining its aspect ratio (major axis to minor axis), hold the Shift key while dragging either blue handle.

Changing Start and End Angles

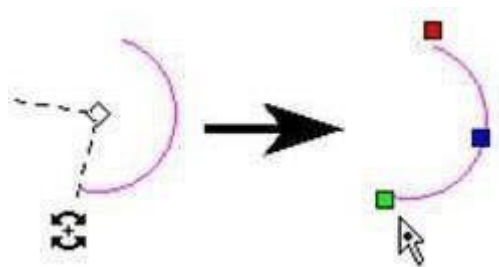
Use the **Edit Tool** on the object whose start and end angles you want to change.

The green handle is used to change the start angle, and the red handle is for the end angle.



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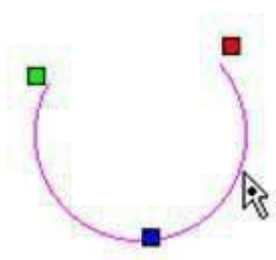
Click and move either angle handle to resize the angles. You can also enter the new angles in the Inspector Bar.



Dividing an Arc

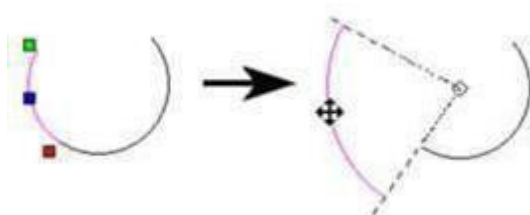
Adds nodes to an arc or circle, breaking it into separate arcs. The resulting arcs can be edited just like an arc.

1. Use the **Edit Tool** on the arc or circle you want to divide.



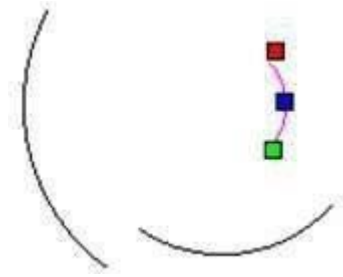
Right-click over the arc you want to divide, and select **Arc Divide** from the local menu. Enter the number of arcs in the inspector bar.

The arc is divided into the specified number of arcs. Each resultant arc can be node edited separately.



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To node edit, a different arc, simply click on it while still in the **Edit Tool**.



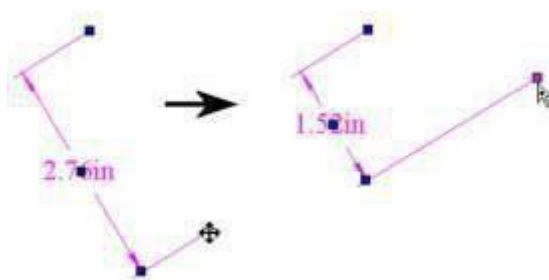
Tip: To divide an arc segment that is part of a polyline, right-click on the green node and select **InsertArc**. This divides the arc into two equal segments.

Editing Dimensions

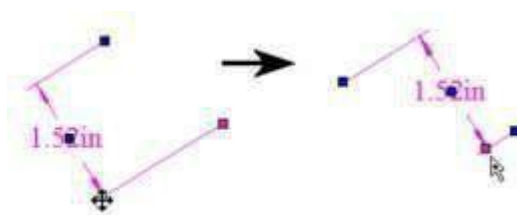
For details on creating dimensions, see [Dimensions](#)

When using the **Edit Tool** on a dimension, there are several nodes for editing: one at each extension line, one on the dimension line itself, and one on the text. Each can be moved to change the dimension.

Move the node on an extension line to change its length and/or position. If this change leads to a new dimension value, the new value will appear after the node is moved.



Move the node on the dimension line to change the position of the dimension text.



Move the node on the dimension text to move the text. How the text is moved depends on the **Text Movement** option set on the **Format** page of the dimension's **Properties** (see [Dimension Properties](#)) **Keep Dimension Line with Text:** The dimension line will move with the text.

Move Text, Add Leader: The moved dimension text will be attached to the existing dimension line by a leader.

Move Text, No Leader: Only the dimension text will move.

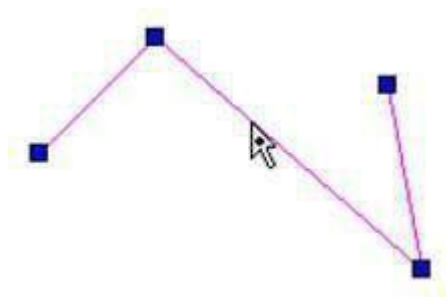
Editing Linear Objects

In addition to adding, deleting, or moving nodes, there are several **Edit Tool** features unique to linear objects.

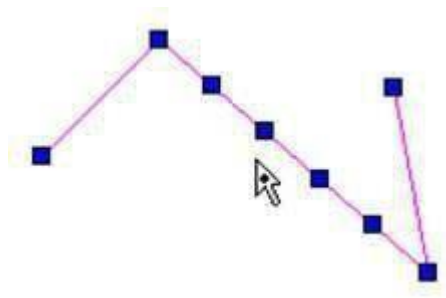
Dividing a Line Segment

Adds nodes to a line or line segment, breaking it into a polyline composed of segments of equal length. Double lines and multi-lines can also be divided.

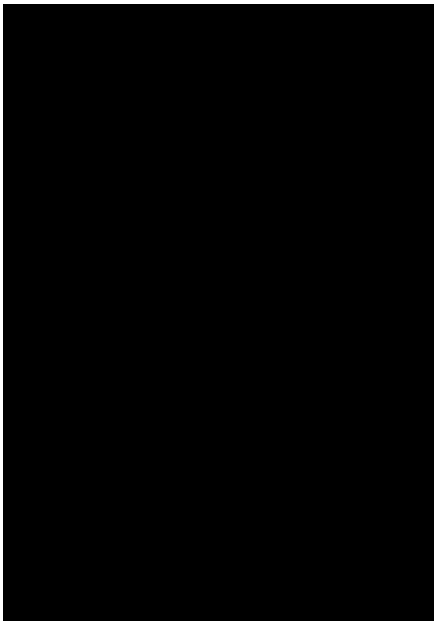
1. Use the **Edit Tool** on the object you want to divide.



Right-click over the segment you want to divide, and select **Divide Segment** from the local menu. Enter the number of segments in the inspector bar. The selected segment is divided into the specified number of segments.



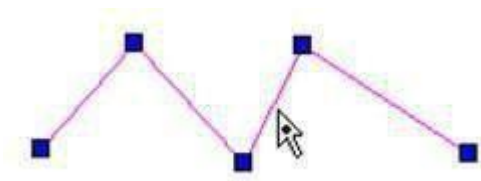
3. You can now drag any of the new nodes to create new line segments.



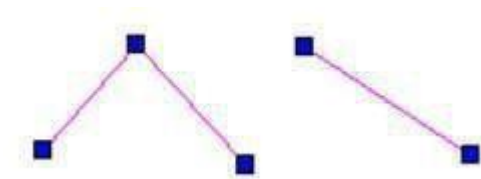
Hiding or Showing a Line Segment

Applies only to single-line segments.

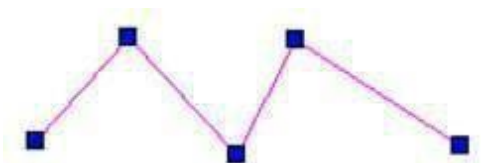
1. Use the **Edit Tool** on the object whose segment you want to hide.



Right-click over the segment you want to hide, and select **Hide Segment** from the local menu. The segment is no longer displayed.



To show the segment again, select **Show Segment** from the local menu.



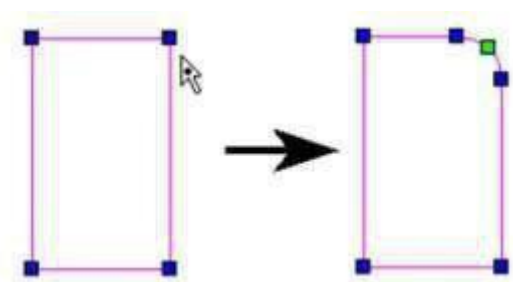
Filleting Two Line Segments

Adds an arc connecting two line segments, creating a fillet.

Use the **Edit Tool** on the multi-segmented object you want to fillet (polyline, rectangle, polygon, etc.).

Enter the fillet radius in the Inspector Bar.

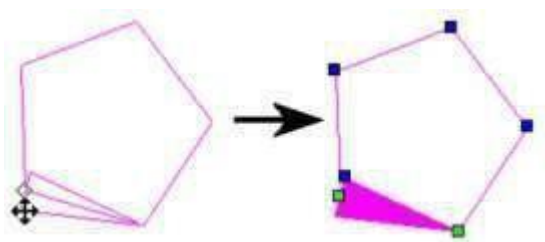
Right-click on the node to fillet and select **Insert Arc**.



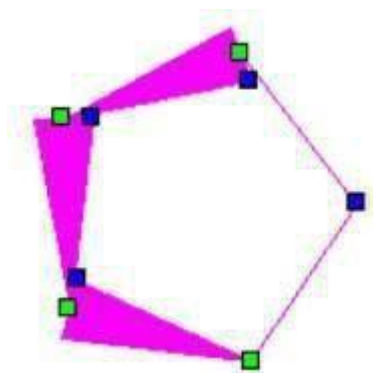
Modifying Line Widths

Use the **Edit Tool** on the line object (line, rectangle, polygon, etc.) whose width you want to change. Arc segments of polylines can be edited as well.

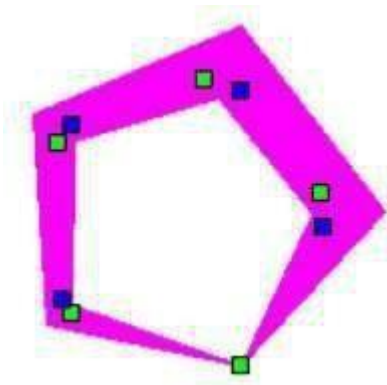
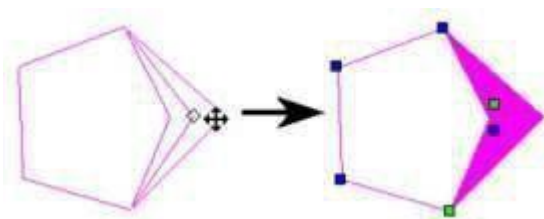
Right-click and select *E. Drag an endpoint to widen it, or enter the width in the Inspector Bar.



With this method, each endpoint will be sized individually, with no relation to any adjacent segments.



To modify the width of all adjacent segments, right-click and select Edit Line Width.

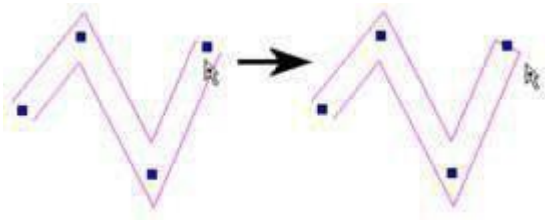


Closing and Opening Endpoints of Double Lines

Creates or removes a start or end cap of a double line. This can also be controlled in the double line's **Properties** - see [Double Line Properties](#)

Use the **Edit Tool** on the double line object whose endpoint you want to close.

Right-click on the end node, and select Close Double End.



3. To open the endpoint, select Open Double End.

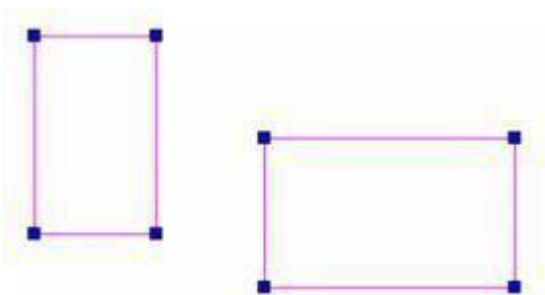
Modifying Double Line Intersections

Right-click on the intersection node you want to modify. For the options, see [Double Line Options](#)

Editing Multiple Nodes

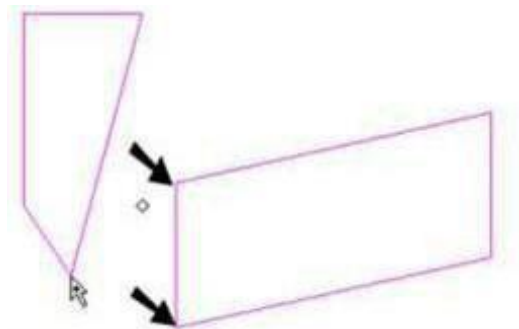
You can use a selection window to edit a group of nodes, even if the nodes belong to different objects.

Activate the **Edit Tool**. To select multiple objects use the Shift key. You can use Ctrl+A to select all objects for node editing.



2. Use a selection window to enclose all the nodes you want to edit.

The selected nodes turn from blue to magenta. When you move any of the selected nodes, all nodes move as a group.



To deselect nodes, draw an "empty" selection window (one that does not encompass any nodes).

Node Filter

In some circumstances, objects will have so many nodes that indicating and editing them all simultaneously slows down operations considerably.

To deal with this there is an option called On/Off "Node Filter" available from the Local menu and Inspector bar.

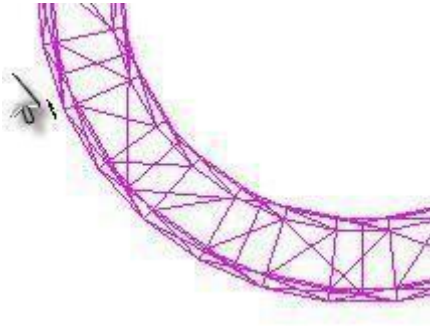
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Select the Edit tool.

Turn on the Node Filter.

Select the object.

Drag to select the desired nodes.



Only those nodes are selected.

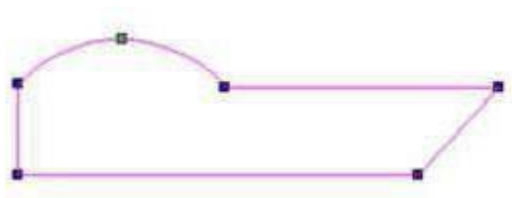
Hold down the SHIFT key to select multiple discrete areas,

Proceed with editing.

Editing Segments

For objects with multiple segments, you can move segments so that they remain parallel to their original location. Arc segments can also be edited.

1. Use the **Edit Tool** on a multi-segmented object like a polyline, rectangle, polygon, etc.

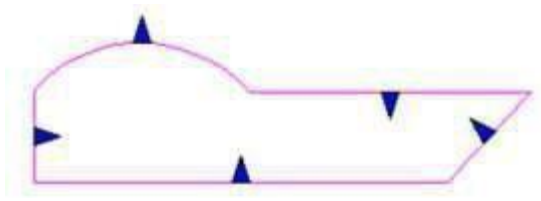


2. From the local menu or Inspector Bar, select **Edit Segments**.



The nodes are no longer marked, and segments are marked with triangles.

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Click and drag one of the segments. It remains parallel to its original location, and its length updates to meet adjacent segments.

4. If you edit an arc segment, its radius will change and its center will remain in place.

Note: This also works for walls, but each wall can be node edited separately when using **Edit Segments**. See [Editing Walls](#)

Editing Segments Push Mode



When Push Mode is off segments are simply moved as you drag an arrow and re-sized as needed.

However, when Push Mode is on rather than re-sizing, new geometry is created to fill the resultant gaps.

If the object is a standard 2D polyline etc. you can create self -crossing geometry.

However, you cannot create self crossing geometry if the object is a region. Attempting to do so will generate the following error:

Editing Splines and Bezier Curves

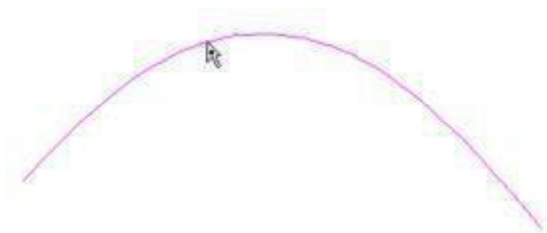
In addition to adding, deleting, or moving nodes, there are several **Edit Tool** features unique to splines and Bezier curves.

Changing the Shape of a Spline or Bezier Curve

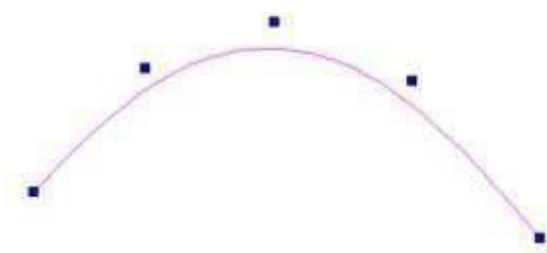
Use the **Edit Tool** on the curve you want to edit.



Click and drag any point along the curve.



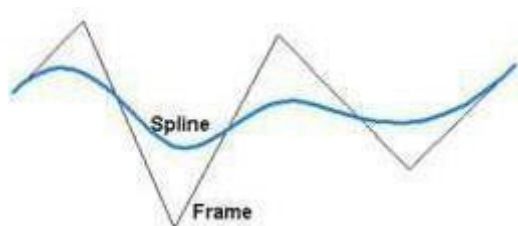
The control points update accordingly.



Changing Control Points and Fit Points

Splines are created by specifying either a series of fit points (points through which the spline passes) or control points (guide points). For either type of spline, you can view and move both types of points. It is easier to understand the types of points when the spline is displayed with its frame. Open the spline's **Properties** to the **Curves** page and check **Show Frame**.

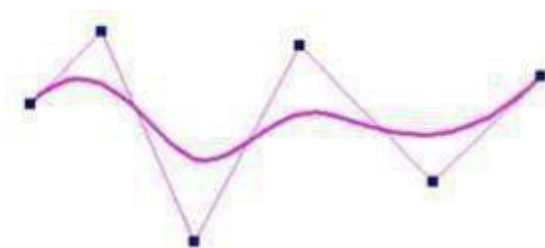
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Use the **Edit Tool** on the curve you want to edit. By default, **Edit Control Points** is active in the Inspector Bar and local menu.



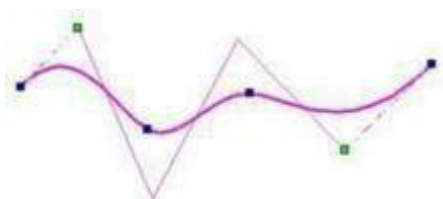
The control points are highlighted at the ends of frame segments. Each point can be clicked and dragged manually or assigned values in the Inspector Bar or Coordinate Fields.



Switch to Edit Fit Points.



Now the points through which the spline passes are highlighted. If the spline is open, you can use the green points to change the slope at the spline ends.



Note: Control points and fit points are not available for Bezier curves. For details on node-editing Bezier curves, see [Curve Properties](#)

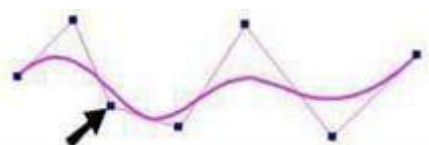
Adding Nodes and Knots

A knot is basically a control point. You can add knots to increase the editing detail of a spline. A node is a point through which the spline will pass (fit point).

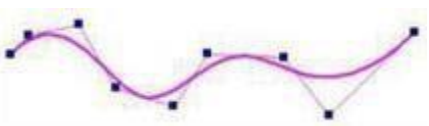
Use the **Edit Tool** on the spline you want to edit. In this example, the frame of the spline is displayed (check **Show Frame** on the **Curve** page of the spline's **Properties**).



It does not matter whether fit points or control points are highlighted. To add a control point (knot), right-click on the spline (not on the frame) where you want to add a knot and select **Add Knot**. The control point is added along the frame.



If you continue adding knots, you can "tighten" the frame to the spline, providing more control points for editing.

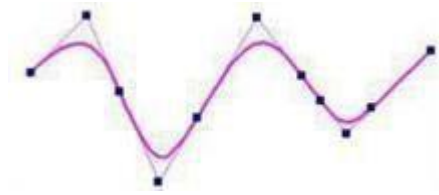


To add a fit point (node), right-click on the spline, near where it passes through the frame. Select **Add Node**. The fit point is added along the frame.



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If you continue adding nodes, you can "tighten" the spline to the frame, providing more fit points for editing.



Note: Nodes can be added to Bezier curves as well, but knots can be added only to splines.

Changing Node Curvature of Beziers

Bezier curve use weighted nodes to control the curvature at each node point.

You can control how each node controls the curvature by selecting a node, and right clicking to open the local menu. In the local menu, there are three options for controlling the curvature of a node.

Equal Curvature at point – This is the default. When selected, adjusting one of the green control handles causes the paired handle to change an equal distance, but the handles remain locked to a common control line. This option makes the curve smooth and equal on both sides of the node.

Nonequal Curvature at point – When this option is selected, adjusting one of the green control handles does not affect the distance of the paired handle, but the nodes remain locked to a common control line. This option makes the curve smooth but unequal on either side of the node.

Nonsmooth at point – When this option is selected, adjusting one of the green control handles does not affect the paired handle at all, the nodes have separate control lines. This option allows you to make a sharp point in the curve.

Note: These controls will have no effect if the "keep Curve Smooth" option is turned off in the curve properties. See [Curve Properties](#)

The effect of selecting a different curvature only appears after you begin moving an affected handle.

Highlight End Points

You can use the Selection Tool function to highlight the end points of the polyline, polyline 3D, double line/walls, spline/bezier.

If the “Highlight end points” option is ‘on’, then endpoints marks are red.

Selector Mode in Edit Tool

You can use the Selection tool functions to edit nodes by turning on the Selection Mode. Selection mode is available from the local menu or from the Inspector bar.

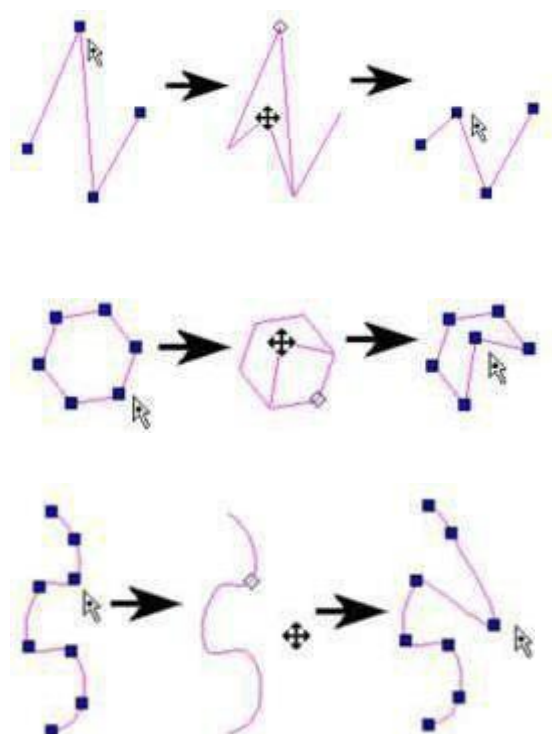


While in this mode you can select multiple nodes and move them, rotate them, and scale their collective size just as you would any other object using the selector.

Note: The selector type (2D or 3D) that will be used in Edit tool is dependant upon the type that is current for the selector.

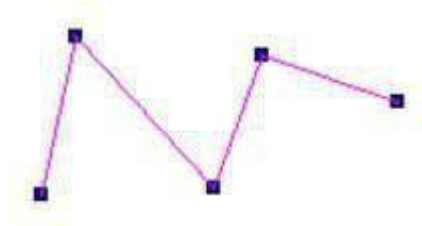
Moving a Node

The most basic method of node editing is to move nodes. Simply click on any node and drag it to its new location.



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If you want to move nodes orthogonally, you can press Shift while dragging. This example will use a polyline:

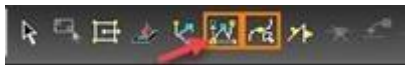


In the local menu or Inspector Bar, if Ortho Origin by Entity* is disabled, the nodes will be moved orthogonally according to the active UCS.



In this mode, you can move the node directly above, below, left, or right of its current position.

If **Ortho Origin by Entity** is enabled, the node will move so that the segment itself will become orthogonal.



In this mode, the segment before the node will become either vertical or horizontal.

For Bezier curves, you can choose to have additional control while node editing. By checking **Show additional control points** in the curve's **Properties**, you can add two more control nodes at every node.

You can also control the tangency at each node. See [Curve Properties](#)

Node Editing Perpendicular Vectors

It is possible to lock node editing to allow movement only parallel to the standard Perpendicular vectors (x, y, z) by using the local menu item "Move along coordinate vector"

This function is available only if one object is selected. If this mode is switched on and some nodes are selected, you can select one of three vectors (x, y, z). and drag it. Note: if the object is 2D then only 2 vectors will be available..

If one of the vectors is selected, then all selected nodes move along the selected vector.

Selector Mode

The Selector mode acts like Selection editing, but applies only to the selected components of the objects rather than the object as a whole.

You can use the Selector Mode within the Edit tool to move selected nodes, edges, and faces.

Workplane by 3D Object



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

When this option is selected the current workplane (UCS) will automatically switch to the workplane of the selected 3D object.

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This allows for easier control under some node editing situations.

6.5. Formatting Objects

This section covers tools for setting the appearance and format of drawing objects and other objects.

Aligning Objects

(Available in all TurboCAD Variants)

Default UI Menu: Modify/2D Align

Ribbon UI Menu:



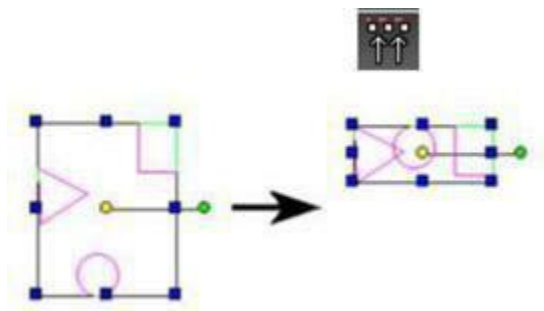
Aligns all currently selected objects, relative to the bounding box of the selection. You can display the **Align** toolbar by right-clicking in any toolbar area and selecting **Align**.



Select the objects you want to align.

From the **Align** menu (or the **Align** toolbar), select the type of alignment.

For example, **Top** moves the objects to the top of the bounding box.



Align Along Line

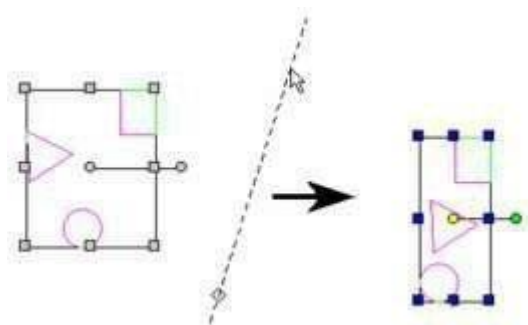
Align objects along a line.

Select the objects you want to align.

Select **Align line**, or click the icon in the **Align** toolbar.



Define the location of the alignment line by selecting one point, then select a second point to set the angle. You can also enter the angle in the Inspector Bar.



Distributing Objects

(Available in all TurboCAD Variants)

Default UI Menu: Modify/2D Distribute

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Ribbon UI Menu:



Distributes all currently selected objects, relative to a specified location on the objects (top, left, center, etc.). There must be at least three objects selected for this function to be available.

Objects are distributed between the two most extreme objects, i.e. the farthest left and the farthest right, or highest and lowest. Object order is maintained (top to bottom or left to right).

You can display the **Distribute** toolbar by right-clicking in any toolbar area and selecting **Distribute**.



Select the objects you want to distribute.

From the **Distribute** menu (or the **Distribute** toolbar), select the type of distribution.

For example, **Top** arranges the objects so that the tops of their bounding boxes are evenly spaced vertically.

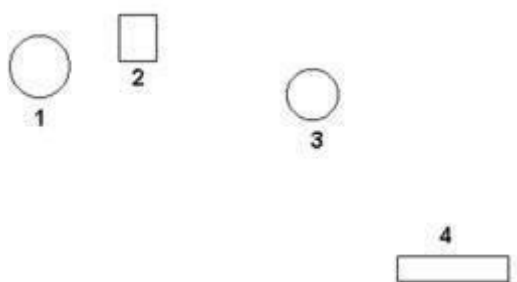
Bottom arranges the objects so that the bottoms of their bounding boxes are evenly

spaced vertically.

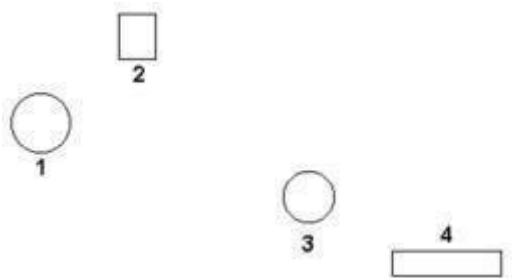


For vertical distribution, **Middle** means equal spacing between reference points, and **Space** means equal spacing between object bounding boxes. For horizontal distribution, **Center** means equal spacing between reference points, and **Distance** means equal spacing between object bounding boxes.

In this example, Object 2 is the highest, followed by 1, 3, and 4.



After vertical distribution, the highest and lowest objects (Objects 2 and 4) remain in place. Objects 1 and 3 are moved so that top-to-bottom order remains the same: 2, 1, 3, 4.



Exploding Objects

(Available in all TurboCAD Variants)

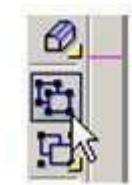
Hotkey: Alt+Shift+E

Default UI Menu: Modify/Explode

Ribbon UI Menu:



Breaks an object, group, or block into its constituent parts. Each explosion is one level deep, to preserve the hierarchical structure of objects. Objects must be selected first in order to be exploded.



Note: The opposite of **Explode** is to join objects into a group (**Format / Create Group**). See [Groups](#). You can also use **Explode** to split solids that are difficult to split using 3D editing tools. Different types of objects are exploded differently, and use different explosion levels. Blocks and groups are first broken into their individual objects, then each object can then in turn be exploded. Polylines are broken into individual segments. Dimensions are exploded into lines, arrows, and text, then each of these can also be exploded. Double-line objects are exploded into single lines. You can explode a solid object twice to turn it into an exploded (node-editable) surface object.

When using **Explode**, it can be helpful to open the Selection Info Palette (see [Selection Info Palette](#)), so that you can see the start condition and end results.

Paper Space Viewport Explode

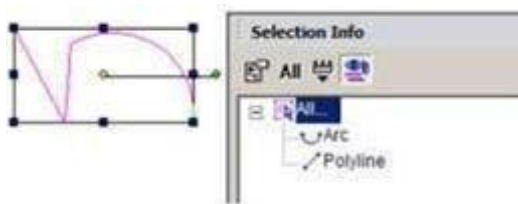
Possibility to explode paper space viewport in Hidden line render mode when switch on Foreground wireframe option - show non render able objects. Now you can explode Hidden line VP with wireframe draw non render able objects (2d objects: dimension, text, polyline,...)

Exploding Example - Polyline

Create a multi-segmented polyline (with **Auto Add Constraints** off) and select it. In the Selection Info Palette, the object type is indicated as "Polyline."

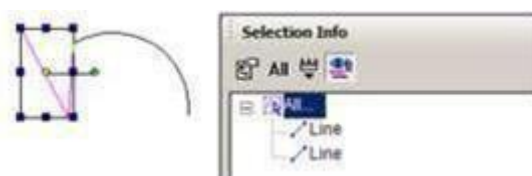


Select **Explode**, or click the **Explode** icon. The polyline is exploded into one arc and one small polyline.



Select the small polyline.

Explode it into its constituent parts - two lines.



Exploding Example - Text

Use the **Text** tool (see [Inserting Text](#)) to create a text object, and select it.



Select **Explode**, or click the **Explode** icon. The text is exploded into a Group of Graphics.



Explode again, and now the group consists of one polyline representing each character.



4. Select any character; it is identified as a polyline.



Format Painter

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum, Professional and Deluxe)

Hotkey: Ctrl+Shift+P

Default UI Menu: Format/Format Painter

Ribbon UI Menu:



Takes properties from one object and applies ("paints") them onto other objects. You can paint all properties or only selected ones.

Select the source object - the object with the properties you want to apply to other objects.

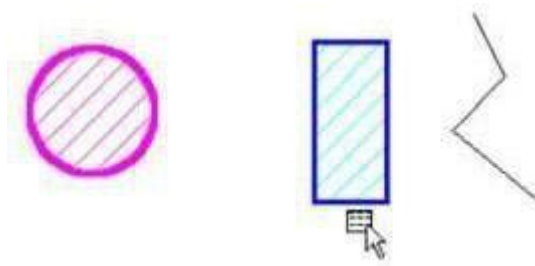


The Format Painter Palette appears in the palette area, listing all the properties that can be painted.

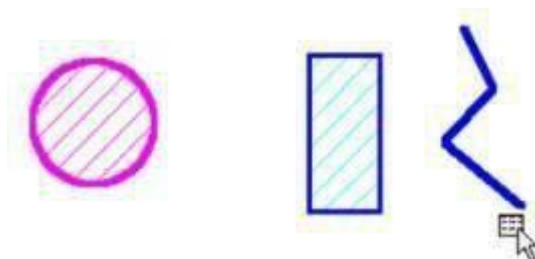
The list of properties depends on the selected object. For instance, a text object will have a category for **Text** properties; a sphere will have a category for **Sphere** properties.

By default, all properties are selected. If you want to remove any properties, uncheck the relevant box.

Select the object to which you want to paint the properties.



Select additional objects, if needed. Only the relevant properties will be applied. For example, the circle's **Brush** properties are not applied to the polyline (open objects cannot be filled).



4. Select **Cancel** from the local menu or Inspector Bar.

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Local menu options:

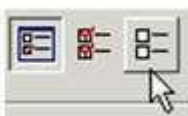
Make property values preset: If this is selected the Make Property Value Presets dialog will appear. You can specify a name and type of preset that you want to create. The parameters of the preset will be generated from the object you selected, and the properties you specified in the Format Painter palette. **Use Palette:** Displays or hides the Format Painter Palette.



Mark All: Selects all properties for painting. If the Format Painter Palette is not displayed, the properties will appear in a separate window.



Unmark All: Clears all properties for painting. If the Format Painter Palette is not displayed, the properties will appear in a separate window.



Regions

(Available in Platinum)

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Default UI Menu: Modify/Create Region

Ribbon UI Menu:



Combines 2D closed objects to form one object, called a region. Any overlap between selected objects is removed. The results are identical to using **2D Add**, except that this tool requires that the objects be selected first. When using **2D Add**, you select the objects during the operation.

Stacking Objects

(Available in all TurboCAD Variants)

Default UI Menu: Tools/Draw Order

Ribbon UI Menu:

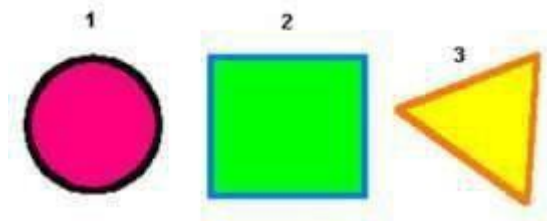


Objects are "stacked" when created - each object is created "on top" of the previous object. This sometimes has no visual effect, but stacking order can matter in the case of images and filled or hatched objects.

Note: Stacking is not related to layers, and changing an object's layer does not effect its position in the object stack. See [SnapModes](#)

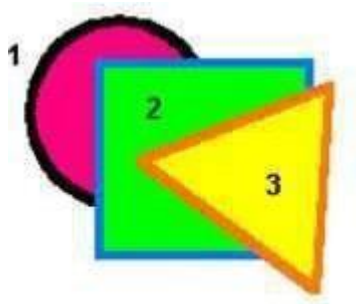
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This example contains three filled objects that were created in the indicated order:

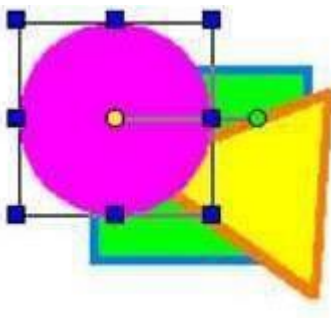


Tip: You can press F6 to select the first object you created. Keep pressing F6 to scroll through the objects in their creation order. Press F7 to select the last object created, and repeat F7 to scroll backwards.

If you move the objects so that they overlap, you can see that the first object is at the bottom of the stack; the last object is at the top.

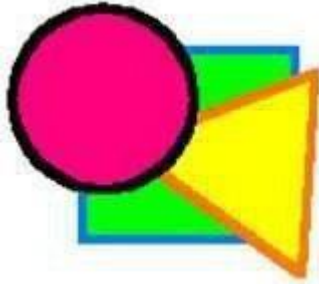


Select the circle and select **Bring to Front**.

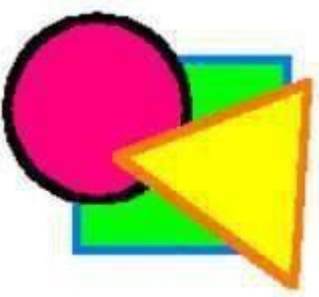


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The circle is now at the top. This, in effect, changes the creation order - the circle is now considered the most recent object. (You can verify this by pressing F7.)

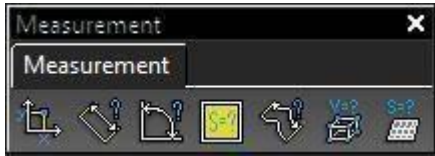


Select the circle again and select **Back One**. Rather than sending the circle to the bottom of the stack, it moves just one level down.



If you change the stacking order of multiple selected objects, the selection set will move as a whole. The relative stacking order of objects within the selection set is maintained.

6.6. Measuring



Measurement tools are constrained to the current workplane. If you snap to a point that does not lie on the workplane, that point will be projected onto the workplane. Make sure the workplane is set properly for the objects you wish to measure, or the measured values will not be accurate. See [Workplanes](#)

Results of measuring are displayed in the Measurement Info Palette, which is opened automatically when using a measuring tool.

Tip: To copy the contents of the Measurement Info Palette to the Windows clipboard, use the Copy command from the local menu. You can then paste the information into any word processor, text editor or other program that will accept text.

The Measurement Info Palette can be set to keep a history of all the measurements you take. Use the Keep History button at the top of the palette to retain measurement history. Use the Clear Content button to delete the content of the palette.

Previously, the measure coordinates of the point tool could only measure the X and Y values as it functions like a 2-D tool and even though 3-D point was selected it showed a zero value. We need a measure 3-D point tool that is independent of the work plane much like the 3-D line tool and be drawn regardless of the work planes on. The only workaround for this was to draw 3-D line to check its starting and ending coordinates. Now measurement tool can show x, y, z coordinate. Output coordinates is in world CS.

The default coordinate units are those set in the **Space Units** page of the **Drawing Setup** (see [Space Units](#)) You can adjust the units by selecting an option from the Inspector Bar or local menu. English units are available in fraction and decimal formats, and metric units can be displayed as well.



Demo Video

https://youtu.be/8K1WCqJ7H_g

2D Geometric Parameters

(Available in Platinum)

Default UI Menu: AddOns/Special Tools/Analyze/2D Geometric Parameters

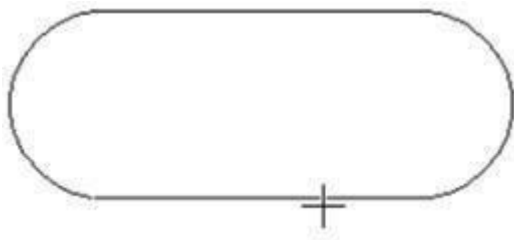
Ribbon UI Menu:

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Note: You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

Calculates engineering properties (area, center of gravity, moments of inertia, etc.) for a 2D closed object.

1. Select the closed area.



2. The area is highlighted, and its axes are indicated in red.

The parameters of the closed area appear in the **Geometric Parameters** window.

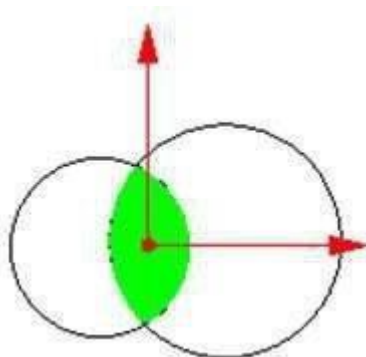
Click OK to close the window, and select **Finish** from the local menu or Inspector Bar to exit the tool.

Local menu options

By Closed Graphic (default): Selects a single closed object.



By Closed Area: Click inside a closed area to obtain the properties for only that area.



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Relative Axis: Enables you to define another axis for the calculation of the properties.



Finish Geometric Parameters: Completes the 2D geometric parameters operation.



Section Properties



(Available in Platinum)

Default UI Menu: Tools/Inquiry/Section Properties

Ribbon UI Menu:



Feature to display Ssection Properties of any closed 2D object.

To display Section Properties of any 2D closed object, “Section Properties” needs to be selected first. Clicking on a 2D object with Section Properties option enabled displays the Section Properties dialog box.

To display Section Properties:

1. Draw a 2D Closed Graphic

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Select Section Properties

Select the 2D Closed Graphic

Graphic is highlighted and Section Properties Dialog Box is shown

Add Main Axis: Adds main axis to graphic.

Units of Measurement: Selection of Measurement units (in, ft, yd, mm, cm, m).

Coordinates of a Point

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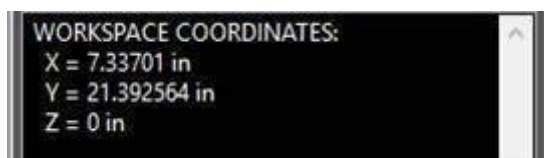
(Available in all TurboCAD Variants)

Default UI Menu: Tools/Inquiry/Coordinates of the Point

Ribbon UI Menu:



Select the point, and its coordinates are displayed in the Measurement Info Palette.



Curve Length



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Inquiry/Curve Length

Ribbon UI Menu:



This tool works exactly the same as the Measure Distance. Click on a 2D or 3D curve. Its length is displayed in the Measurement Info Palette.

Select a second point on the curve to measure a partial length of the curve.

Measuring Angles



(Available in all TurboCAD Variants)

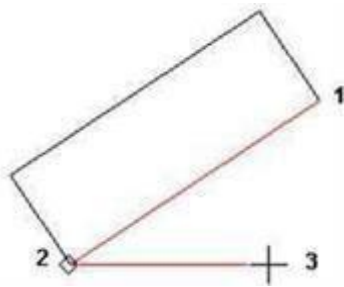
Default UI Menu: Tools/Inquiry/Angle

Ribbon UI Menu:

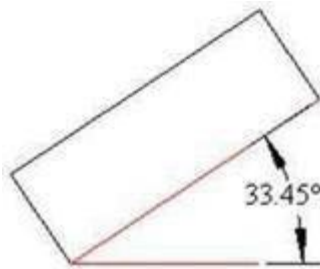


Measures the angle between three points, or between two lines.

1. Select three points that define the angle you want to measure.



The angle is measured and displayed. It is calculated according to the order of selected points.



The angle is also displayed in the Measurement Info Palette.

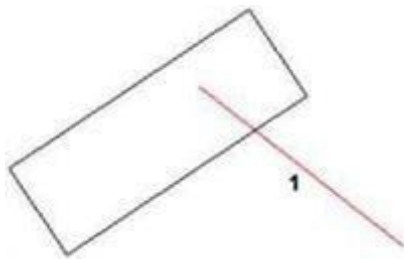


Local menu option:

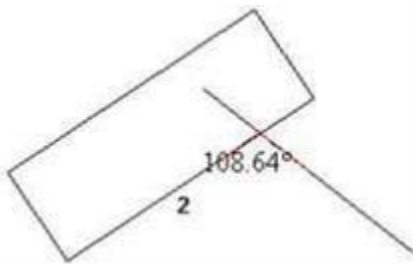
By Entity: Measures the angle between two lines. The lines must intersect (this tool does not infer where extensions of lines will meet)



Select the first line. The point you select on the line defines the angle that will be measured.



2. Select the second line, and the angle is displayed.



In this example, if you selected the lines in a different location, you will measure a different angle.

Measuring Area



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(Available in all TurboCAD Variants)

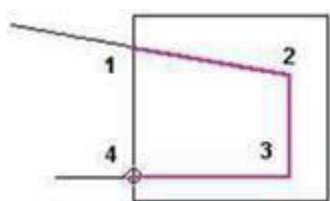
Default UI Menu: Tools/Inquiry/Area

Ribbon UI Menu:

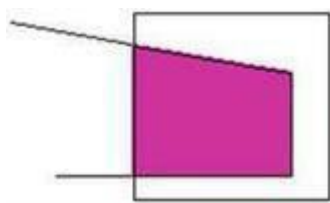


Measures the area of a closed defined boundary.

Select points to define segments of the boundary. The points do not have to lie on existing objects; they can be anywhere. A thick line indicates the progress of the boundary.



Select the first point again to close the boundary, or select **Close** from the local menu or Inspector Bar. The defined area is filled.

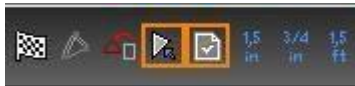


3. The calculated area is displayed in the Measurement Info Palette.



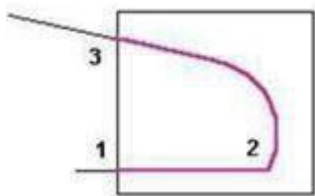
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Local menu option: By Entity: Enables you to select the entire perimeter of an object.

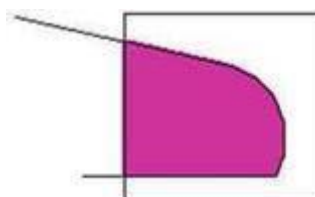


You can also use this option to define a chain along a portion of an object. This is useful when you need to measure the area of curved objects.

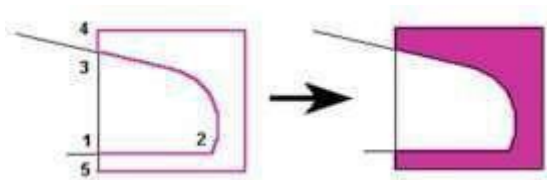
1. Select a closed object, or select points along a portion of an object.



Select the first point again to close the boundary, or select **Close** from the local menu or Inspector Bar. The defined area is filled.



3. You can also use **By Entity** to define a boundary along more than one object.



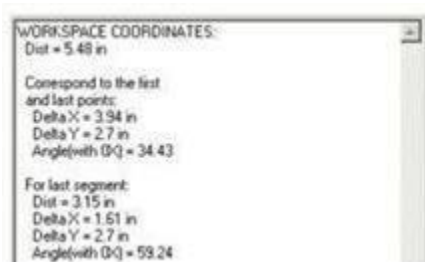
Measuring Distance



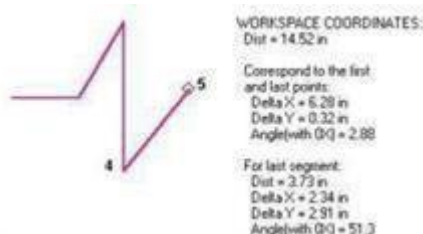
(Available in all TurboCAD Variants)

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The first listed measurement is the total length of segments. The second group of measurements relate to the first and last points - in this case, Point 1 and Point 3. The last group applies to the last segment only (Point 2 to Point 3).



3. Continue selecting points, in order, to determine the perimeter of the object.



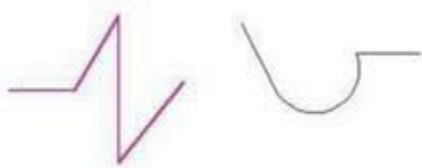
Note: If you want to obtain the measurements of curved objects, you need to use the local menu options.

Local menu options:

By Entity: Enables you to select two or more objects to determine their total length.

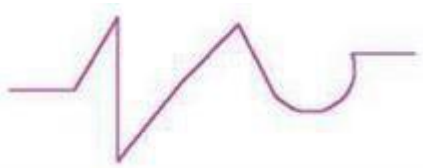


Select the first object.

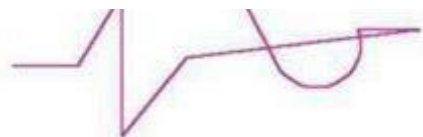


Select the second object. The total length is calculated based on the shortest line connecting the two endpoints.

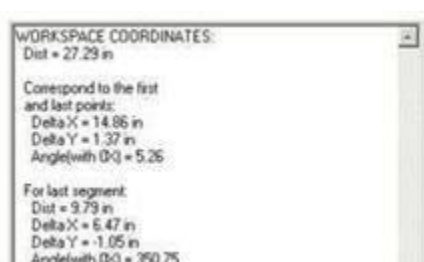
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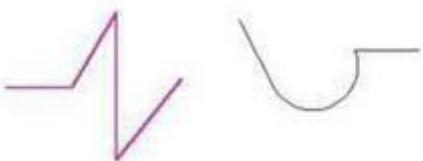
Note: When selecting objects, be sure to click close to the desired start point. If you select close to the wrong end, you may obtain an incorrect measurement.



Select additional objects if needed. The total distance, deltas between the first and last points, and data for the last segment are listed in the Measurement Info Palette.

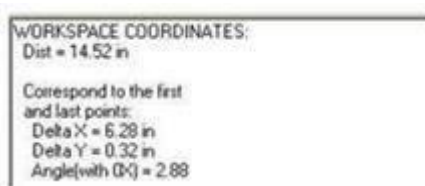


Single Entity: Measures the distance of a single object.
7. Select the desired object.

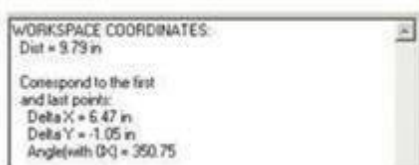
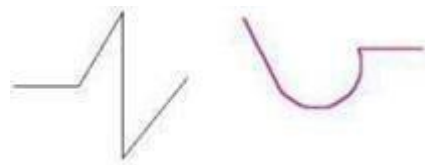


The total distance and data deltas between the first and last points are listed in the Measurement Info Palette.

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If you select another object, the data from the first object disappears, and is replaced by that for the new object.



Measuring Surface Area



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Inquiry/Surface Area

Ribbon UI Menu:



Select a surface. Its surface area is displayed in the Measurement Info Palette.

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Note: To select multiple surfaces hold down the Shift key.

Measuring Volume



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Inquiry/Volume

Ribbon UI Menu:



Select the object. Its volume is displayed in the Measurement Info Palette.

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Note: To select multiple surfaces hold down the Shift key.

Mass Properties



(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Tools/Inquiry/Mass Properties

Ribbon UI Menu:



You can access the Mass Properties of ACIS® solid objects in a drawing. Additionally, when selected, the Mass Properties are viewable from the Custom properties of the select 3D object, and these properties, in turn, can be automatically placed in the Selection Info palette

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To access Mass Properties:

1. Draw or select one or multiple 3D Solid Objects.

Select Mass Properties from the Tools/Measurement menu.

Select one or multiple 3D Solid objects.

Press Finish

The following Dialog Box opens:

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When you press “Apply”, the selected entities will be assigned as Custom Properties accessible in the Local Menu:

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With this local menu displayed, click 'Regenerate' in order to see these properties displayed in the Selection Info Palette:

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Note: In order to add more Materials and Densities, the user can go to:

“C:\Program Files\IMSDesign\TCW2023\Materials” and edit “tc_matsDensity.ini”

6.7. Modifying Geometry

(Available in all TurboCAD Variants)

The **Modify** tools are generally used to change the shapes of existing objects. These tools do not change physical properties such as layers or line widths; rather they modify the actual geometry.

You can display the **Modify** toolbar by right-clicking in any toolbar area and selecting **Modify**.



Some **Modify** tools require input of two or more objects. If you are working in 3D, all 2D objects selected for modification must lie on the same workplane (see [Workplanes](#)). To change an object's workplane, see [Place on WorkPlane](#)

Arc Complement



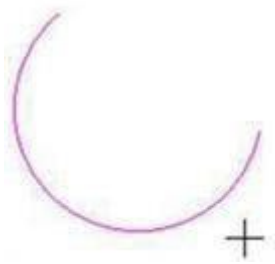
(Available in all TurboCAD Variants)

Default UI Menu: Modify/Arc Complement

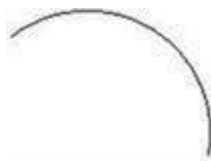
Ribbon UI Menu:

Reverses the start and end angles of an arc. In other words, the arc will become the portion of the original circle that was cut.

1. Select the arc you want to modify.



2. Click anywhere to create the arc complement.



Tip: When using this tool on arcs that are the result of exploding a polyline, or created as the result of using the **Arc Divide** option of the **Edit Tool**, this tool may select more than one arc at a time. If this happens, move the arcs from each other.

Chain Polyline



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Chain Polyline

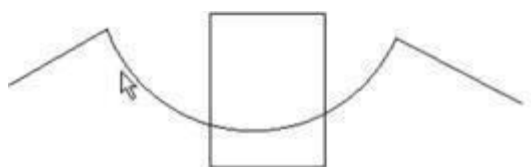
Ribbon UI Menu:

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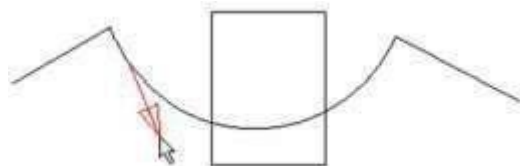
Connects intersecting objects or portions of objects into a single polyline. A chain of connected objects is created, starting from a selected object and proceeding in a specified direction.

Note: The difference between this tool and the **Auto Joining** option of **Join Polyline** (see [Join Polyline](#)) is that **Chain Polyline** enables you to create a partial chain, and to work with overlapping objects.

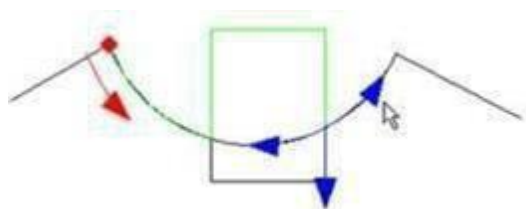
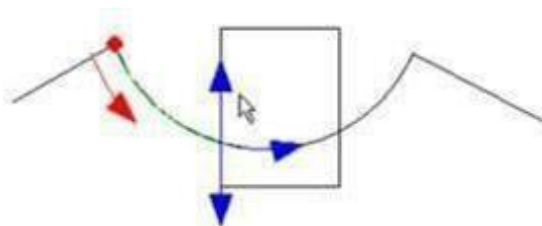
1. Select the object from which the chain will start.



Set the direction in which the chain will proceed.

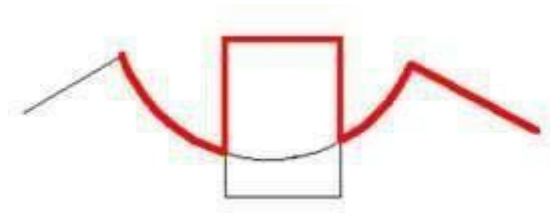


If overlaps are encountered, you must specify the path to continue the chain.



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4. The chain proceeds until it reaches the last object, and the polyline is created.



If you do not want the chain to proceed all the way to the last object, you can select **Finish** from the local menu or Inspector Bar at any time.

Note: The original objects remain in the drawing.

Chamfer

(Available in all TurboCAD Variants)

Default UI Menu: Modify/Chamfer

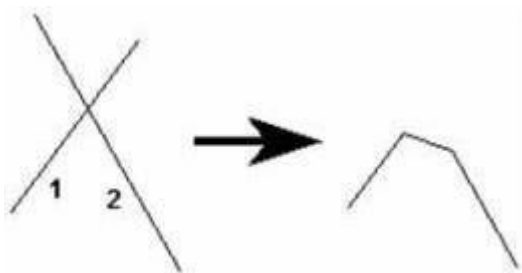
Ribbon UI Menu:

Connects two lines (single or double lines) with a beveled corner. The selected objects do not have to intersect, and they can overlap.



Note: The resulting objects are separate line segments, unless the **Polyline** option is used, in which case the result is a polyline.

If the objects overlap, they will be trimmed to create the chamfer. Be sure to select the side of the object you want to remain.



There are three methods for creating chamfers: **Distance / Distance**, **Distance / Angle**, and **Length / Angle**. For all methods, the same local menu options are available.

Local menu options:

Polyline: Chamfers all corners of a polygon or polyline in one step.

Enter the chamfer distances, angles, or length in the Inspector Bar.

Select the multi-segmented object you want to chamfer.

Select **Polyline** from the local menu or Inspector Bar.



The chamfer is applied to all corners. The resultant object is one polyline.



Warning: Be careful when using the **Polyline** option with different chamfer distances. The results will not be symmetric.



The other local menu options apply to double lines. See [Double Line Options](#)

Chamfer Distance - Distance

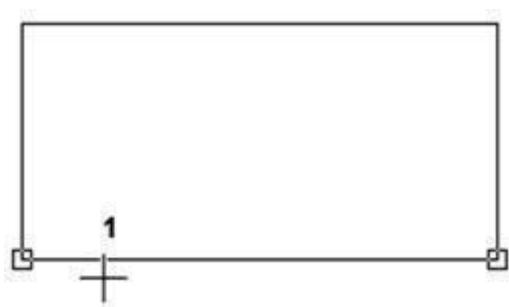


The chamfer is created by specifying its length along each line.

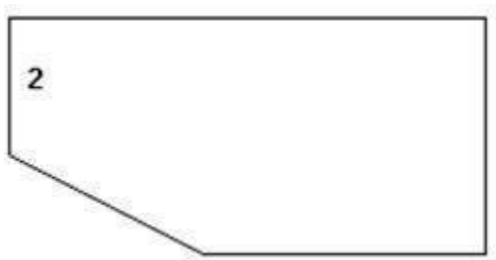
Enter both chamfer distances in the Inspector Bar. These are the distances along the lines that are to be chamfered. Distance A will be applied to the first line you

select. 

Select the first line to chamfer.



3. Select the adjacent line. The corner is chamfered.



Chamfer Distance - Angle

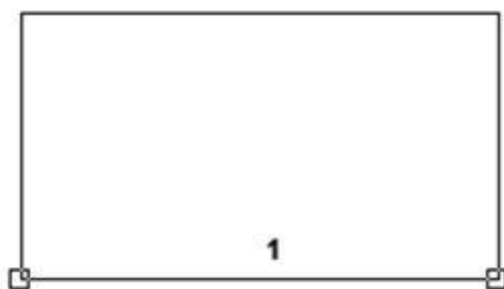


The chamfer is created by specifying its distance along one line and its angle from the second line.

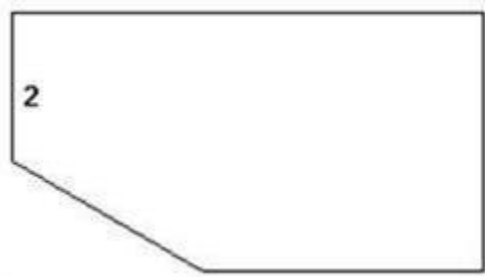
Enter the distance and angle in the Inspector Bar. The distance is measured

along the first line you select.

Select the first line to chamfer.



3. Select the adjacent line. The corner is chamfered.



Chamfer Length - Angle

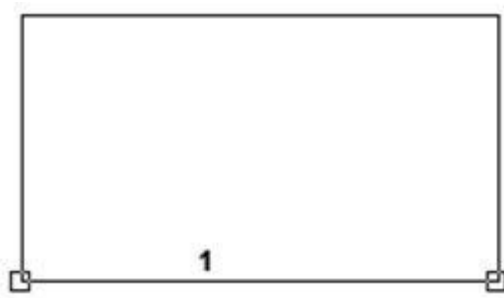


The chamfer is created by specifying the total chamfer length (as opposed to the distance along the lines) and chamfer angle.

Enter the length and angle in the Inspector Bar. The length is the total length of

the diagonal chamferline.

Select the first line to chamfer.



3. Select the adjacent line. The corner is chamfered.



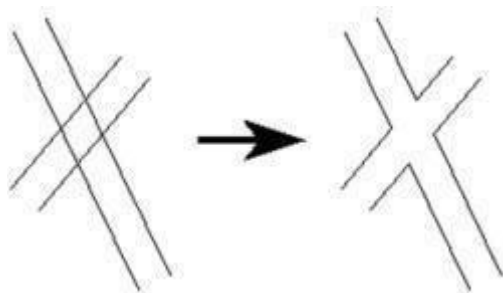
Double Line Options

(Available in all TurboCAD Variants)

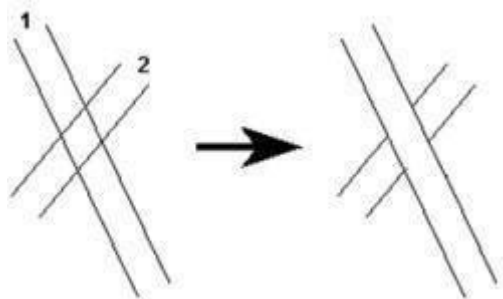
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You can specify how to create intersections of double lines, and whether cut lines and gaps will occur at the intersections.

For example: if you form a clean T-intersection of two double lines at an angle other than 90 degrees, the end of one of the first line will be cut at an angle, and the second double line will have a gap to accommodate the first double line. **Cleanup:** Forms a clean intersection.



If you deselect **Cleanup**, the first line selected will remain unbroken.



DropCutLines: Removes the cut lines in the first line, to even out the end of the line.



Drop Gaps: Fills in the gap in the second line.



Fillet



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Fillet 2D

Ribbon UI Menu:

Connects two lines (either single or double lines) or arcs (except elliptical arcs) with a smoothly fitted arc. The filleted objects do not have to intersect, and they can overlap. The result is a rounded corner.

Note: You can also fillet corners with the **Edit Tool** (see [Editing Linear Objects](#)). To apply fillets to a 3D polyline, see [Fillet 3D](#)

Enter the fillet radius in the Inspector Bar.

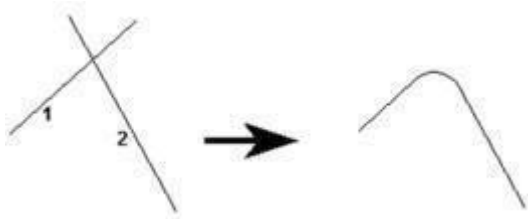
Select the first line or arc to fillet.



3. Select the adjacent line or arc. The corner is rounded.



If the objects overlap, they will be trimmed to create the fillet. Be sure to select the side of the object you want to remain.



Local menu options:

Polyline: Fillets all corners of a polygon or polyline in one step.

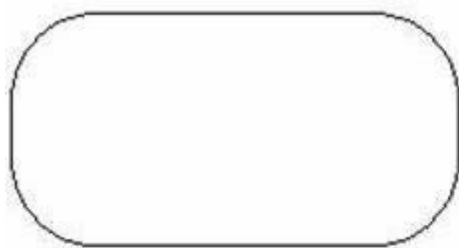
Enter the fillet radius in the Inspector Bar.

Select the multi-segmented object you want to fillet.

Select **Polyline** from the local menu or Inspector Bar.



The fillet is applied to all corners.

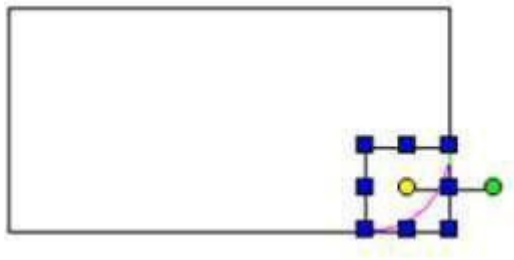


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To Lines and Arcs: The resulting object is broken into lines and arcs. (The default, **To Polyline**, leaves the filleted object as a single object, even if the objects were originally separate.)



In **Select** mode, you can verify that the fillets are separate objects.



The other local menu options apply to double lines. See [Double Line Options](#)

Note: You can use Fillet on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs.

Note: You can use Fillet to change an arc on a polyline, but only on arcs that are between two straight segments. Simply select the two straight segments and the arc will be adjusted to the radius of the fillet.

Fillet 3D

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(Available in Platinum and Professional)

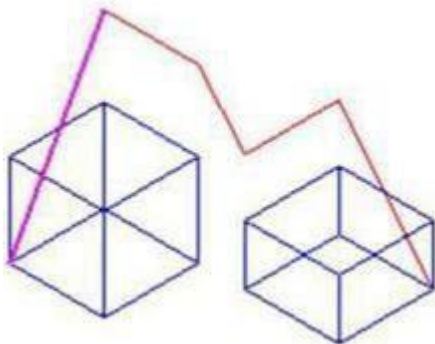
Default UI Menu: Modify/Modify 3D Objects/Fillet 3D

Ribbon UI Menu:

Creates a fillet on a 3D polyline

Enter the fillet radius in the Inspector Bar.

Select the first line or arc to fillet.



3. Select the adjacent line. The corner is rounded.

Note: You can re-fillet to the same to segments with a different radius and the fillet will be adjusted.

Local menu options:

Fillet All: Fillets all corners of a polygon or polyline in one step.

Enter the fillet radius in the Inspector Bar.

Select **Fillet All** from the local menu or Inspector Bar.



Select the multi-segmented object you want to fillet. The fillet is applied to all corners.

Intersect 2 Double Lines



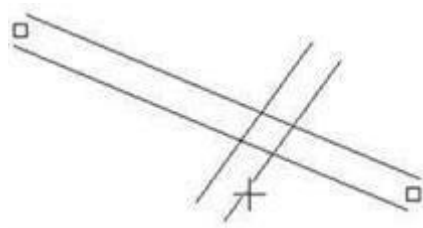
(Available in all TurboCAD Variants)

Default UI Menu: Modify/Intersect Double Lines/Intersect 2 Double Lines

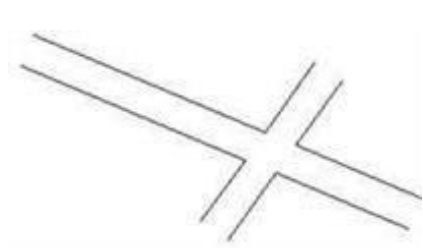
Ribbon UI Menu:

Forms an X intersection of two double lines, thereby cleaning up the intersection.

1. Select both lines whose intersection you want to create.



2. The X intersection is created.



Local menu options:

See [Double Line Options](#)

Join Polyline



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Join Polyline

Ribbon UI Menu:

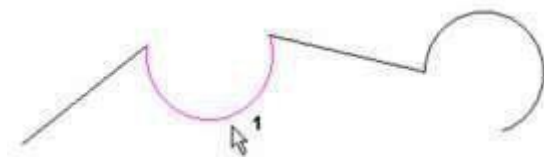
Connects two or more separate objects, connected at their endpoints, into a single-object polyline. Objects that overlap cannot be joined.

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Note: If you want to create a polyline from a **partial** chain of segments, or if you have overlapping objects, see [Chain Polyline](#)

Select the tool.

Select the first object you wish to include in the polyline. Selection order is not important.

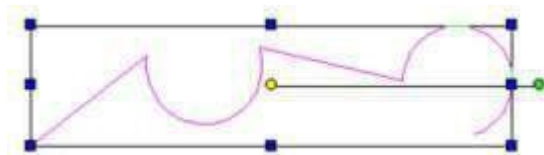


One by one, select the remaining objects to include.



Select **Finish** from the local menu or Inspector Bar.

The objects now comprise one object, as you can see in **Select** mode.



As of TurboCAD 2023 when you are joining a set of all Beziers the resulting object will be a Bezier.

Local menu options:



Delete Original Objects: Removes the original segments, leaving only the polyline.

Auto joining: Automatically selects all objects connected to the selected object. This

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option must be active before selecting any segments. You can select any object in the chain, not only the first or last object. If more than one segment branches from an endpoint, the chain will stop.

3D Polyline: Enables you to include objects located on different workplanes.

Line Length



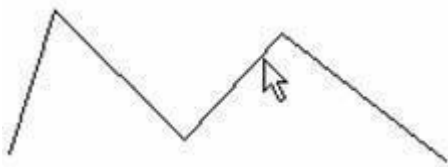
(Available in all TurboCAD Variants)

Default UI Menu: Modify/Line Length

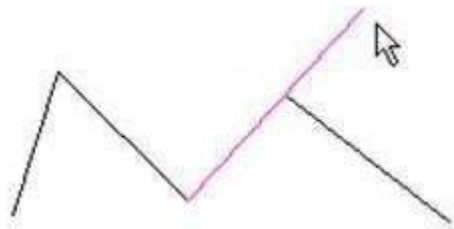
Ribbon UI Menu:

Shrinks or extends a line or line segment from either endpoint. Double lines and multi lines can also be used. If you use this tool on a polyline or polygon, any adjacent segment will move to accommodate the new endpoint.

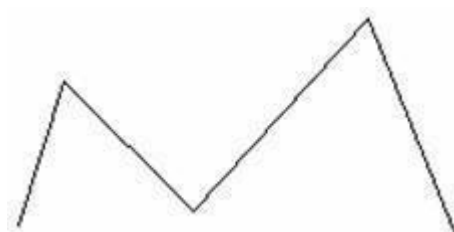
1. Select the line or line segment close to the endpoint you want to move.



Move the cursor to set the new length, or enter the length or delta in the Inspector Bar. Negative values can be used, thereby extending the line in the opposite direction.



If another segment meets the moved endpoint, it is changed as well.



Note: You can use Line Length on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs. 3D Polylines, and splines can also be used, but the resulting geometry will be derived relative to the objects workplane.

Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with walls.

Meet 2 Lines



(Available in all TurboCAD Variants)

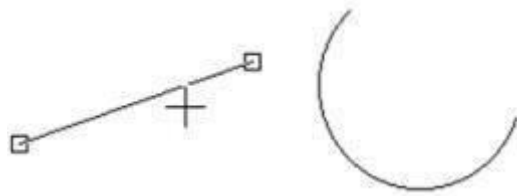
Default UI Menu: Modify/Meet 2 Lines

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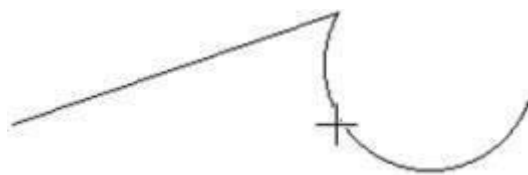
Ribbon UI Menu:

Extends two lines, double lines, walls or arcs so that their endpoints meet.

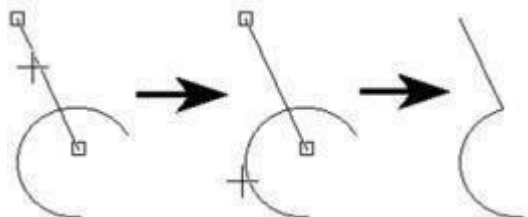
1. Select the first object.



2. Select the second object. The objects are trimmed or extended to meet one another.



If an object is to be trimmed, select the portion of the object you want to remain.



Local menu options:

The local menu options apply to double lines. See [Double Line Options](#)

Note: You can use Meet 2 Lines on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs. 3D Polylines, and splines can also be used, but the terminal resulting geometry will be placed on the concurrent workplane of the two objects used.

Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with walls.

Shrink-Extend Line



(Available in all TurboCAD Variants)

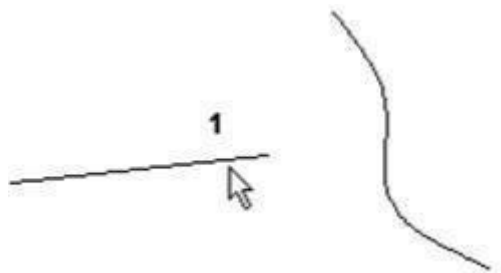
Default UI Menu: Modify/Shrink-Extend Line

Ribbon UI Menu:

Trims or lengthens a line so that it meets another object. Double lines and multi lines can also be used.

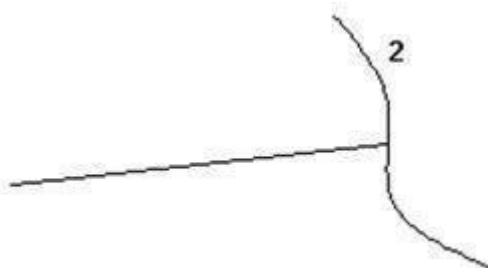
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Select the line you want to shrink or extend. Click near the endpoint you want to



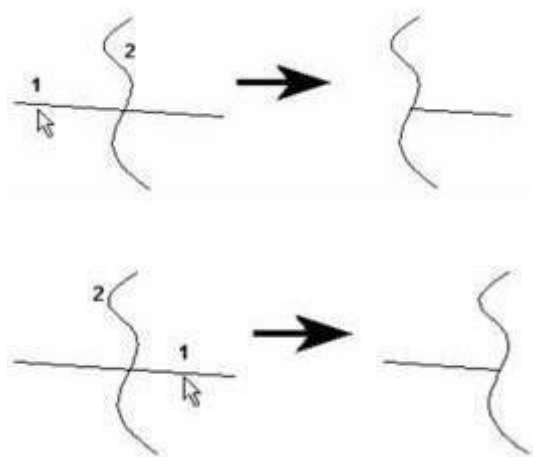
move.

Select the object that the line is to meet. The endpoint of the line meets the



selected object.

When shrinking a line, be sure to click near the endpoint that will move, otherwise the wrong part of the line will be deleted.



Local menu option:

Cleanup: Relevant for double lines, makes the cutlines invisible, forming a clean intersection. See [Double Line Options](#)

Note: You can use Shrink / Extend Line on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs.

Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with

walls.

Multi Shrink-Extend Line



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Multi Shrink-Extend Line

Ribbon UI Menu:

Similar to **Shrink / Extend Line**, trims or lengthens a group of line so that they meets another object. Double lines and multi lines can also be used.

Select two points to draw a temporary "fence" that passes over all of the lines you want to shrink or extend. Be sure the fence intersects each line closest to the endpoints you want to move.

The endpoints that will move are indicated by small squares. Select the object that the lines are to meet.

3. The endpoints of the lines meet the selected object.

Note: You can use Multi Shrink / Extend Line on splines and Beziers, however the extension will be straight element of the curve, not curved continuations as they are with arcs.

Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with walls.

Object Trim



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(Available in all TurboCAD Variants)

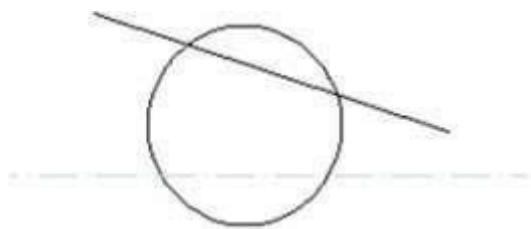
Default UI Menu: Modify/Object Trim

Ribbon UI Menu:

Uses one or more cutting edges to trim objects.

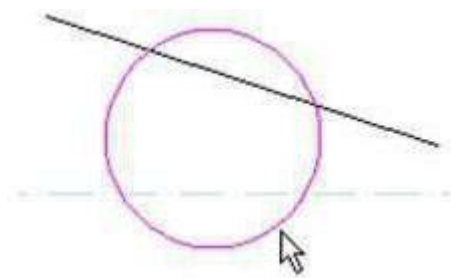
Note: Complex objects such as dimensions and text must be exploded before they can be trimmed. Text must be exploded twice - first into groups of polylines, then into polylines.

The example for **Object Trim** contains three objects: a circle, line, and construction line.

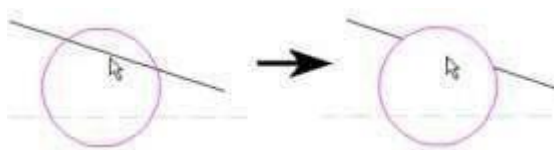


Using One Cutting Edge

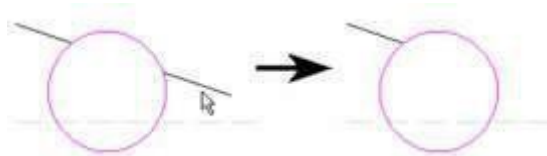
1. Select the object to be used as the cutting edge.



Select the object to trim, clicking on the portion you want deleted.



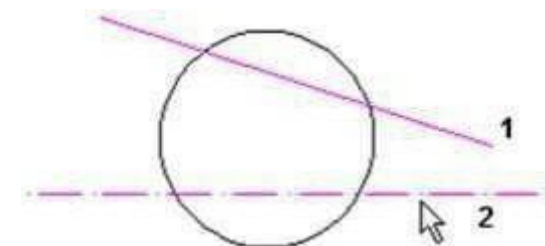
Continue selecting objects to trim, if needed.



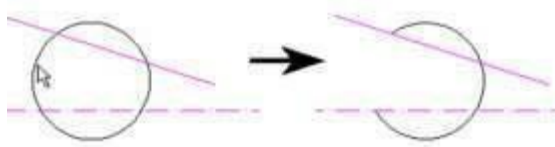
Press Esc if you want to select another cutting edge. Select **Cancel** from the local menu or Inspector Bar to exit.

Using Multiple Cutting Edges

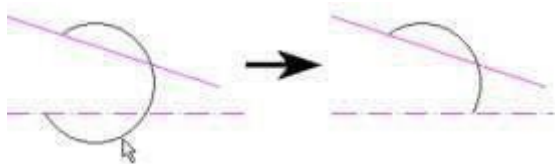
1. Select the objects to be used as the cutting edge. Use Shift to select multiple objects.



Select the object to trim, clicking on the portion you want deleted. One or both cutting edges will be used, depending on what is selected.



Continue selecting objects to trim, if needed.



Press Esc if you want to select another cutting edge. Select **Cancel** from the local menu or Inspector Bar to exit.

Local menu option:

Operation by Line: Enables you to define the cutting edge by selecting two points.

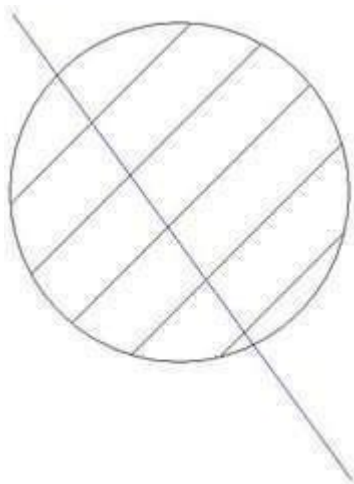
Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with walls.

Trimming a Closed Object



You can trim a closed or hatched object and retain closure.

1. Select the Object Trim tool.



From the local menu select the **Extract closed contour from closed entity** option.

3. Click on the trimming object/s. Use Shift to select multiple trimming objects.

4. Click on the object to be trimmed.
5. Click on the part of the object you wish to trim.

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Trimming a closed object will convert the result into a polyline so you will get the following message:

7. Click Yes.

Trim



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Trim

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Ribbon UI Menu:

The Trim tool uses 2D objects to cut off parts of other 2D objects.

Select the objects to be used as the cutting edges by clicking on each one. If you accidentally select an edge you can de-select it by holding down the Shift key and clicking in it.

2. Once you have selected your cutting edges, right click and select Finish Selection.

Click on each part you wish to trim. Trimmed parts must always be between cutting edges. You can cut trimming edges as well as non-trimming objects.

The local menu also provides you the Select All option to select all 2D elements in the drawing to use as cutting edges.

Additional Local Menu Options:

Trim by Line: Enables you to define the cutting edge by selecting two points.

Multiple:



Multiple option lets you select more than one entities to be used as the cutting edge.

Split



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Split

Ribbon UI Menu:

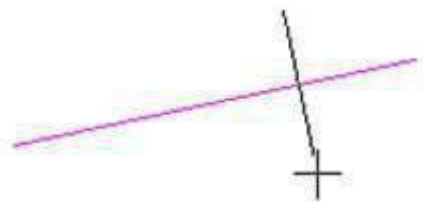
Divides an object into two separate objects. This tool works on any single-line or double-line object, as well as on circles, arcs, and curves. You can split by using the cursor, or by specifying a ratio in the Inspector Bar. The ratio is the length of the first resulting segment to the length of the second resulting segment. The first segment starts at the start point of the object, which depends on how the object was created.

Splitting Lines and Double Lines

Select the line or line segment you want to split.



Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.

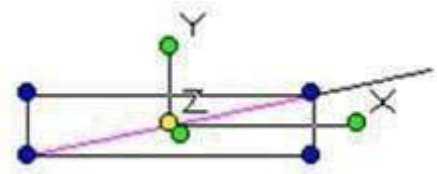


3. Click to locate the split. A temporary red X indicates the location.

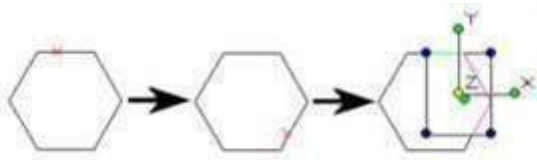


Y

You can verify that the line is now two separate objects by selecting one of the lines.



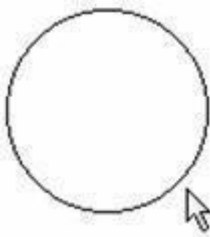
To split a closed object such as a rectangle or polygon, you need to create at least two splits. For each one, click on the segment you want to split.



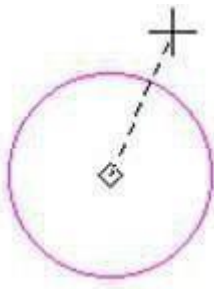
When splitting a polyline, you place the split on one segment (line or arc), but the entire polyline is split at the selected point.

Splitting Arcs and Circles

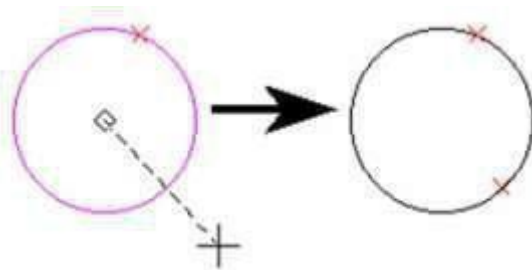
1. Select the circle, arc, or arc segment you want to split.



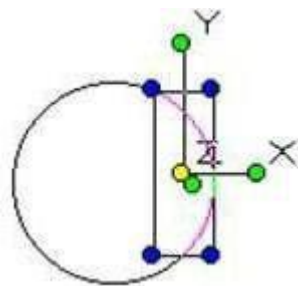
Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.



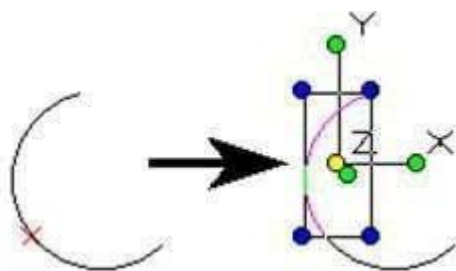
Click to locate the split. A temporary red X indicates the location. For a circle, indicate a second split point.



You can verify that the circle is now two separate objects by selecting one of the arcs.



For arcs, you only need to define one split point.



Splitting Curves

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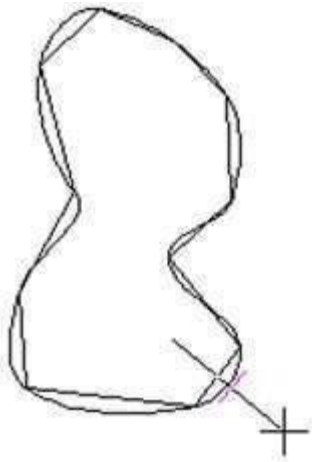
It is helpful to display the curve's frames before splitting it. This is because the split actually takes place on the line segments of the frame.

To display the frame, open the **Properties** window to the **Curve** page and check **Show Frame**.

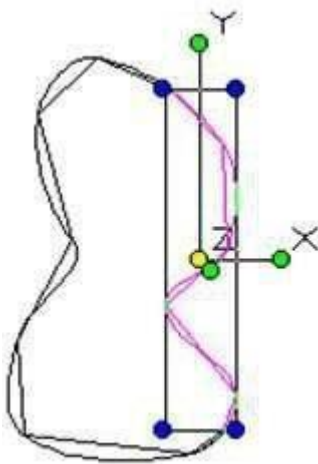
Select the curve. Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.



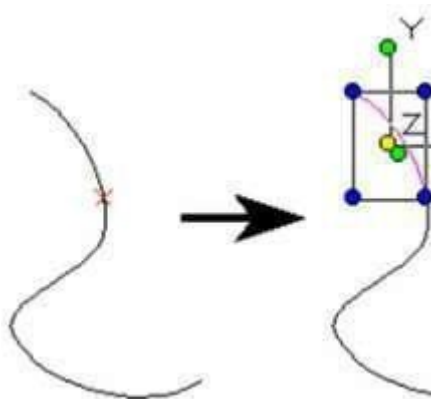
Click to locate the split. A temporary red X indicates the location. For a closed curve, select a second split point.



You can verify that the curve is now two separate objects by selecting one of the segments.



For open curves, you only need to define one split point.



Splitting 3D Curves

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It is helpful to display the curve's frames before splitting it. This is because the split actually takes place on the line segments of the frame.

Select the curve. Position the split indicator where you want the split, or enter the split ratio in the Inspector Bar.

Click to locate the split. For a closed curve, You may want to select a second split point.

3. You can then move the split off section.

Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with walls.

Trim by Entity



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Trim by Entity

Ribbon UI Menu:

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Uses an object as a cutter to trim multiple objects.

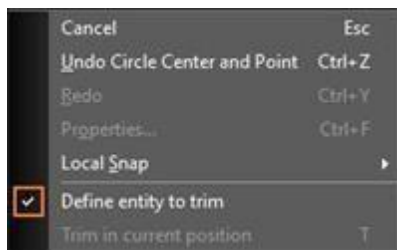
Select a closed 2D object (cutter) to cut another object/ object placed inside it.

Transform/ move the selected cutter to cut the object inside it.

Local menu options:

Define Entity to Trim: Enables you to cut the defined entity.

i) Enable 'Define entity to trim' in the inspector bar or local menu.

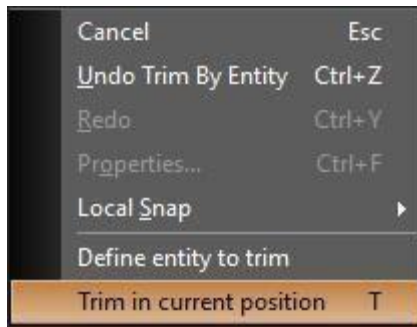


ii) Move the selected cutter to cut the defined object.

Trim in current position: Enables you to trim object in current position.

Select 'Trim in current position'.

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As soon as the option is selected the object is trimmed at current position.

Demo Video:

<https://youtu.be/xMMWxjQQ2bk>

Stretch

(Available in Platinum, Professional and Deluxe)

Note: Stretch tool is not available in Designer



Default UI Menu: Modify/Stretch

Ribbon UI Menu:

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Moves a group of nodes of one or more objects by defining a movement vector.

Drag a selection window from right to left around the elements you want to stretch. Be careful to only select nodes that you want to move. If you select an entire segment (line or arc) or an entire object it will be moved and not stretched..

Right click and select Finish

Select two points to define the movement vector, or enter the length and angle in the Inspector Bar.

The selected nodes move by the defined vector.

Local menu option:

Open Window Mode: It lets you select the object by a selection window.

Power Stretch:



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Power Stretch

Ribbon UI Menu:

Moves a group of nodes of one or more objects by defining a movement vector. You can select multiple objects by using shift key.

Local menu options:

Multiple: If this option is selected the stretch tool will continue to operate until you press ESC or select another tool. Multiple options appears in LTE workspace in local menu and inspector bar for power stretch. In default workspace, objects can be selected using shift key.

Displacement: It appears only after choosing "Finish" for selection operation. If this option is selected the node or object will be displaced by the UCS X and Y value of the point you select. For example if you click on the coordinate X= -10 and Y= 5 the displacement will be -10 in the X axis and 5 in the Y axis. Displacement option for power stretch activates in LTE workspace only.

T-Meet 2 Double Lines



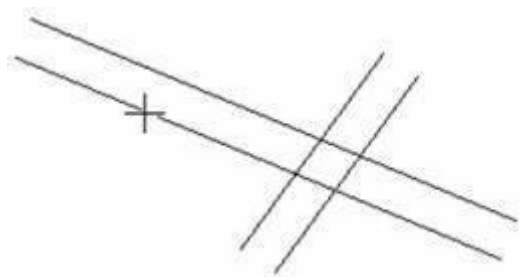
(Available in all TurboCAD Variants)

Default UI Menu: Modify/Intersect Double Lines/T-Meet 2 Double Lines

Ribbon UI Menu:

Forms a T intersection of two double lines. The first double line selected will shrink or extend to meet the intersection.

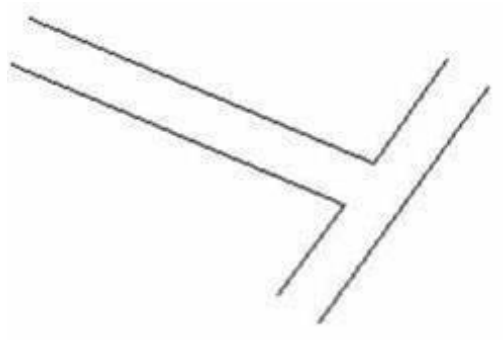
Select the double line that will be the stem of the T. Click on the portion you want to remain.



2. Select the line that will be the top of the T.

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The T intersection is created.



If the stem of the T does not meet the other line, it will be extended to create the intersection.

Local menu options:

See [Double Line Options](#)

Thick Profile



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Thick Profile

Ribbon UI Menu:

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Draws grid(s) of lines/polylines/circles/arcs on plane. It is a another convenient way to create 2D profiles (closed objects with area) from polylines and curves.

You can take either an open or closed 2D Line, Arc, Polyline, or Curve (Spline or Bezier) and 'thicken' the profile of that entity by defining a distance on both sides of that entity where that thickening will occur.

Apply thick profile tool with defined width and corners type options to groups of entities

As result user gets set of polylines, set of connected polylines (these two options are for TC versions without ACIS), region or set of sheet ACIS entities (those two are for Pro Platinum)

Received entities can be used for extrudes/thick profile.

Note: The Thick profile tool automatically calculates thicken profiles accordingly to width defined for original entities

For example:

1. Apply to a closed Polyline

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Original closed polyline with line and arc segments:

After Thicken Profile (of 15 mm each side) applied and vertices filleted:

After clicking 'Finish, final Thick Profile object:

2. Apply to an open Arc

Original Arc Segment:

Apply thick profile of 5 mm to each side of arc:

Final Thick Profile Object:

Local menu options:

Chamfer Corners: Corners of the resulting object are chamfer.

Fillet Corners: Corners of the resulting object are fillet.

Ext ended Corners: Corners of the resulting object are extended.

As Set of Polylines: Resulting object will be a set of polylines.

As Set of Joined Polylines: Resulting object will be a set of joined polylines.

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Region: A region will be created as a result that can be extruded.

As sheet entity: Resulting object will be an ACIS solid which can be used for sheet thicken.



Demo Video:

<https://youtu.be/fNaAw6mz9RI>

Convert to Curve



(Available in all TurboCAD Variants)

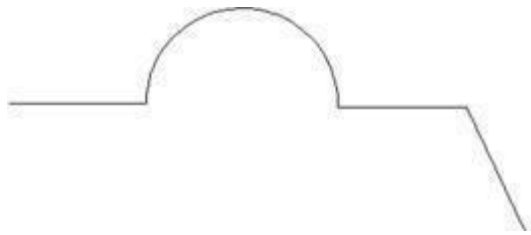
Default UI Menu: Modify/Convert to Curve

Ribbon UI Menu:

Converts a 2D object into a Bezier curve. Any single-line 2D object can be converted. Objects like double lines, dimensions, and text must be exploded first before converting.

Start with a 2D object such as a polyline. This example uses a polyline consisting of both linear and arc segments.

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If you select the polyline and view the Selection Info Palette, you can see the object's type.

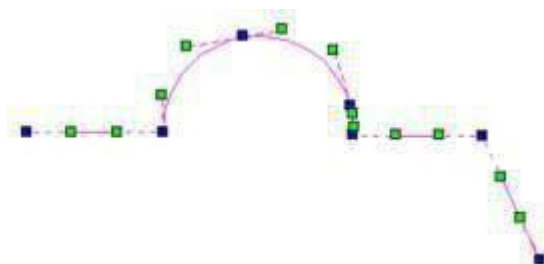


To convert the polyline, select it and activate the **Convert to Curve** function. The **Tolerance** value sets the precision of the conversion; a small tolerance means high precision, and therefore more control points available for node editing.

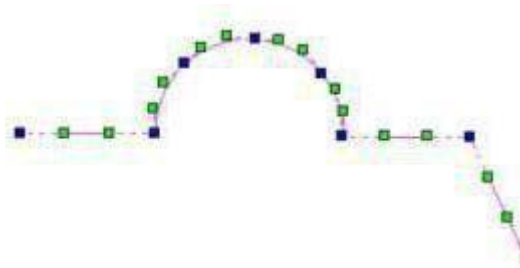
The polyline is replaced by a Bezier curve. This can be verified in the Selection Info Palette.



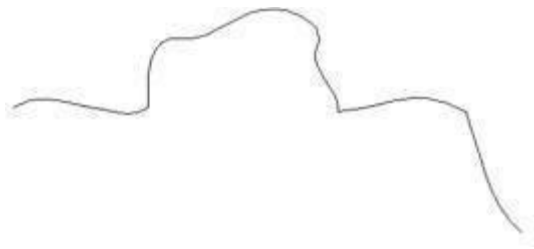
If you edit the curve with the **Edit Tool**, you can see the control points available for changing the curve's shape. (For details on Bezier options, see [Curve Properties](#).) This curve used a low precision (Tolerance = 0.1).



This curve used a higher precision (Tolerance = 0.001).



Nodes and control points can be moved to change the curve's shape.



Convert to curve also works on 2D objects that are converted into 3D by entering a thickness in the **3D** page of the **Properties**.

Sharp corners will remain sharp when two adjacent line segments cannot be approximated by an arc within the specified tolerance. This could happen when the vertex is too "sharp" which is defined by vertex angle, line lengths, and tolerance.

Convert to Arc Polyline

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Convert to Arc Polyline

Ribbon UI Menu:

Converts a 2D object into a Polyline composed of arcs. Any single-line 2D object can be converted. Objects like double lines, dimensions, and text must be exploded first before converting.

Start with a 2D object such as a polyline. This example uses a polyline consisting of both linear and arc segments.

To convert the polyline, select it and activate the **Convert to Arc Polyline** function.

The **Tolerance** value sets the precision of the conversion; a small tolerance

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means high precision, and therefore more control points available for node

editing.

6.8. Tracing

(Available Only in Platinum, Professional, and Deluxe)

The **Trace** tools originate from the stand-alone ScanPro application, which has now been incorporated into TurboCAD. They enable you to create 2D raster-to-vector tracings of inserted pictures (see [Inserting a Picture](#)) or of geometric objects.

Trace tools are available on the **Tools** toolbar, which you can display by right-clicking in any toolbar area and selecting **Tools**. Before creating a tracing, it's important to understand the local menu options, since these greatly affect the appearance and quality of the trace.

Trace Options: Settings for trace quality.

The **General** page provides options for the **Picture Type**. The **Detail Level** and **Color Level** values update with the picture type.



The **Trace** page controls the quality of the tracing.



Smooth Level: Reduces jags and discontinuities in lines and curves.

NoiseLevel: Reduces spots.

Thinning Level: Reduces line widths.

The **Lines Connection** page contains parameters relevant when **Connect Lines** is

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activated.



Max Gap for Connection: The largest gap which will be closed when connecting lines. **Max Length for Deletion:** The largest segment that can be deleted, i.e., breaking the lines on either side.

Ortho Angle: The angle at which line segments will be joined into polylines.

Curve Recognition: Traced polylines will be created as curves. Otherwise they will be created as lines and polylines.

Preview Mode: A trace preview will appear in magenta over the image.



Tracing Colors: By default, all colors will be used for tracing. This option is used when you want to trace around only selected colors.



The following two options are relevant only for **Trace by Point**:

Show / Hide Trace Rectangle: Toggles the display of the tracing rectangle, defined by its aperture size.

Nearest Graphic Only: Traces only the object closest to the tracing rectangle's center, identified by crosshairs.

Trace by Point

(Available in Platinum, Professional and Deluxe)

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This tool is basically the same as **Trace by Rectangle** , with a different way of defining the trace area. The selection area is sized by the **Aperture** field, which is the length of each side of a square whose center is represented by crosshairs.

Using the same example picture as for **Trace by Rectangle**, place the cursor at the center of the desired selection area.

The results are the same - all objects within the area will be traced.

If **Nearest Graphic Only** is active, then only the object closest to the crosshairs will be traced.

Trace by Rectangle

(Available in Platinum, Professional and Deluxe)

Creates a tracing of all objects within a selection rectangle. This example uses the following inserted picture:

To easily see the results of the trace, it is good practice to place the inserted picture on its own layer (see [Snap Modes](#)), which can then be blanked.



1. Activate **Trace by Rectangle** and define the selection rectangle. (You can also use fields in the Inspector Bar to control the rectangle size.) If **Preview Mode** is on, the traced lines will appear in magenta.

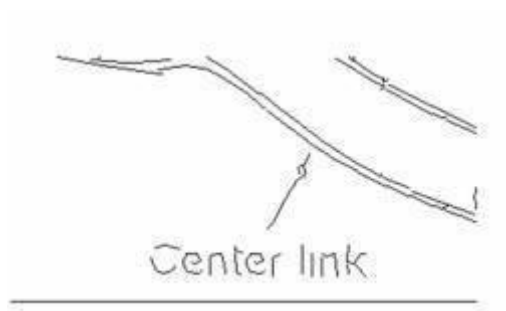
2. Select **Finish** from the Inspector Bar or the local menu.



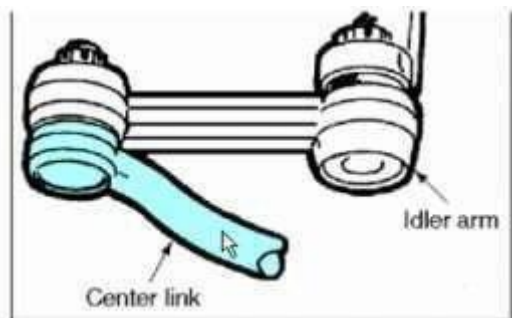
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3. Move or delete the picture, or simply blank its layer, to see the tracing.

In the above example, the text lines did not trace completely. To improve trace quality, you can increase the zoom of the traced objects.



The previous example traced around all colors. If you only wanted to trace around the shaded area of the center link, you would use the **Tracing Colors** option, which must be set before starting the trace.

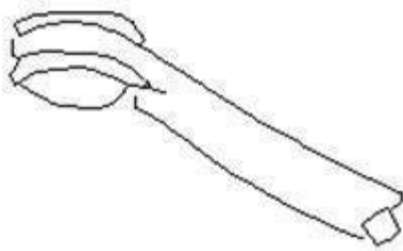


When the **Select Colors to Trace** window opens, click on the colors you want to include. The colors then appear as blocks in the window.

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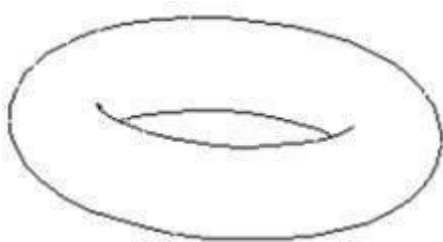


The result is a trace only around the center link.



Note: Trace colors remain active for future traces, so it is important to click **Reset** (to zero colors) if necessary.

In addition to pictures, you can also trace around geometric objects. This example is a solid torus, rotated and displayed in Hidden Line mode.



The trace is created in the current viewing plane. The result is shown below, rotated so that the 2D trace can be seen:

6.9. Remove Overlapping

Remove Overlapping

(Available in TurboCAD all versions)

Default UI Menu :Modify/Remove OverLapping

Ribbon UI Menu:

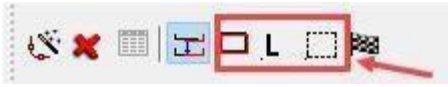
This will remove overlapping lines and arc,it allows you to simplify the drawing by removing duplicate entities.

Combine collinear objects that partially overlap.



Ignores Colour,Layer,line type.

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6.10. Construct Similar

Construct Similar

(Available in TurboCAD all variants)

Default UI Menu: Modify/Construct Similar

Ribbon UI Menu:

The “Construct Similar” tool takes properties from one object and applies them onto the active tool. You can set an object's properties before it is created in one click. Analogous to the ‘Format Painter’ tool found in MS Office applications.

To use Construct Similar, first choose the Object type you wish:

Menu  **Construct Similar**

The menu item is enabled if the active tool is a tool for the construction geometry ('Draw Line', 'Draw Arc' etc.).

Command Description: Construct similar object

Tool Prompt: Select the entity to construct similar

Demo Video: You can learn in detail about the functionality of this tool from the below video.

<https://youtu.be/CLh5akJccfE>

6.11. 2D Edit Mode

2D Edit Mode



(Available in TurboCAD Platinum and Professional)

Default UI Menu: Edit/2D Edit Mode

Ribbon UI Menu:

2D Edit Mode makes work easier for 2D users. When enabled, it turns 3D object accessibility off and lets the user work conveniently in a 2D drawing and editing mode.

When the 2D Edit Mode is switched on, all the 3D Drawing and Modification tools as well as 3D WorkPlane options, which need Z coordinate, will be disabled.

Only the 2D selector mode is enabled. This restricts the object's coordinate system (CS) from changing. The 2D selector can only work in a selected object's CS. If the user selects an object, the CS of the selected object becomes the current work plane. After that, all 2D Drawing and Modification tools will create 2D objects on current work plane, different from World CS.

Several 2D WorkPlane options are also disabled – Auto WorkPlane by Face, Set Named WorkPlane, Edit Current WorkPlane, and WorkPlane Origin. If the 2D Edit tool is active, the Current WorkPlane changes to World CS.

7. Annotation

(Available in all TurboCAD Variants)

Annotation tools enable you to enhance your drawing with non-geometric objects, including text, dimensions, and hatching.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

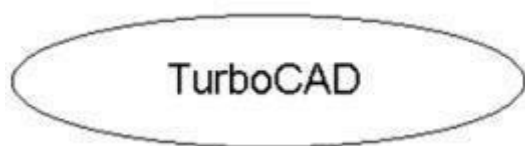
7.1. Text

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Text/Text

Ribbon UI Menu:

These tools enable you to add strings of letters and other characters into your model.



The text tools are available on the **Insert** menu, and can be accessed on the flyout toolbar on the **Drawing Tools** toolbar.



You can also display the **Text** toolbar by right-clicking on any toolbar area and selecting **Text**.



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TurboCAD supports Special Signs when entering text. Local menu options include: Degree, Plus/Minus, and Diameter signs.

Note: With this tool you can add single straight lines of text. To add multiple lines in paragraph format, see Multi Text . To create text that follows a curve, see Text Along Curve .

Editing Text

You can edit existing text by accessing its **Properties** (see [Object Properties](#)). Open the **General** page, and edit the text in the **Attribute** field.

You can also edit the content of the text by clicking on the text with the Text tool, or, if in the Selector General properties the Double Click Action is set to Edit content, you can edit the text content by double clicking on the text with the Selector tool.

If you use the Selection Info Palette, you can use the **Edit in Place** tool to edit text directly on the screen. See [Selection Info Palette](#). In this palette, the text is also listed (and can be edited) in the **General** category, next to **Info**.

Exploding Text

You can use **Explode** to explode text into individual characters, and explode characters into polygons and polylines, and explode these into lines. This can be useful if you need to change a very small aspect of text, or if you need a text-shaped polyline to extrude.

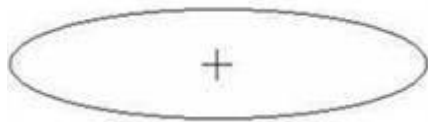
See [Exploding Solids](#)

Inserting Text

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Set the desired font and other text parameters. See [Text Properties](#) .

Click on the point where you want to place your text.



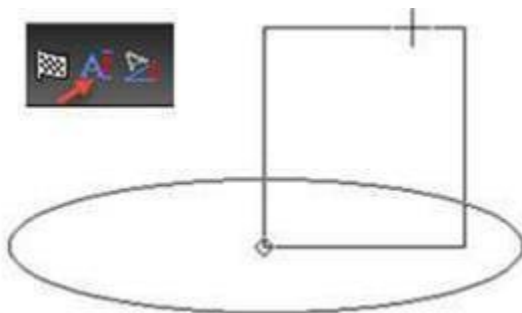
Type the text, using the Backspace key to make corrections. Press Enter to add a new text line. To finish, click on the drawing, press Shift+Enter, or select **Finish** from the local menu.

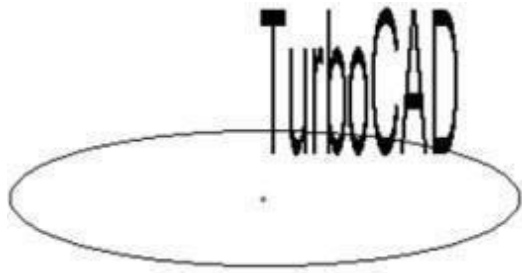


Note: By default, the text is centered at the insertion point. You can change this, however, via the **Properties** window, or by using the **Align** local menu option.

Local menu options:

Height: Changes the text height. Move the mouse to adjust the height rectangle, or enter a height in the InspectorBar.





Angle: Adjust the angle of the text line (not the text slant). Move the mouse to rotate the text rectangle, or enter the angle in the InspectorBar.

Text Properties

Certain parameters of text (color, font, and text height) can be set on the **Property** toolbar.



Other parameters (bold, italics, etc.) can be set in the **Text Properties** toolbar, which can be displayed by right-clicking in any toolbar area and selecting **Text Properties**.

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Note: You can use the Manager palette to create multiple styles of text, which set certain basic text properties such as font and height. See [Text Styles](#)

The **Text Properties** toolbar can be used while creating text (i.e. when **Text** is active), and it can be used to edit selected existing text. You can access all properties of the **Text** tool by right-clicking on the tool icon. For details on changing properties of selected text only, see [Object Properties](#)

Shx (AutoCAD compatible). The available TrueType fonts depend on your Windows setup. AutoCAD (.shx) fonts are installed by TurboCAD, and are located in the Program\Fonts folder.

Choose: Regular, Italic, Bold, or Bold Italic. These four styles are not available for every TrueType font.

Height: You can set the height in either World or Paper units, depending on the current mode - Model Space or Paper Space. Text can be measured in points only if you are using Paper units.

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Note: There are also options for setting text height in the **Space Units** page of the **Drawing Setup (Options / Space Units)**.

Mode:

Standard: Always proportional even if resized.

Scalable: Can be resized non-proportionally.

Flexible: Pen, brush and slant options are available. **Non-rotating, Non-zooming:** Text will not change position or be scaled when rotating or zooming.

Obliquity: Degree of text slant (slant of each character), available only for **Flexible** mode. A positive value slants to the left.

Justification: Left-justified text is typed to the right of the insertion point; right-justified text is typed to the left of the insertion point, and center-justified text is centered on the insertion point.

Tip: You can get more precise control of the placement of text relative to the insertion point by using **Align**, accessed by right-clicking in the drawing area while the **Text** tool is active. This feature can only be applied to text as it is being drawn; you cannot use it to format existing text.

Background Color: The color that appears behind each character.

7.2. Multi Text

Multi Text

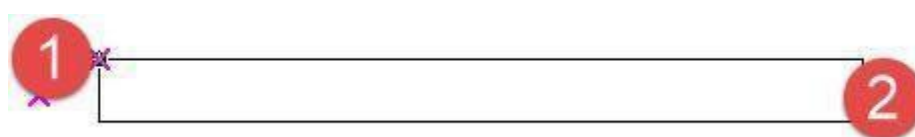
(Available in all TurboCAD Variants)

Default UI Menu: Draw/Text/Multi Text

Ribbon UI Menu:

Creates multi-line paragraphs that fit within a paragraph width boundary. Each multiline text object is a single object, regardless of the number of lines it contains.

Select the first point of the text, then a second point to define the paragraph width and angle. You can also use the Inspector Bar to enter the width and angle.



The **Character** palette opens, in which you can define the text properties that will be active for multi text created from now on. (You can change these properties mid-text.) These properties are explained below.

Font: Sets the color and font type. The "B" icon creates **boldtext**, the "I" icon creates italics, and the "U" icon underlines the text.



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Text Height: Sets the height of the tallest letter. The "X" icon enables you to set the height as a factor of the current height.



Width Factor: is the width multiplier of each character.



Oblique angle: Sets the italicization angle



Symbols: While entering the text, you can insert a degree, plus/minus, or "phi" (angle) symbol.



The ellipses button open the Character map.

The a/b is used to insert a fraction. You can chose to divide the fraction by a line or a slash. Usability has been improved by the addition of OK and Cancel button in the dialog box of the Insert Fraction in the Text Editor palette.

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Demo Video:

<https://youtu.be/hvOjIHbuBfo>

These three icons are used for top, center, or bottom **Justification**.



Paragraph: This button opens the Paragraph properties for the selected section of text:

Import / Save: Once text is created, you can **Save** it as a .txt file. This text can later be inserted elsewhere by clicking **Import**.



Insert Column Break: Generates a new column.

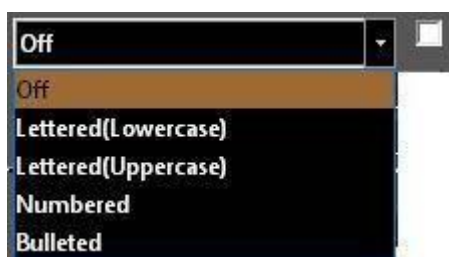


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To delete a column break move the insertion cursor to the top of the column in front of the first character, and then press the Backspace key. The distance between the columns is set by the Column Gutter Property

10. **List:** Allows insertion standard Bulleted and Numbered lists.



To create the text, simply start typing. The text will start where you defined the first text point. You can enter line breaks manually, and breaks will be created by word-wrapping, according to the paragraph width you set.

If you want to change text properties (font, height, etc.) while typing, make the change in the **Character** window and continue entering text. You can also enter symbols by clicking the relevant icon in the **Character** window. While entering text, you can use the Backspace key or the **Undo** and **Redo** icons. 

In the Inspector Bar or local menu, select **Finish** when the text is complete. You can also choose **Cancel** which will terminate the text editor without any changes being

made.



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The text is complete, and the preview box disappears.

Editing Multi Text

You can edit text before or after it is created by editing it on-screen:

You can also edit the content of the multi-text by clicking on the multi-text with the Multi Text tool, or, if in the Selector General properties the Double Click Action is set to Edit Content, you can edit the multi text content by double clicking on the multi text with the Selector tool.

You can also use the old mode of editing Multi line text. Make sure you are in **MultilineText** mode (**Draw / Text / Multi Text**) and select **Edit Text** from the local menu, command line, or Inspector Bar.



The cursor appears as the letter "A"; select the text you want to edit.

Edit the text as you would in any text editor. To make a change to a specific string, such as the first word in this example, highlight that string.

Set the properties for that string in the **Character** window. In this example, the text was underlined and its height was increased.

6. Select **Finish** to implement the change.

Multi Text Properties

If you open a multiline text object's **Properties** (see [Object Properties](#)), there are certain parameters of the text you can edit.

The properties also appear in the Selection Info palette, when the multiline text is selected. All parameters can be edited, except for the text string itself.

Font: Sets the font name, style, and height.

Mode: Select **Scalable** for text that can be re-sized non-proportionally or **Flexible** if you want pen, brush and slant options available.

Oblique Angle: Available for **Flexible** mode only, sets the slant of the text (slant of each character).

Width Factor: Sets the width of each character.

Justification: Adjusts the text relative to its paragraph box.

Factor: Sets the distance between lines.

St yle: Select **At Least** to automatically set the line size relative to the largest character of a line. Select **Exactly** to keep all lines the same size. **Angle:** Sets the angle of the paragraph.

Width: Sets the width of the paragraph.

7.3. Text Numbering

Text Numbering



(Available in all TurboCAD variants)

Default UI Menu: Draw/Text/Text Numbering

Ribbon UI Menu:

Text Numbering enables you to select multiple text entities and to choose a starting number plus an increment and the text would either be prefixed or suffixed by the auto-numbering or the auto-numbering will replace the existing text. this is extremely useful when you are trying to count up hundreds of parking places. You can copy a single text entity into each parking place then select all the text and auto number thereby counting each parking place.

Enabling this tool, you are able to select multiple text entities and generate numbering which can be sorted by X, Y, or Order.

To generate text Auto-Numbering:

Enable "Text Numbering" from menu

Select multiple Text entities

Choose desired sorting and placement order from the Inspector Bar or Local Menu

Select Finish. Text is Auto-numbered

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Before Numbering:

After Numbering:

Local menu options:

Sort:-

Sort by X: Generates text numbering according to the 1st to last entity appearing in x-axis.

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Sort by Y: Generates text numbering according to the 1st to last entity appearing in y-axis.

Sort by Order: Generates text numbering according to the 1st to last entity being selected by the user.

Placement:-

Overwrite: Overwrites the text with the text numbering.

Prefix: Adds text numbering before the original text.

Suffix: Adds text numbering after the original text.

Find & Replace: Finds the text written in "Find" and replaces it with the chosen numbering format. The format can be changed, by default its <N> (representing number). It can be changed via "Format" in Inspector Bar.

Additional Inspector Bar Options:-



Start: Sets the starting number value in the sequence of text which will be auto-numbered

Step: Sets the incrementing number amount for each, subsequent text item being numbering

To generate text numbering:

Enable "Text Numbering" from menu

Select multiple Text entities

Choose desired sorting and placement order from the Inspector Bar or Local Menu

Select Finish.

7.4. Dimensions

(Available in all TurboCAD Variants)

Default UI Menu: Dimension

Ribbon UI Menu:

TurboCAD contains a variety of dimension tools that you can use to display the measurements of lines and angles.



Dimensions consist of three basic components:

Dimension lines, with arrows pointing to either end of the dimension. A linear dimension can have an interior dimension line or two exterior dimension lines. Exterior dimension lines can be supplemented with an optional interior line.

Extension lines, which connect the dimension line to the object being dimensioned.

Extension lines have optional line segments that continue the extension outward beyond the dimension line (extensions to the extension).

Dimension text, typically displaying the distance being dimensioned in World units.

Note: For information on editing dimensions, see [Editing Dimensions](#). To create a group of dimensions automatically, see [Auto Dimension](#). To constrain dimensions, see [Constraining Dimensions](#)

Associative Dimensions

(Available in Platinum)

When a dimension is associated with objects, you can freely move, rotate, or resize the associated objects and the dimension will retain its position relative to the objects, and the dimension text will change automatically to reflect changes.

Note: To make dimensions associative to other dimensions, see [Constraining Dimensions](#)

Before creating associative dimensions, associativity needs to be activated in the default **Properties** of the dimension tools. Associativity cannot be set in the **Properties** window of a selected dimension; this parameter must be set before the dimension is created. Right-click on any dimension tool to bring up the **Properties** window, and open the **Format** page. Make sure that **Associative Dimension** is checked.

Tip: To tell whether a dimension is associative, select the object. Any dimensions associated with that object will be colored blue. There are the following limitations on associativity:

Dimension cannot be associative if you override the automatic dimension text, either in the Inspector Bar or in the **Attribute** field of the **General** page of a dimension's **Properties**.

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Datum dimensions cannot be associative. However, grouping datum dimensions with their objects can achieve a similar effect to associative dimensioning. Be sure to explode to edit objects later.

The following dimensions will only be associative if originally created using **Snap to Vertex** or **Snap to Center**, or if created using **Segment** or **Entity Dimensioning**: Baseline, Continuous, Incremental, Leader Orthogonal, Parallel.

When dimensioning over a Viewport in Paper Space, the dimensions will associate with the viewport itself, but not with the corresponding objects in Model Space. See [Viewports](#)

To break the associativity between a dimension and its objects, select both the object(s) and the dimension. Right-click and select **Drop Link** from the local menu.



Creating Dimensions

(Available in all TurboCAD Variants)

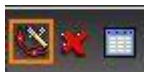
Creating dimensions involves two basic steps: selecting the objects to be measured, and locating the dimension. By default the magnitude of the dimension is calculated automatically and is recorded as the dimension text, measured in current World units. (Leader dimensions, which contain text labels, are the exception to this rule.) You can override the default text of a dimension by specifying the text in the Inspector Bar prior to finishing the dimension.



You can also change the dimension text in the dimension's **Properties**, by changing the **Attributes** field of the **General** page.

Note: To create several types of dimensions automatically in one step, see [Auto Dimension](#)

When **Auto Add Constraints** is active, any dimensions you assign are created as variables that appear in the Calculator Palette.



Dimension variables can be constrained to other dimensions, or to other variables or numbers. This is particularly powerful when used in conjunction with **Auto Constraints** [Constraining Dimensions](#). See

Orthogonal Dimension

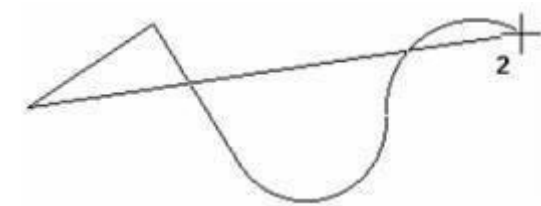
Creates a horizontal or vertical dimension.

Note: To create orthogonal dimensions automatically, see [Auto Dimension](#)

1. Select the first point.

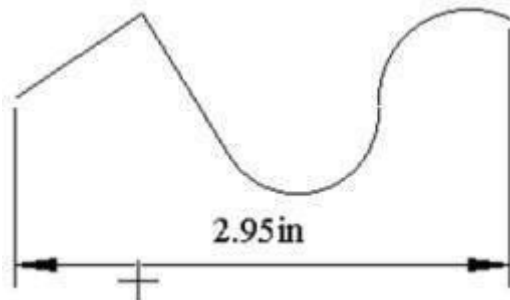


Select the second point, or enter the length and angle of the dimensioned line in the Inspector Bar.

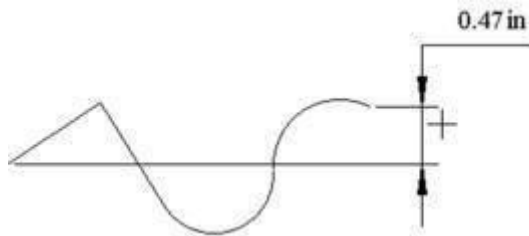


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For a horizontal dimension, move the mouse above or below the two points and click to define the location. You can also enter the length of the witness lines in the Inspector Bar.



4. For a vertical dimension, define the dimension at either side of the two points.



Local menu options:

Segment Dimensioning, Entity Dimensioning

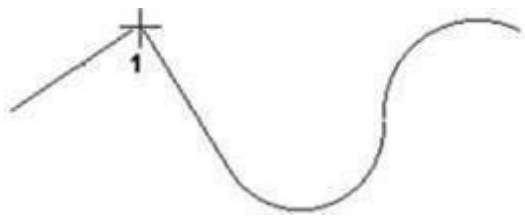
Horizontal / Vertical Mode only: creates either type of dimension, no matter where the dimension is located.

Parallel Dimension

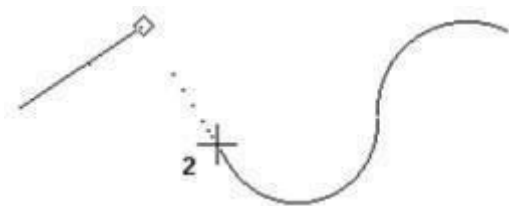
Creates a dimension showing the absolute length of an object.

Note: To create parallel dimensions automatically, see [Auto Dimension](#)

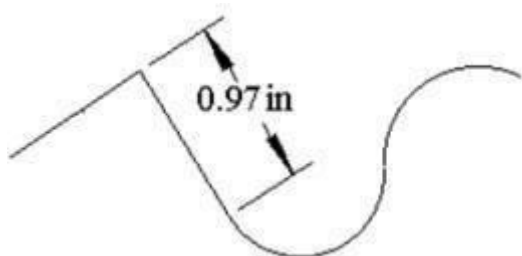
1. Select the first point.



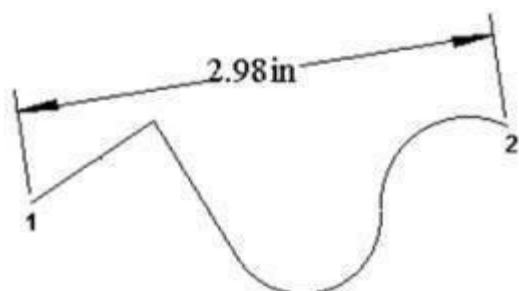
Select the second point, or enter the length and angle of the dimensioned line in the Inspector Bar.



Move the mouse and click to define the location. You can also enter the length of the witness lines in the Inspector Bar.



For any two endpoints, the absolute length is parallel to the line between the points.

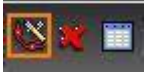


Local menu options:

Segment Dimensioning, Entity Dimensioning

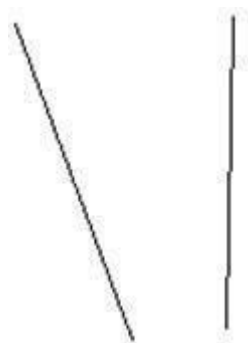
Distance Dimension

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Sets two lines parallel to one another and places a constrained dimension between them. For this tool to be available, **Auto Add Constraints** must be on. 

Note: To create distance dimensions automatically, see [Auto Dimension](#)

1. Select the two lines, or select them by snapping to either of their endpoints.



2. Place the dimension, and the lines are made parallel.

Note: Because **Auto Add Constraints** is on, the Distance dimension appears as a variable in the Calculator Palette and can be edited. See [Constraining Dimensions](#)

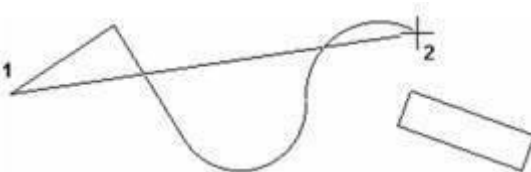
Rotated Dimension

Creates a dimension projected in a specified direction.

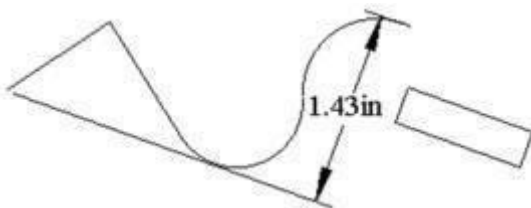
Select two points to define the dimension's direction. The dimension will be measured normal to this line. You can also define the first point, then specify the angle of the vector in the Inspector Bar.



Select the first point of the object to be dimensioned. Select the second point, or enter the length and angle of the dimensioned line in the Inspector Bar.



Move the mouse and click to define the location. You can also enter the length of the witness lines in the Inspector Bar.



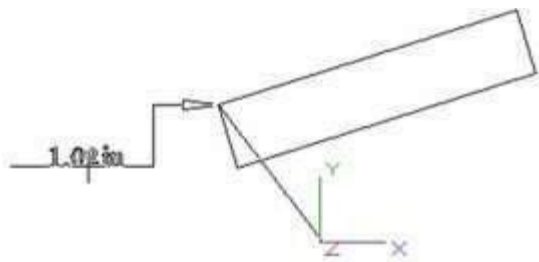
Local menu options:

Segment Dimensioning, Entity Dimensioning: Available only after the dimension direction has been defined.

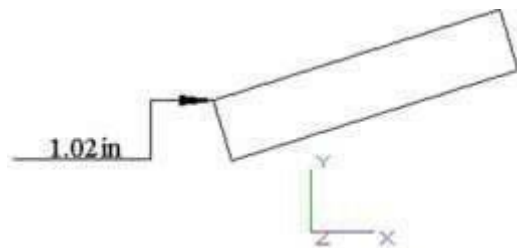
Datum Dimension

Creates a dimension showing the horizontal or vertical distance from a point. By default, the point is the absolute origin, but you can change this.

Select the point whose distance from the origin is to be displayed. A rubberband line appears, indicating the origin of the dimension.



To display the Y coordinate (horizontal dimension text), move the mouse to either side of both the origin and the selected point, and click to locate the dimension. You can also enter the angle and length of the extension lines in the Inspector Bar.



To display the X coordinate (vertical dimension text), move the mouse above or below both the origin and the selected point.

Local menu options:

Set Origin: By default, datum dimensions are created relative to the absolute origin. Use Set Origin to select a new origin. This origin will remain in effect until changed.

Baseline Dimension

Creates a series of parallel linear dimensions that follow the axis of an existing linear dimension.

Select an existing linear base dimension. Click near the side you want to serve as the baseline. A temporary rectangle appears at the baseline end.

2. Select the first point where you want a new baseline dimension.

Continue selecting points. Each new baseline dimension will be created at an offset from the previous one.

Note: The offset distance is controlled by the **Baseline Increment** value on the **Advanced Format** page of the **Properties** window.

When finished, select **Cancel** from the local menu or press Esc.

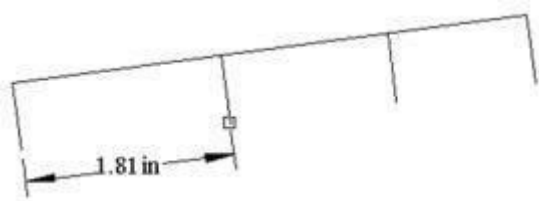
Local menu option:

Select Dimension: Select a new base dimension.

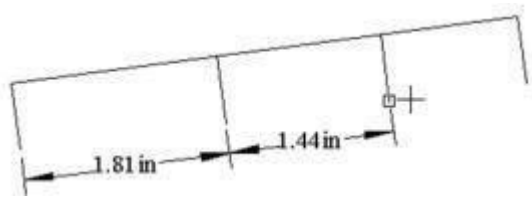
Continuous Dimension

Creates a series of parallel linear dimensions measured from the previous dimension. The dimensions follow the axis of the base dimension.

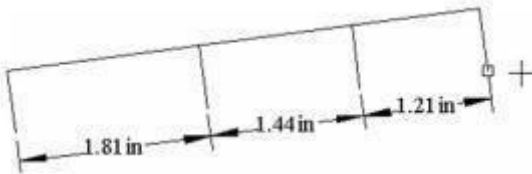
Select an existing linear base dimension. Click near the side you want the next dimension to be created. A temporary rectangle appears at the baseline end.



2. Select the first point where you want a new continuous dimension.



Continue selecting points. Each new continuous dimension will be measured from the previous dimension.



When finished, select **Cancel** from the local menu or press Esc.

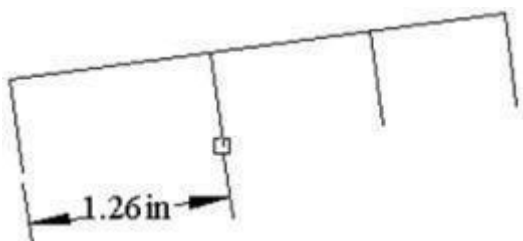
Local menu option:

Select Dimension: Select a new base dimension.

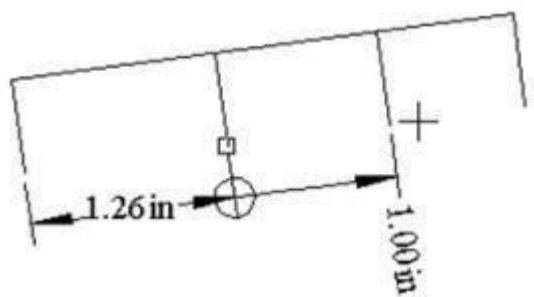
Incremental Dimension

Creates a series of parallel linear dimensions measured from the previous dimension. The dimensions follow the axis of the base dimension, and are displayed normal to the axis.

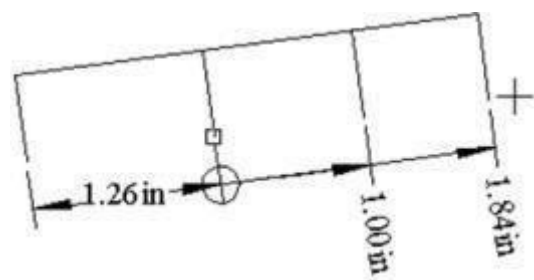
Select an existing linear base dimension. Click near the side you want the next dimension to be created. A temporary rectangle appears at the baseline end.



2. Select the first point where you want a new incremental dimension.



Continue selecting points. Each new incremental dimension will be measured from the previous dimension.



4. When finished, select **Cancel** from the local menu or press Esc.

Local menu option:

Select Dimension: Select a new base dimension.

Angular Dimension

Creates dimensions measuring angles. You can dimension the angle between two lines, the angle of an arc, between two points of a circle, and between a node and two points.

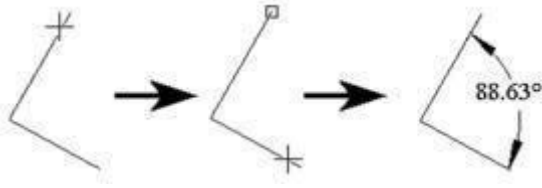
Note: To create angular dimensions automatically, see [Auto Dimension](#)

Angle Between Two Lines

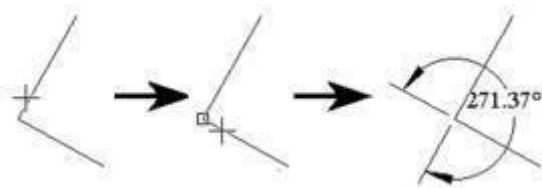
1. Select the two lines.

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Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.



Be careful where you select the lines. If you select close to the angle vertex, you could dimension the complementary angle.



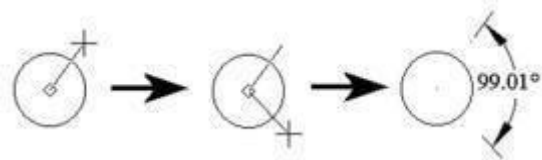
Angle within a Circle

Select the circle.

Select the start angle, or enter the angle in the Inspector Bar.

Select the end angle.

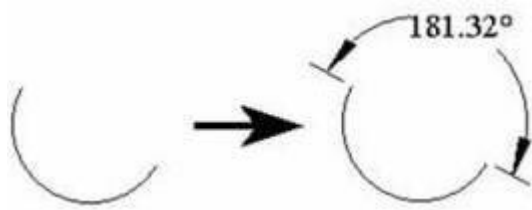
Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.



Angles of an Arc

Select the arc.

Move the mouse to dimension the acute or obtuse angle. Click to locate the dimension, or enter the length and angle in the Inspector Bar.



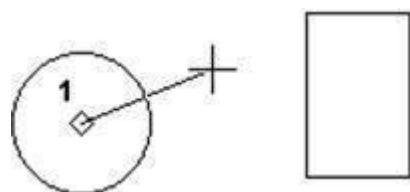
Note: If **Arc Length** is checked in the **Format** page of the **Properties** window, the arc length will be dimensioned instead of the angle.

Local menu option:

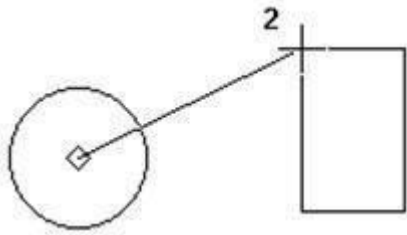
Angular Dimension option is available in local menu

Angle Node (freeform angle): Dimensions an angle by selecting the angle vertex then two points.

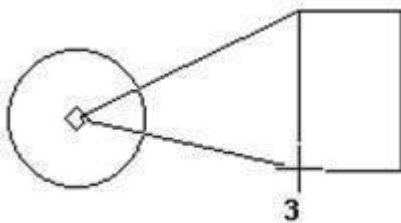
1. Select the angle vertex of 2D entity.



2. Define the first angle endpoint.



3. Define the second angle endpoint.



After clicking the second angle endpoint, the obtuse Angle Node dimension is displayed. To see the acute Angle Node dimension move your mouse cursor down away from the display of the Obtuse Angle Node Dimension

Example:

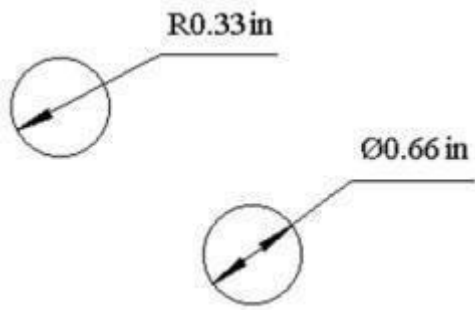
Radius - Diameter Dimension

Dimensions the radius or diameter of an arc or circle.

Note: To create radius or diameter dimensions automatically, see [Auto Dimension](#)

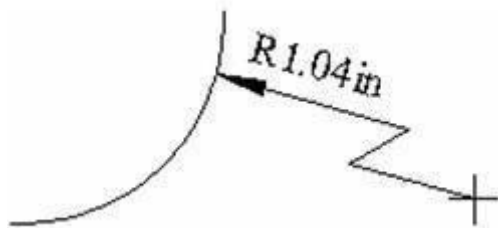
Select the arc or circle.

Move the mouse and click to locate the dimension, or enter the length and angle in the Inspector Bar.



Local menu option:

Large Radius: If the arc center is out of your drawing space, use this option to display the dimension from outside.



Leader

Creates dimension text attached to a simple line leader pointing to a location in your drawing. The leader is similar to a polyline.

Note: Check **Draw as Spline** on the **Advanced Format** page of the **Properties** window to create a curves leader rather than line segments.

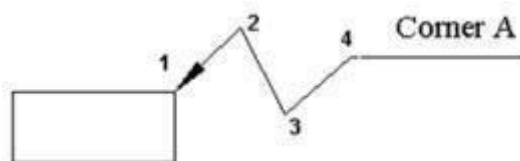
Enter the text in the Inspector Bar.

Select the leader start point (the end with the arrow). Select or more additional segment endpoints, or enter the length and angle of each segment in the Inspector Bar ---

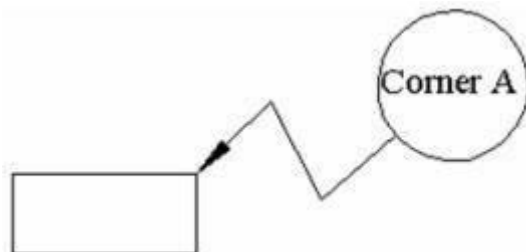
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Note: The arrowhead is defined by **Arrowheads / 1st** on the **Format** page of the **Properties** window ---

3. Double-click to finish, or select **Finish** from the local menu, or press Alt+F.



A text box can be created, by selecting one from the **Advanced Format** page of the **Properties** window.



Wall Dimensions

There is a special tool for dimensioning walls created via the **Architecture** tools.

See [Wall Dimension](#)

Move Text for Dimensions

(Available in Platinum)

The orthogonal, Parallel and Rotated dimension tools have a feature called Move Text available from the Local menu and the Inspector bar.

When this option is selected a final step is added. After the dimension is placed the user can drag the text to an offset position.

Click to set the final position for the text.

Dimension Properties

(Available in Platinum)

Controls the format of dimensions. You can set the properties for all dimensions, or for just a single dimension. See [Object Properties](#)

Text

There is no **Text** page in dimension **Properties**, but parameters can be set in the Selection Info palette. For details on **Text** parameters.

Format

Controls the shape of dimension arrows and the position of text relative to the dimension line.

Variable Name and Value: If the dimension is produced as a result of **Auto Dimensions** (most commonly used together with **Auto Constraints**), the dimension will have a variable name and value assigned to it. This value appears in **Properties / Format** window, as well as the Calculator Palette.

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Note: See [Auto Dimension Auto Constraint](#) and [Calculator Palette - Variables Palette](#)

Dimension Size Scale: The scaling factor for displaying the dimension.

Arc Length: Available for **Angular** dimension, displays the arc length rather than degree measurement.

Associative Dimension: Checked by default. Associative dimensions retain their positions and the dimension text will update when their associated object changes. To tell whether a dimension is associative, select the object. Any dimension associated with the object will be colored blue.

Note: This parameter is only available when setting the default properties of dimension tools (see [Object Properties](#)). You will not see it in the **Properties** window for a selected dimension. This parameter must be set before the dimension is created.

Arrowheads:

1st and 2nd: Select arrow shapes for the start and end of the dimension.

Leader: Select the arrow shape for leader dimensions.

Size: Length of the arrowhead or diameter of the dot

Tick Extension: specify the length of the tick extension line. You can specify a custom

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user defined arrowhead. User defined arrow heads are based by selecting a block.

Text:

Text Style: Select the Text style for the dimension. Text styles are specified in the Style Manager.

Height: Sets the height of the dimension text.

Font: Specifies the font use in the dimension text.

Color: Sets the color of the dimension text.

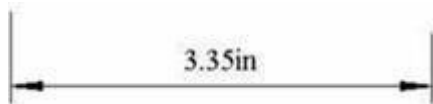
Frame around text: Draws a rectangular frame around the dimension text.

Fill Color: Specifies a color which will fill the text frame. The options are: None, Background and Custom. Background fills the frame with the background color. Custom allows you to select a color.

Text Position: Controls the dimension text position:

Text Movement: These options control how the dimension text is moved when editing the dimension. See [Editing Dimensions](#)

Horizontal: Select the text alignment with respect to witness lines. The preview window displays each option. Options vary for dimension and leader text.



Dimensions - Centered



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Dimensions - First Extension Line



Dimensions - Second Extension Line

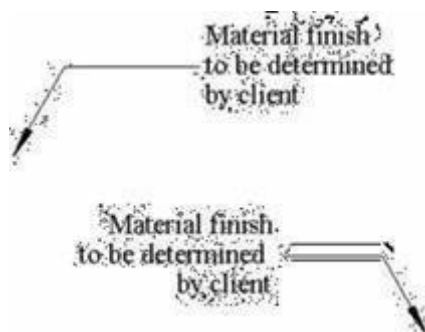


Dimensions - Over First Extension

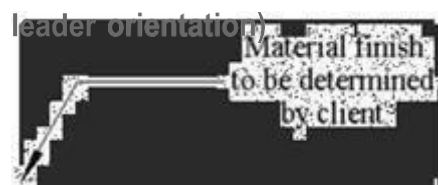
Line



Dimensions - Over Second Extension Line



Leaders - Default Justification (Depends on the



Leaders - Center Justification

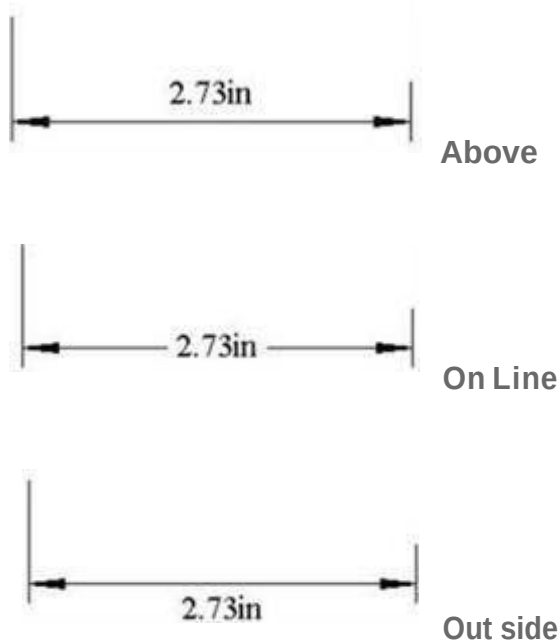
Leaders - Left Justification

Leaders - Right Justification

Vertical: Select the text alignment with respect to the dimension/leader line. **Outside**

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places the text on the side of the dimension/leader line opposite the selected dimension/leader points.



Text Gap: The distance between the dimension text and the dimension line.

Adjustment: Available for **VerticalOnLine** position, The distance of the text above or below the dimension line. A value of zero (default) will place the text on the same level as the dimension line. Higher values move the text above the line; lower (negative) values move the text below the line.

Force Text Horizontal: Makes the dimension text horizontal no matter how the dimension is rotated.

Force Text Inside Extension Line: Keep dimension text between the extension lines, regardless of the distance between the extension lines. If unchecked, the system will decide where to place text.

Non-Rotatable Text

With a dimension (or dimension tool) selected, you can go to the Selection Info palette, and under the Text-Transformation area you can specify that the text be non-rotatable.

When text is set to non-rotatable it will always face the user no matter the orientation of the drawing.

Advanced Format

Options for drawing and scaling the dimension and extension lines.

Dimension Line:

The line over which the dimension text is located.

Force Interior Line: If checked, the dimension will have an interior dimension line even if the text is outside the extension lines. If unchecked, the interior dimension line will be drawn only if the dimension text is inside the extension lines.

Baseline Increment: The distance between each dimension in a series of baseline dimensions.

Do not Draw: Options for omitting parts of the dimension line.

Color: Sets the color of the dimension line.

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Line Width: Sets the Line width for the dimension line.

Line Type: Specifies the line type for the dimension line.

Extension Line:

Lines that connect the dimension line to the object being dimensioned.

Draw as Spline: If checked, the **Leader** tool will use a spline rather than a line to attach the dimension text to the corresponding drawing location. See [Splines and Bezier Curves](#)

Extension: The length of the extension line segments that extend outward beyond the dimension lines.

Offset: The distance between the extension lines and the dimensioned object.

Do not Draw: Options for omitting parts of the extension lines.

Color: Sets the color of the extension lines.

Line Width: Sets the line width for the extension lines.

Line Type: Specifies the line type for the extension lines.

Generate Fixed-Length - Use the Length field to fix the length of the maximum drawn length for extension lines.

Length - Specify the length used by the Generate Fixed-Length property.

Leader Text Frame:

Available only for **Leader** dimensions. Select a shape for the closed line bounding the dimension text.

Size: When **Circle** or **Quadrant** is selected for the text box, define the box size.

Fit to Text: Check if you want the text box to fit the dimension text.

Units - Tolerance

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Parameters for formatting the appearance of dimension text, and for controlling the display of tolerance - the allowable deviation from the dimension.

Primary Units:

Append Units: If checked, the dimension text will display its unit.

Units: If **Append Units** is checked, select the desired units.

Format: Options for how the dimension text is displayed (decimal, feet, radians, surveyor, etc.) As of version 2015 the Format setting can now be set to By Drawing. This means the format used will be the one globally specified on **Options | Space**

Units Prefix, Suffix: Add a prefix and/or a suffix to the dimension text. (Not available for **Leader** dimensions.)

Scale: Change the scale of the value displayed by the dimension text relative to World units. For example, if you enter a value of 0.1, the dimension will display a value of 0.1 inch when dimensioning a distance of 1 inch in World units. The default value for the linear measurement scale is 1.

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Note: You will probably want to leave Scale unchanged, unless you have an inset in your drawing that uses a different scale than the rest of your drawing.

Round Off: -The decimal place to which the dimension text will round off. If you type a value of 0.1, for example, the text will be rounded off to tenths.

Special Symbol: Prepends a symbol to the dimension text. The options include: None, Diameter, Degree, and Plus Minus.

Precision: The level of accuracy, represented by a number between 1 and 10.

Trailing Zeros: is set to 4, and Round Off is set to 4, the dimension text will read 2.3400.

Note: Trailing Zeros, Round Off, and Precision are interrelated, and need to be considered as a group when establishing dimension settings. If Round Off is less than Precision, dimension text may not accurately reflect the exact measurement. If Round Off is greater than Precision, then Trailing Zeros will show the additional available precision if Trailing Zeros is set to a value equal to Precision. For most applications, Precision and Round Off should be set to the same level (a Precision of 4 is the same as a Round Off of 0.0001). Trailing Zeros should then be used if necessary to display the level of precision in use. Trailing Zeros is not applicable when fractions are used.

Leading Zeros: For dimensions of less than one unit, a zero will appear at the beginning of the dimension. For example: 0.5 feet, as opposed to .5 feet.

Zero Feet, Zero Inches: Relevant for architectural and engineering units. For dimensions less than one foot or one inch, the zero will appear as a placeholder. For example: 0'-0 1/4" as opposed to 1/4".

Tolerance:

Append: If checked, the dimension text will include a tolerance.

As Limits: If checked, the dimension will be shown as a pair of values defining the limits of the dimension value.

Relative Height: The height of the tolerance relative to the rest of the dimension text.

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Upper, Lower: Values for positive and negative tolerance. The remaining tolerance parameters are the same as for **PrimaryUnits**.

Alternate Units

Alternate units are a secondary set of units for each dimension, such as 1" [25.4mm].

Use alternate units: If checked, the dimension and/or tolerance will be displayed with a value in alternate units, in square brackets following the primary dimension. The parameters are the same as on the **Units / Tolerance** page.

Dimensions in_through Viewports

(Available in Platinum)

Dimensioning within a viewport in paper space will provide dimensions of the object. The dimensions will be scaled correctly according to the scale of the view. Misleading dimensions can result if the viewport is not exactly parallel to the object. Therefore attempting to dimension an Isometric view by this method is not

advised.

The dimensions will be associative to model and drawing elements through viewports in paper space layouts. These associative dimensions will update with model and drawing changes. This is automatic, but only applies to the following objects:

- Solid objects
- 2D objects
- Surface objects

Dimension Styles

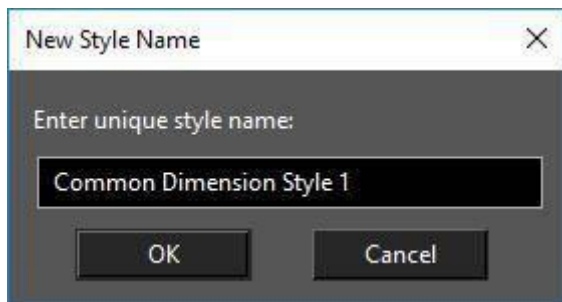
You can manage and create dimension styles using the Style manager. For more on the Style Manager see [The Style Manager Palette](#). In the Style Manager, there is one style, "Standard," listed under "Dimension styles."

Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

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You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New**.

2. Assign a name or accept the default.



This creates a new which is a copy of "Standard."

To change the current wall dimension to the new style, open its **Properties** to the **General** page.

Dimension styles can control all of the properties of your dimensions, They are also automatically imported from DWG files.

Note: While Pasting entity with style, the TurboCAD uses the following merge logic:

Let say the source entity Dimension has a source style called "D-Size"

If the target drawing has no style with the same name - new dimension style is created "D-Size" in the target drawing.

The target drawing has style with the same name - we compare properties of source and target styles. If the styles are equal nothing is done (target style is used).

Otherwise a new style is created with a free default name ("Common dimension style ...") and property values from the source style "D-Size".

Quick Dimensions



Default UI Menu: Dimension/Quick

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Ribbon UI Menu:

Creates a series of dimensions on one more objects. You can choose the type of dimension and which points will be included.

Note: To create a group of various types of dimensions automatically, see [Auto Dimension](#)

Activate the **Quick** function, then choose three criteria:

Whether the dimensions will be orthogonal (horizontal / vertical).

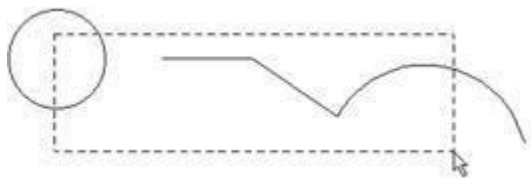
The type of dimension (continuous, baseline, etc.),

The points between which the dimensions will be created.

In this example, the dimensions will be **Orthogonal** and **Continuous** (indicated by the arrows in the picture below). Points will be described later in the example.

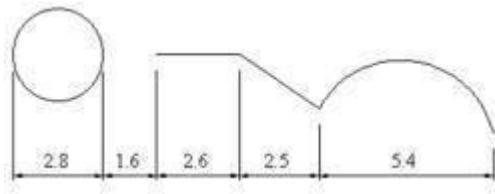


Select the object or objects to dimension. You can use the Shift key to select multiple objects, or drag a selection window. This example consists of one circle and one polyline.



Double-click when all objects are selected. Then move the cursor and click to place the dimensions. Moving the mouse to the right or left of the objects creates vertical dimensions; moving above or below the objects, as shown below, creates horizontal dimensions.

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This would be the result with **Staggered** dimensions:

This would be the results with **Baseline** dimensions.

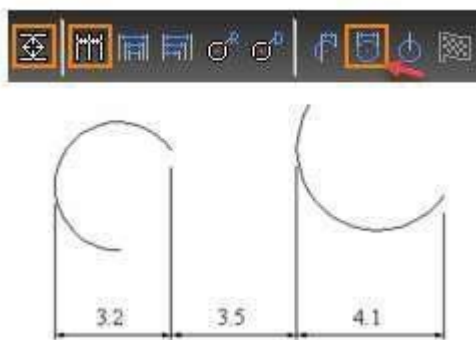
If **Radius** is selected, a radius dimension is assigned to all arc or circle segments found in the selected objects.



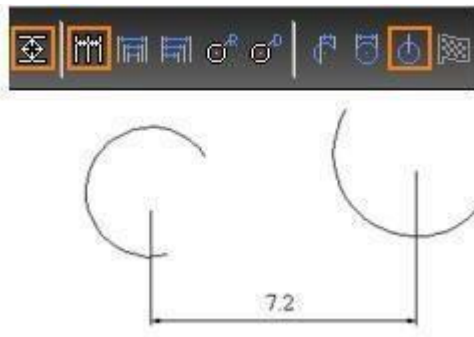
If **Diameter** is selected, a diameter dimension is assigned to all arc or circle segments found in the selected objects.

Points affect dimensioning of arcs, curves, and segments of polylines. **Start / End** creates dimensions between all start and end points of arcs.

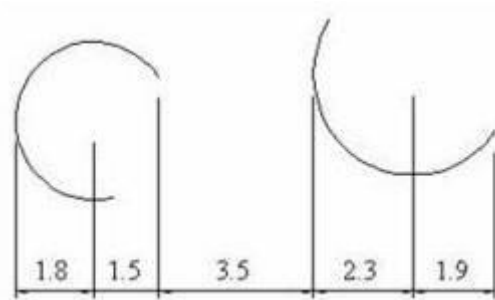
Dimension measures the overall distance of the objects.



Arc Center measures between center points.



You can combine **Points** options. This example uses both **Dimension** and **Arc Center**.



If **Orthogonal** is not selected, you can define the dimension direction. Select two points to define the direction **normal** to the desired dimension line.

This is the result of non-orthogonal, baseline dimensions.

Dimension Groups Only

A local menu option "Group Size" is available. This option allows for dimensioning a Group of graphics. In this way, the user can dimension the maximum size of the group in the X, Y-axis of a group of the graphic.



A local menu option "Dimension Groups Only" is available.

Segment and Entity Dimensioning

For linear dimensions (orthogonal, parallel, and rotated), you can define the dimension by selecting two points (manually), or you can select a segment or entire object to dimension.

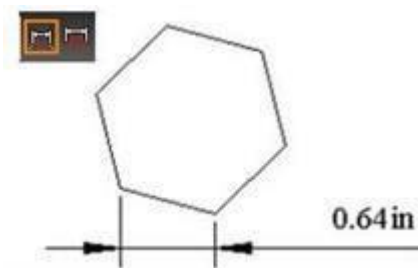
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Segment and **Entity** modes are options on the local menu, as well as on the Inspector Bar while using a linear dimension tool. Once you have selected a dimensioning mode, the mode remains in effect until changed.

Tip: If you generally prefer one of the dimensioning modes over the others, you can save it in a drawing template. To do this, use **File / Save As** to save the file as a **.tct file (TurboCAD Template)**. Place the template file in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use ***File / New**, and select **New from Template**.

In **Segment** mode, select any line segment, and the dimension between endpoints is created.

Segment mode - orthogonal dimension



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Segment mode - parallel dimension

Entity mode - parallel dimension



Smart Dimensions



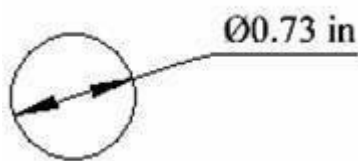
Default UI Menu: Dimension/Smart

Ribbon UI Menu:

Creates a dimension based on the object selected.

Note: To create a group of various types of dimensions automatically, see [Auto Dimension](#)

If you select a circle, a diameter dimension will be created.

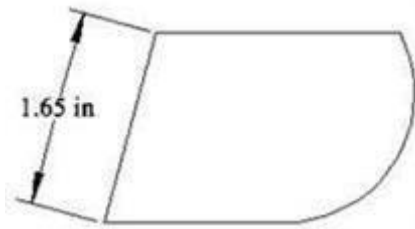


In the Inspector Bar, you can change the text, set the angle of the dimension, and set the length of the leader line.

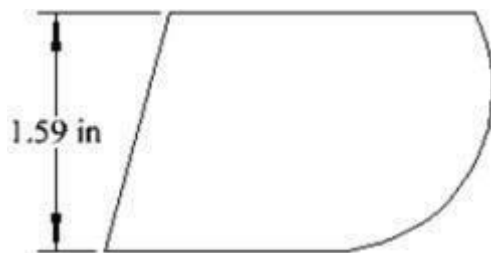


If you select a line or line segment, its length dimension will be created.

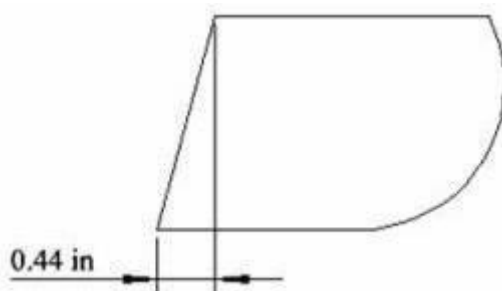
If the line segment is not orthogonal, there are three possible dimensions you can create: **Parallel**



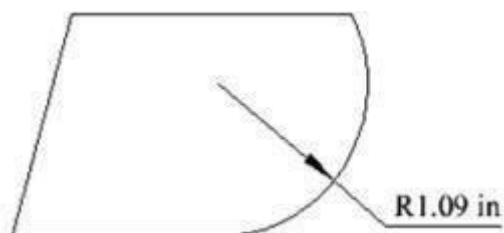
Orthogonal (vertical)



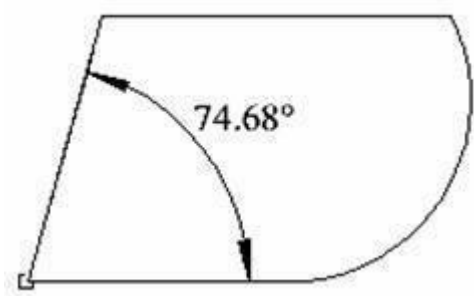
Orthogonal (horizontal)



If you select an arc or arc segment, a radial dimension will be created.



To create an angular dimension, press Shift and select the two lines.



Smart dimensions also work for linear and circular edges of solid ACIS objects. You can use this feature to dimension cross-section generated through the Drafting Palette, as long as the Surface Engine option in the Drafting Palette was turned off when the section was created.

The smart dimension also allow the user to snap to a point when placing the text line.

Linear and angular dimensions can also be created for **3D Polylines**.

Surveyor Dimensions

Linear and Angular dimensions can be displayed in surveyor format.

For linear dimensions, surveyor format is angle (degree-minutes-seconds) and length.

For angles, surveyor format is degree-minutes-seconds.

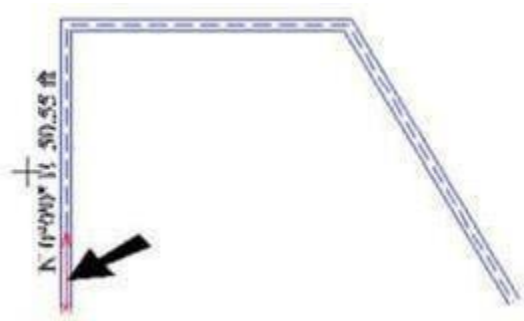
Before creating surveyor dimensions, open the **Properties** for any dimension tool. On

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the **Units / Tolerance** page, set the **Format** to **Surveyor**.

In this example, surveyor dimensions will be placed using **Parallel Dimensions**, on three segments of this **Multi Line Polyline**.

After setting **Surveyor** in the dimension tools' **Properties**, activate the dimension tool you want to use (in this case, **Parallel**, but you can use any linear dimension). Click the segment you want to dimension. (Only segment dimensioning is available when creating surveyor dimensions; you cannot click two points.) The red arrow indicates the direction in which the segment is measured.

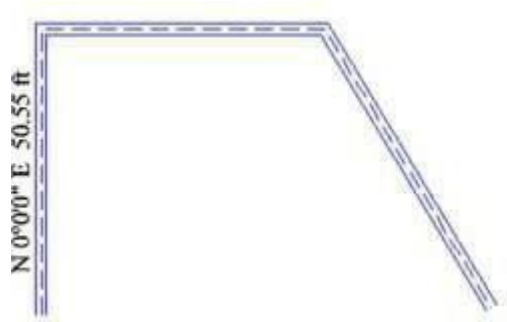


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If you want the segment measured from the other side, select **Invert** from the Inspector Bar or local menu.



Click a second time to place the dimension.



4. Dimension additional segments the same way.

You can dimension an angle using surveyor dimensions as well.

MultiLeader Dimensions

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Default UI Menu: Dimension / MultiLeader

Ribbon UI Menu:

MultiLeader dimension are a special category of dimension tool designed to support entities imported from external DWG files. These leaders attached a single piece of content to multiple leader arrows. the content may be text, a block, or nothing. The content and structure of multileaders is defined by the MultiLeader style.

Local menu option:

Arrowhead First: The first click places an arrowhead. Turning this option on turns off Content First

Content First: The first click places the content. Turning this option on turns of Arrowhead first

Fixed Leader Points: Limit the number of points within each leader to the number specified in the style. The default is 2, one at the arrowhead and one at the content. Once both points have been placed, a new leader will begin for the current multileader. If this option is off leaders may have as many points as you wish.

Insert Next Leader: This option is only available if Fixed Leader Points is turned off. When available the option finishes the current leader at its last selected point and begins the next leader.

Finish: Finishes the entry for all of the leaders.

1. Select the MultiLeader tool.

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Click to select the first point for the leader. If the Content First option is selected this will be the location of the content, otherwise it will be the location of the first

arrowhead.

Click to select the second point for the leader. If the Content First option is selected this will be the location of the first arrow head, otherwise it will be the location of the first content.

If the default settings and style is in use a new leader will begin after the second click.

Click to set the arrow head of the next leader. The content position is already set. So with the standard limit of two points per leader selecting each point sets the

current leader then begins the next.

If you have Fixed Leader Points turned off, continue entering points as needed then use Insert Next Leader to move on to the next leader in the sequence.

Click Finish when you are done creating leaders.

Using the Edit tool you can add and subtract leaders on existing multileaders. you can also add, subtract, and move nodes of multileaders.

MultiLeader Properties

MultiLeader have different properties than standard dimensions.

MultiLeader Format

Overall Scale: Sets the overall scale for the multileader.

Annotative: Set whether the multileader is annotative or not. This has no effect within the application is strictly a setting to support external DWG applications. **Arrowhead:**

Sets the type of arrowhead used in the multileader.

Size: Sets the size of the leader arrowheads.

Leader Type: Sets whether the type is a line, spline, or none.

Color: Sets the color for the leaders.

Line Type: Sets the line type for the leaders.

Line Width: Sets the line width for the leaders.

Landing: Sets whether there will be a landing, a horizontal line attaching the end of leaders to the content.

Landing Distance: Sets the landing width.

Content Type: This is controlled by the Style

MultiLeader Content

The feature of the MultiLeader Properties Content page vary depending upon the content type specified in the selected MultiLeader Style. If the Content Type in the style is set to none the page will be blank. If the Content Type is set to text the page will be

as follows:

Text Style: Set which Text style from the Style manager is used in the multileader.

Height: Sets the height of the text. This will override the settings in the styles.

Frame around text: specifies whether a box will be drawn around the text of the multileader.

Width: Sets the width of the paragraph.

Background Mask: Creates a mask for objects behind the text.

Color: Sets the text color.

Justify: Specifies the text justification.

Left Attachment: Specifies the vertical position where the leaders on the left side of the content.

Right Attachment: Specifies the vertical position where the leaders on the right side of the content.

Text Rotation: Specifies the angle of the content text.

Landing Gap: Specifies the distance between the end of the landing and the content.

St yle: Select **At Least** to automatically set the line size relative to the largest character of a line. Select **Exactly** to keep all lines the same size.

Factor: Sets the distance between lines. If the Content Type in the style is set to block

the Content page will look as follows:

Block Source: Specifies the block to be used in the content.

Block Attachment: Specifies whether the block will be attached to the multileader using its geometric center of the block, or attached using the block reference point (insertion point).

Block Color: Sets the block color.

Block Rotation: specifies the angle for the rotation of the block insertion.

Scale X: Sets a scaling factor for the block along its X axis.

Scale Y: Sets a scaling factor for the block along its Y axis.

Scale Z: Sets a scaling factor for the block along its Z axis.

MultiLeader Styles

You can manage and create MultiLeader styles using the Style manager. For more on the Style Manager see [The Style Manager Palette](#). In the Style Manager, there is one style, "Standard," listed under "MultiLeader styles".

Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

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You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New**.

Assign a name or accept the default.

This creates a new which is a copy of "Standard."

To change the current multileader to the new style, open its **Properties** to the **General** page.

Dimension Formatting Codes

You can insert formatting codes and text into Dimension to control the information that is displayed.

Codes	Definition	Example
<...>	The automatic Dimension text	
\L...I	Turns underline on and off	IMSI \LTurboCADI
~	Inserts a nonbreaking space	IMSI TurboCAD~LTE
\\	Inserts a backslash	IMSI \\TurboCAD
\{...\}	Inserts an opening and closing brace	IMSI \{TurboCAD\}
\Cvalue;	Changes to the specified color	IMSI \C2;TurboCAD
\H value;	Changes to the text	IMSI \ H2;TurboCAD

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	height specified in drawing units	
\H value;	Changes the text height to a multiple of the current text height	IMSI \H3x;TurboCAD
\S...^...;	Stacks the subsequent text at the /, #, or ^ symbol	1.000\S+0.010^-0.000;
\Qangle;	Changes obliquing angle	\Q20;IMSI
\Wvalue;	Changes width factor to produce wide text	\W2;IMSI
\A	Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top)	{\A0; 1}{\A1; 2}{\A2; 3} \SX+A/Y
\P	Ends paragraph	IMSI \PTurboCAD
\X	Creates a paragraph break.	IMSI \X <...>
%%u	Toggles underscoring on and off.	%%uTurboCAD
%%d	Draws degrees symbol (°).	%%dTurboCAD
%%p	Draws plus/minus tolerance symbol (±).	%%pTurboCAD
%%c	Draws circle diameter dimensioning symbol .	%%cTurboCAD
%%%	Draws a single percent sign (%).	%%% 140
%%nnn	Draws character number nnn. NOTE: nnn will only work with ASCII values. If you wish to use Unicode characters of look up an ASCII value. Use the Character Map Utility.	

These codes can be inserted in a dimensions Attribute field, and Info fields.

You can also insert codes into the Prefix or Suffix fields, but these are limited to only a few characters.

7.5. Drawing Symbols

The **AddOns** menu contains several commonly-used symbols you can insert into your drawing.

Most of these tools can also be found on the **Special Tools** toolbar, displayed by right-clicking in any toolbar and selecting **Special Tools**.

Drawing Markers

(Available in Platinum)

Default UI Menu: Architecture/Markers

Ribbon UI Menu:

Use drawing markers to add a shape with text inside. You can choose between a circle, square, diamond, and hexagon.

These tools can also be found on the **Drawing Markers** toolbar.



The marker is inserted with default text (1S, 2S, etc.).



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You can edit the text on the **General** page of the marker's **Properties**. In addition, you can modify properties of either the shape or text by using the Selection Info palette.

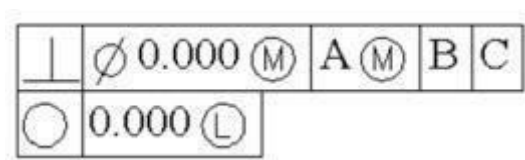
Geometric Tolerance

(Available in Platinum only)

Default UI Menu: Draw/Tolerance

Ribbon UI Menu:

Tolerance is used to specify by how much a dimension can vary. Geometric tolerancing is a way to show maximum allowable deviations of form, profile, orientation, location, and run-out of a feature.



The first of two mandatory symbol frames contains a symbol that represents the geometric characteristic to which a tolerance is being applied, such as form, orientation,

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or run-out. Form tolerances control straightness, flatness, circularity, cylindricity, and profiles of lines and surfaces. The second mandatory frame contains the tolerance value. Where applicable, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. For a tolerance frame with two tolerance values, the second compartments contains the value of Tolerance 1 and is followed by a third, identical compartment that holds the value of Tolerance 2. Other (optional) compartments commonly contain a pair of symbols each. These are a datum reference letter and a material condition symbol.

Select Draw | Tolerance tool.

Click to place the insertion point of the tolerance.

Click to set the angle of the tolerance insertion and to finish.

The **Tolerance** window provides controls for customizing the symbol frames.

Sym: Select one of the available geometric characteristic symbols.

Tolerance (#): Specify three parameters - the tolerance value and two symbols, diameter and material condition, preceding the value and succeeding it, respectively. The diameter symbol can be inserted by toggling the box preceding the tolerance value field.

MC (Material Conditions): Relevant for features that can vary in size

Maximum material condition (M or MMC): A feature contains the maximum amount of material stated in the limits. At MMC, a hole has minimum diameter, whereas a shaft has maximum diameter

Least material condition (L or LMC): A feature contains the minimum amount of material stated in the limits. At LMC, a hole has maximum diameter, whereas a shaft has minimum diameter

Regardless of Feature Size (S or RFS): A feature can be any size within the stated

limits.

Datum (#): A theoretically point, axis, or plane from which you make measurements and verify dimensions. Usually, two or three mutually perpendicular planes perform this task best. These are jointly called the datum reference frame. Specify one of three datum references - primary, secondary, and tertiary (A, B, and C). The datum reference can consist of a value and a modifying symbol

Height Protected Tolerance Zone: Controls the variation in height of the extended portion of a fixed perpendicular part and refines the tolerance to that specified by positional tolerances. Enter a value in the data entry field. A projected tolerance zone symbol (P) can be inserted or discarded by toggling the box succeeding the height entry field.

Use the **Format** page to set the parameters for the lines and text of the Tolerance.

To edit a tolerance symbol, open its **Properties** window (see [Object Properties](#)) and click **Tolerance**.

Surface Roughness



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

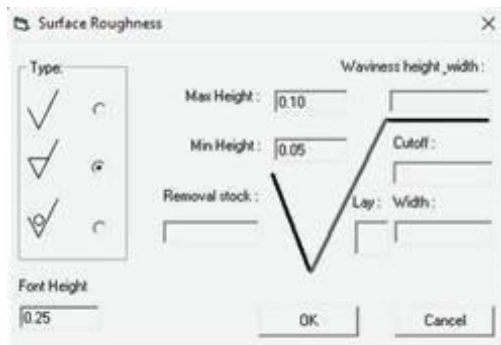
(Available in Platinum)

Default UI Menu: Draw/Surface Roughness

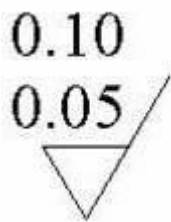
Ribbon UI Menu:

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Use the **Surface Roughness** window to enter the symbol parameters.

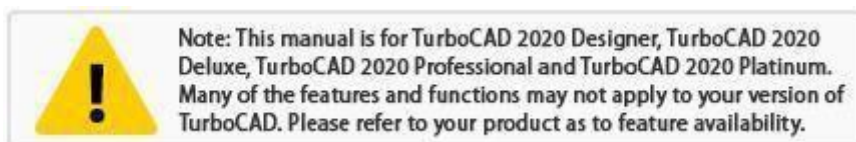


Click OK when finished, and locate the symbol in your drawing.



To edit a surface roughness symbol, open its **Properties** window (see [Object Properties](#)). Open the **Other** page, click **Surface Roughness**, and click **Go To Page**. This opens the original design window, in which you can change any symbol parameters.

Weld Symbols



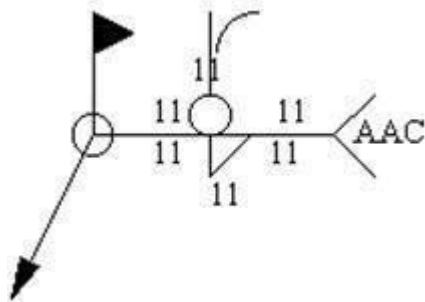
(Available in Platinum)

Default UI Menu: Addons/SDK Samples/Insert/Weld Symbol

Ribbon UI Menu:

Use the **Weld Symbol** window to enter the symbol parameters.

Click OK when finished, and locate the symbol in your drawing.



To edit a weld symbol, open its **Properties** window (see [Object Properties](#)). Open the **Other** page, click **Weld Symbol**, and click **Go To Page**.

This opens the original design window, in which you can change any symbol parameters.

7.6. Hatching

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Hatches

Ribbon UI Menu:

You can display the **Hatch** toolbar by right-clicking in any toolbar area and selecting **Hatch**.

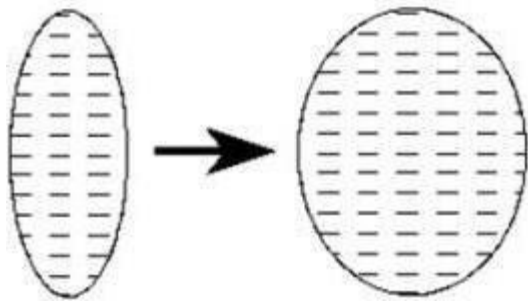


You can also fill a closed object by adjusting the **Brush** settings in the object's **Properties** window. Another way is to use the Colors and Brushes palette; see [Colors and Brushes](#)

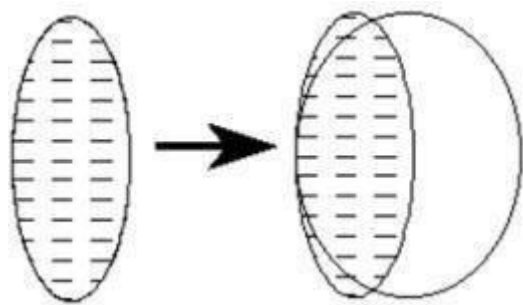
Note: If no pattern is specified when an object is selected for hatching, the fill will be solid.

Associative Hatching

Associative hatches are linked to their boundaries, and update when their boundaries are modified.



Non-associative hatches are independent of their boundaries.



To set hatching to be associative, check **Associative Hatch** on the **Display** page of the **Drawing Setup (Options/Display)**. To break the link between an associative hatch and its boundary, select both the boundary and the hatch, and select **Drop Link** from the Inspector Bar or local menu.



Editing a Hatch Pattern

You can change a hatch pattern by selecting an object and changing the pattern listed on the **Brush** page of its **Properties** window (see [Properties Window](#)).

If you want to edit a hatch pattern, you will first have to explode it (see [Exploding Objects](#)). For associative hatches, you must first explode the object into its boundary and hatch components.

Hatch



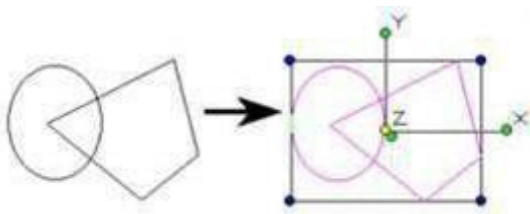
Default UI Menu: Draw/Hatches/Create Hatch

Ribbon UI Menu:

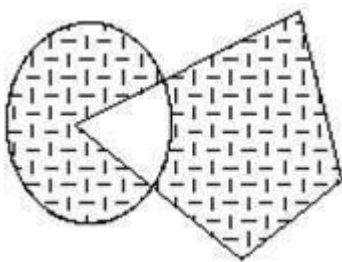
Applies a hatch pattern to a selected closed area. The closed objects must be selected before the **Hatch** tool can be used.

Set the hatch pattern by opening the **Properties** window of the **Hatch** tool. On the **Brush** page, select the pattern and other hatch parameters.

Use the **Select** tool to select the objects you want to hatch.



3. Click Hatch or select Format / Create Hatch.



The hatch is not applied to overlapping areas.

Path Hatching



Default UI Menu: Draw/Hatches/Path Hatching

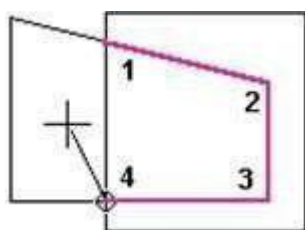
Ribbon UI Menu:

Applies a hatch pattern to a closed boundary of segments, defined manually.

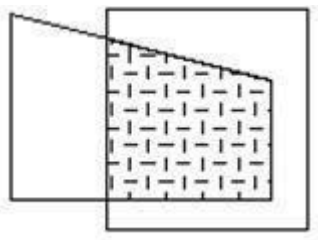
Set the hatch pattern by opening the **Properties** window of the **Hatch** tool. On the **Brush** page, select the pattern and other hatch parameters.

Activate **Path Hatching**.

Select points to define segments of the boundary. The points do not have to lie on existing objects; they can be anywhere. A magenta line indicates the progress of the boundary.



Select the first point again to close the boundary, or select **Close** from the local menu or Inspector Bar.



Local menu option:

By Entity: Use this option to select the entire perimeter of an object. If you have already selected boundary points, **By Entity** will select the perimeter up to the next point. This is especially useful when you are trying to hatch curved objects.

Pick Point and Hatch

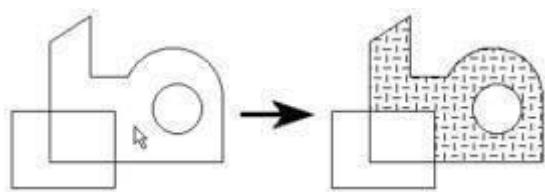


This command has to be activated via the Customize dialog. Applies a hatch pattern to a closed boundary, by clicking one point within the desired boundary. This tool recognizes overlaps and islands.

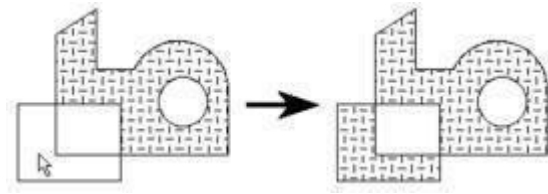
Set the hatch pattern by opening the **Properties** window of the **Hatch** tool. On the **Brush** page, select the pattern and other hatch parameters.

Activate **Pick Point and Hatch**.

Click any point within the desired boundary to create the hatch.



4. To hatch another area, you must activate the tool again and pick another point.



Pick Point Hatching

TurboCAD (All versions)

Default UI Menu: Draw/Hatches/Pick Point Hatching

Ribbon UI Menu:

Pick Point Hatching is an enhanced version of Pick Point and Hatch.



Applies a hatch pattern to a closed boundary, by clicking one point within the desired boundary. This tool recognizes overlaps and islands.

With Pick Point Hatching you can pre-set the hatch's pattern Line Color, Line Pattern, Line Width, Brush Color and Brush Pattern from the Properties toolbar directly above the drawing area.

Activate **Pick Point Hatching**

Click any point within the desired boundary to create the hatch.

You can hatch additional areas by clicking until you cancel out of the tool.

Pick point hatching is now working in internal areas. Pick point Hatch now also works when there are gaps between boundary areas.

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New “Gaps” field is added in the Inspector Bar.

This will work with gaps in 2D objects.(For example,trying to fill an area bounded by a Polyline, when there are gaps in the Polyline)

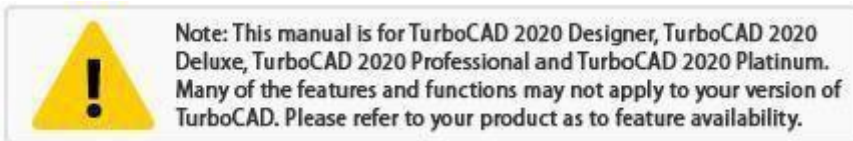
It will also apply to ADT objects (Walls, Doors and Windows), allowing to similarly fill in gaps between these objects.

Demo Video:

<https://youtu.be/QhzXqSTMxYc>

7.7. Text Along Curve

Text Along Curve



(Available in Platinum only)

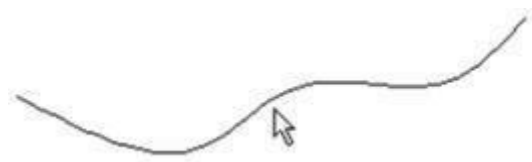
Default UI Menu: Draw/TextAlongCurve

Ribbon UI Menu:

Note: You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

Creates text that follows a curve.

Select the curve, picking the center of the desired text.



In the **Text Properties** window, type the text string and set other properties such as font, height, and thickness. The **Text Location** field enables you to place the height above, below, or centered on the curve.

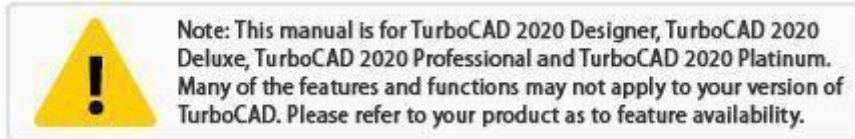
Note: For definitions of other properties, see Text Properties .

3. Click OK, and the text is created.



To edit curved text, open its **Properties** window (see [Object Properties](#)). Open the **Other** page, click **Text Properties**, and click **Go To Page**. This opens the **Text Properties** window, in which you can change any parameters, or the text string itself. If the text was created upside-down, click **FlipText** to right it.

7.8. Entity Marks



(Available in Platinum and Professional)

Default UI Menu: Insert/Entity Mark

Ribbon UI Menu:

Create custom marks to track, or count objects, and add information using Property Sets and Multi-blocks. Entity marks can be composed of one or both of two elements:

Property Set Definitions - Create custom information for objects, and derive custom information from object geometry.

Multi-View Blocks - Define composites of blocks that can vary their appearance depending upon viewpoint.

Demo Video:

<https://www.youtube.com/watch?v=QXCzSOGcW0g>

Property Set Definitions

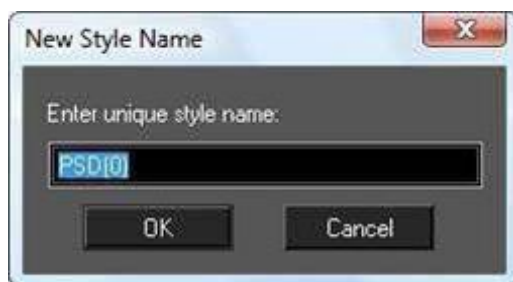
Property Set Definitions (PSD) are used to define what kind of objects will associate with Entity Marks and assign the type and nature of data to be used with Entity Marks.

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Property Set Definitions are assigned in the Styles Manager palette.

Select Property Set Definitions in the Style Manager.

Click the New button at the top of the palette.



When the New Style Name dialog appears enter a name then click OK.

Select the new Property Set Definition in the Style Manager.



In the Data area of the Style Manager (at the bottom), select Property definitions, then click on the data button that appears.

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6. The Property Set Definition dialog will open.

Select the objects to which the PSD will apply, and then proceed with creating the property definitions for the set, either Automatic, or Manual, or Both. Note that the only objects which will work for Entity Marks are those associated with Property Definition Sets.

Automatic Definitions

Automatic definitions for PSD's are use to extract data directly from the object types.

Select a single property from the property list. A name reflecting the property will automatically be assigned.



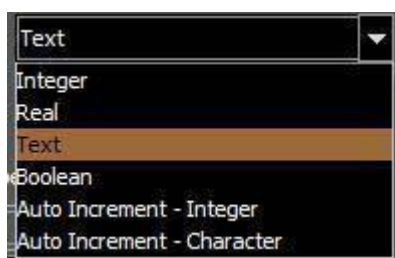
Modify the name if needed.

Click Ok create the property definition.

Manual Definitions

Automatic definitions for PSD's are use to extract data directly from the object types.

Specify a Type for the property definition. Options include: **Integer** - a set number in integer format e.g. 5, 20, 3 **Real** - a set number in real format e.g. 1.5, 2.645, 3.14159 **Text** - a string of text e.g. Made by Troponopo inc. **Boolean** - a True/False value. Allows you to add a checkbox property e.g. (property name=SecurityGlass) True **Auto Increment-Integer** - an automatically incrementing integer value e.g. 0, 1, 2 **Auto Increment-Character** - an automatically incrementing character value e.g. A, B, C



Specify a default value for the property definition.

Create a name for the property definition.

Click Ok create the Property definition.

Auto Increments

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Auto Increment in manual property definitions (Auto Increment - Integer, Auto Increment - Character) are use to create sequential markers for object in your drawing. The increment value will depend upon the order of the objects creation, not the order in which you create the Entity Marks.

Assigning Property Sets to Objects



Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

In order for Entity Marks to work with your objects you must assign the Property Sets with individual objects.

Select all the objects of a specific type (Line, Circle etc.) to which you wish to assign property definition sets. Only select a single object type at a time.

Open the Selection Info palette, expand the General area, and then click Property Sets.

Click the Data button adjacent to Property Sets to open the Add/Remove Property Sets dialog.

A list of PSD's will appear, but only those which apply to the selected object type.

Select the desired PSD's, and then click OK.

Entity Marks and Attributes



Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Block Attributes can be used to include object data with Entity marks. When you create the Block Attribute, type the name (tag) of the attribute in the following format:

`<Name_of_PSD><:>Name_of_Property_Definition>`

For example, if the PSD is called 'LineData' and the name of the Property Definition is 'Length' the Tag would be LineData:Length. When the attribute appears in the inserted mark on a line the line's length will be shown in the mark. The values shown in the displayed attribute will update if the associated object is modified.

Inserting Entity Marks

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Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Once the Property Set Definitions, Multi view Block and Entity Mark Definition are all created and Property sets have been assigned to objects you can proceed to insert Entity Marks.

Select the Entity Mark tool

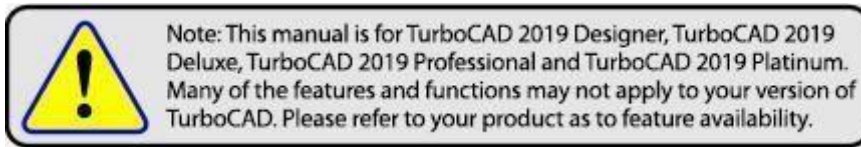
Click on the relevant object.

Move the cursor and click to place the Entity Mark.

In the example above the Entity Mark show the Number of the Line, and the Length of the Line.

Multi-View Blocks

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Multi-View Blocks (MVB) are used to create the visual elements of Entity Marks. MVBs are composed of pre-created blocks and block attributes. Multi-View Blocks are assigned in the Styles Manager palette.

Select Entity Marks Definitions in the Style Manager.

Click the New button at the top of the palette.



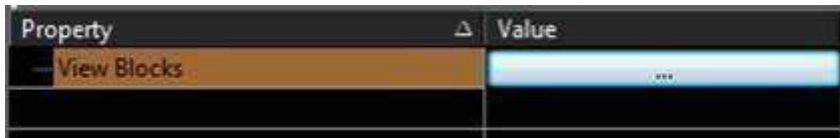
When the New Style Name dialog appears enter a name then click OK.

Select the new Entity Mark Definition in the Style Manager.

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In the Data area of the Style Manager (at the bottom), select View Blocks, then click on the data button that appears.



The Multi-View Block Definition dialog will open.

7. Click the Add button to add a new block to the MVB.

8. Select a block from the list of blocks, and then click OK.

Under View Directions select which directions will apply. The views are relative to the World Coordinate System. The Other directions is any view not expressly Top, Bottom, Front etc. View Directions do not re-orient blocks. If the block is flat on the world plane it will remain so even if you set it to a Left direction. Orient your blocks as you want them to appear.

Add as many blocks as you want to the MVB then click OK.

7.9. Center Mark

(Available in Platinum only)

The center mark tool is designed to create an associative center mark entity on circles and arcs.

Default UI Menu: Draw/Point/Centre Mark

Ribbon UI Menu:

To create a center mark:

- Start the tool.

- Click on an arc or circle.

If you move the associated arc or circle the center mark will move with the entity. However, if you move the mark separately you will disassociate it from its related entity. You can create a center mark that is not associated by simply clicking anywhere in the drawing space, except on a circle or arc.

Properties

Show Extensions: Sets whether the extension lines are shown or not.

Cross size: Set the size of the center mark cross. The cross size, by default, is designated as a multiple of X, where X is the radius of the associated circle/arc. For example 0.5x would be a center mark of 1/2 the radius.

Gap size: Sets the size of the gap between the cross and the extension lines. The gap size, by default, is designated as a multiple of X, where X is the radius of the associated circle/arc. For example 0.2x would be a center mark of 1/5 the radius.

Left: Sets the distance the Left extension line will go beyond the radius of the associated arc/circle.

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Right: Sets the distance the Right extension line will go beyond the radius of the associated arc/circle.

Top: Sets the distance the Top extension line will go beyond the radius of the associated arc/circle.

Bottom: Sets the distance the Bottom extension line will go beyond the radius of the associated arc/circle.

Rotation: Sets the rotation angle of the center mark.

7.10. Center Line

(Available in Platinum only)

The center mark tool is designed to create an associative center line entity between lines or line segments.

Default UI Menu: Draw/Point/Centre Line

Ribbon UI Menu:

To create a center Line:

- Start the tool.

- Click on an line or line segment.

- Click on another line or line segment.

If you move an associated line or line segment the center line will move with the entity. However, if you move the center line separately you will disassociate it from its related entity.

Properties

Show Extensions: Sets whether the extensions of the center line is shown or not.

Start Extension: Sets the distance the start of the center line will go beyond the average length of the two of the associated lines or line segments.

End Extension: Sets the distance the end the end of the center line will go beyond the average length of the two of the associated lines or line segments.

7.11. Map

(Plug in - Available for Platinum, Professional and Deluxe)

The map utility allows you to insert map objects, pulled from Google into TurboCAD. First select the tool, the Map dialog will open.

Options

Drawing Units: Specifies the Drawing Units that will be used in the drawing. by default, these will be the units from the drawing you are inserting into. If the units are changed the unit in the drawing will be changed.

Latitude: Show the latitude of the center of the map you are viewing.

Longitude: Show the longitude of the center of the map you are viewing.

Search: Search for a location

Map/Satellite: Select whether a Map or Satellite view will be shown. **Labels/Terrain:**

Map and Satellite each have sub-options: terrain and Labels, respectively. Terrain shows the topology of the ground, and Labels adds names and locations to the image.

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Cross-hairs: these cross-hairs indicate the center of the map.



Show center of map: This option toggle the cross-hairs visibility.

Update drawing geolocation: This forces the drawing geolocation to change to the geolocation to the map. The drawing geolocation is always the 0,0,0,(origin) point in the drawing.

Create terrain: Derives terrain information from the map and attaches it to the inserted map. This option will grey-out is the selected region is too large.

Use current drawing geolocation: This forces the map geolocation to be inserted relative to the geolocation to the drawing.

Select custom region: When toggled on you are prompted to select a region of the visible map to be used.

Place a map: Places the map into the drawing.

Cancel: Cancels and closes the Map dialog.

Properties

Map t ype: Show the type of map that was inserted

GPS coordinates: The GPS location of the center of the map

Show center of map: This option toggle the cross-hairs visibility.

Data provider: Source of the information

Entity t ype: This has two option which control what the inserted map will be: Plain - a map image, Terrain - a TurboCAD terrain object derived from the map data.

Example Terrain

When in Terrain form the map has additional properties see [Terrain](#) for more details.

7.12. Spelling Tool and Spell Options

(Available in all TurboCAD Variants)

Default UI Menu: Tools/Spelling , Tools/Spell Options

Ribbon UI Menu:

Spelling tool and spell options both exist in Tools menu. They can also be accessed from standard tool bar. They work for both text and multi text.

Spelling tool:



It lets you proofread and edit the text in your drawing.

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Select the text you want to proofread. Make sure you have set the desired spell options according to which you want to proofread the text in your drawing. For example

Select the text and select spelling from tools menu.

The Spelling tool will show possible correction for the first misspelled word.

There are options like ignore or replace for quick editing of the selected text.

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A warning dialogue appears to notify that the spelling check is complete.

Spell options:



It lets you set the options according to which you want to proofread the text in your drawing. You can choose from several options which you want to include or exclude for example if you want the "spelling" tool to ignore capitalized words then you can enable option "Ignore Full Caps".

8. Working in 3D

This section focuses on setting up your drawing to work in 3D, creating 3D objects, and manipulating the view so that you can see your objects from any angle.

You can use 3D tools (as well as most 3D editing tools) in both wireframe and render modes. See [Creating a Rendered View](#)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

8.1. 3D Coordinate Systems

(Available in Platinum, Professional and Deluxe)

Each model has an internal, fixed coordinate system - the WCS, or World Coordinate System. The WCS cannot be changed, but you can define your own coordinate system the UCS, or User Coordinate System. When you start a new drawing, the UCS by default coincides with WCS. All objects are defined and manipulated with respect to the current UCS.

Selecting an object using **2D Selector** mode (see [2D-3D Selector](#)) changes the UCS (and therefore the workplane) to the one attached to the selected object.

When you move an object in **3D Selector** mode, or an object that is part of a group or block, the UCS and workplane do not change. The Entity Coordinate System and its workplane travel with the object.

The UCS indicator is helpful if you want to see how the UCS moves. You can display markers for the UCS and WCS via View | Display | UCS symbol menu, and customize the markers' appearance via the **Preference** page of the **Program Setup (Options / Preference)**. The WCS indicator is always shown at the lower left corner of the screen; the UCS indicator is shown at the UCS origin.

8.2. Workplanes

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/New UCS

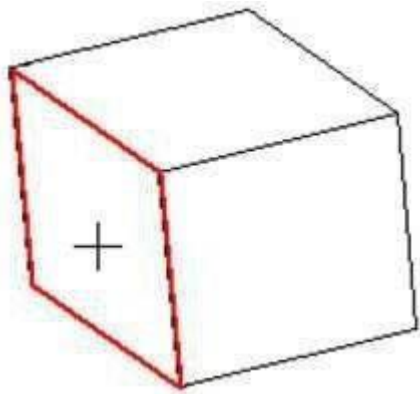
Ribbon UI Menu:

A workplane is the plane on which 2D objects are created within the the current coordinate system (UCS or WCS), and on which most 3D objects are based. In 2D, you always work on the same workplane - the XY plane of the current UCS. In 3D, however, the workplane may change frequently.

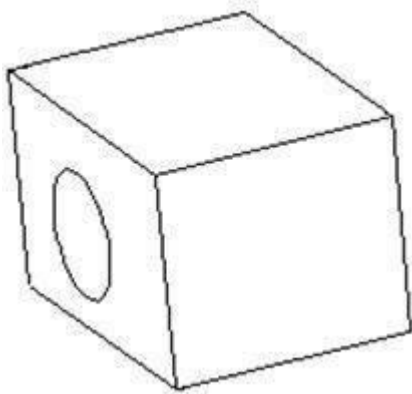
In 3D, the workplane is still in the XY plane by default. But if you have 3D objects in your model, you can temporarily move the workplane to the facet where your cursor is. This is set in the **Drawing Aids** window, which can be opened by right-clicking on the SNAP or GEO field at the bottom of the screen.

Note: For details on **Drawing Aids**, see [Snap Settings](#)

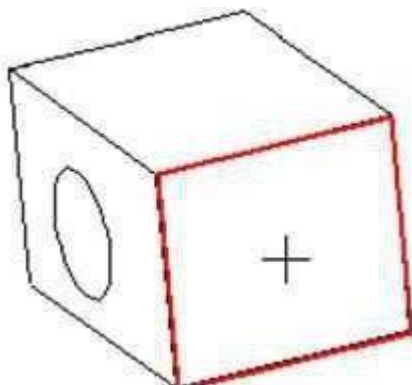
When Auto Workplane by Face* is checked, the facet will highlight in red when the cursor passes over it.

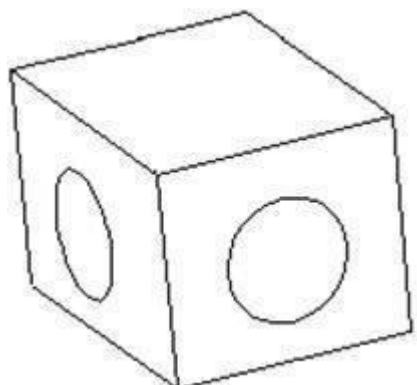


The object you draw will be placed on that facet.



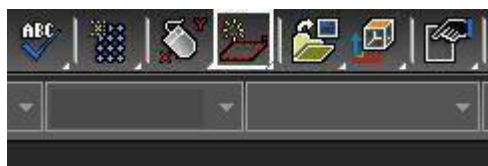
The facet workplane is temporary; after the object is created, the workplane returns to the XY plane. If you highlight another facet, that workplane will be temporarily active for the next object you create.





Note: If you create an object with no facet highlighted, the default workplane will be used.

If you uncheck **Auto Workplane by Face** in **Drawing Aids**, the workplane will always be the default workplane. If you don't want to use the XY plane of the current UCS, you can use one of the **Workplane** commands. These commands can be found on the flyout toolbar on the main toolbar.



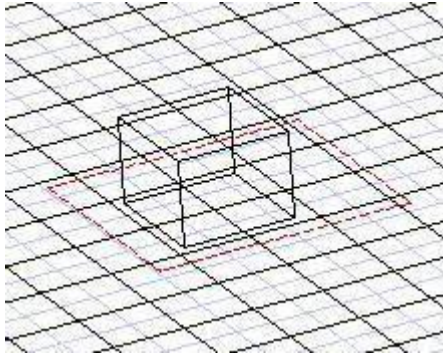
You can also display the **Workplane** toolbar by right-clicking in any toolbar area and selecting **Workplane**.



Note: The Design Director also contains tools for manipulating workplanes, creating workplanes, and displaying the view according to a selected workplane. See [Design Director-Workplanes](#)

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If you are working with the grid turned on (see [Grid](#)), the grid will lie on the current workplane.



Changing the Workplane

While working in 3D, you will probably have to change the workplane frequently to create objects that have the correct location and orientation.

For example, create a cube based on the default workplane. If you want to create a cylinder on the top face of the cube, you must move the workplane up to this face. Then you can place the cylinder in the correct location.

The workplane can also change when you select objects, depending on whether the 2D or 3D Selector is used. See [2D-3D Selector](#)

Workplane by View



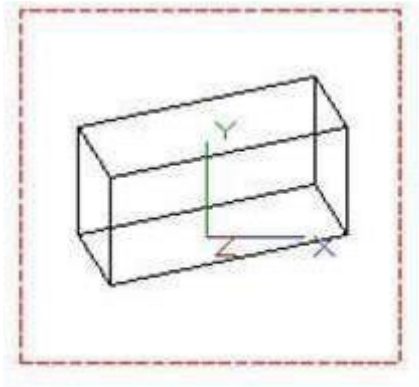
Default UI Menu: Tools/New UCS/By View

Ribbon UI Menu:

Sets the workplane according to the current view.

Position the model to the desired view. In this example, the current workplane is **By World**.

Select **By View**. The workplane is turned so that it is parallel to the view plane. The Z axis becomes perpendicular to the drawing screen.



Workplane by World



Default UI Menu: Tools/New UCS/By World

Ribbon UI Menu:

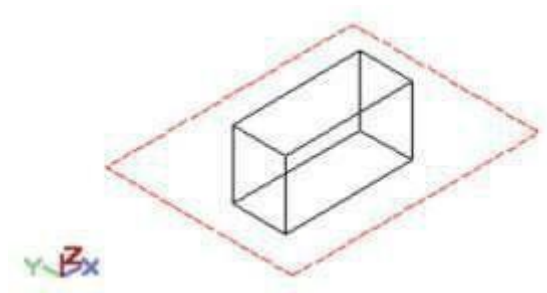
Sets the workplane based on the WCS (World Coordinate System).

Note: By World is the default workplane when selecting objects that reside on different workplanes.

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You can display the WCS at the lower left corner of the screen by opening the **Preferences** and checking **Show World CS** .

By World, top view



By World, Isometric_SW view

Workplane by Entity



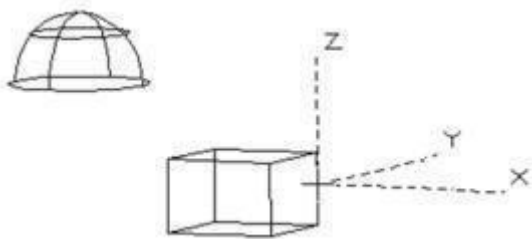
Default UI Menu: Tools/New UCS/By Entity

Ribbon UI Menu:

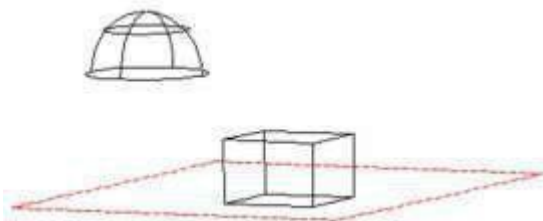
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Sets the workplane according to the coordinate system of a selected object, block, or group.

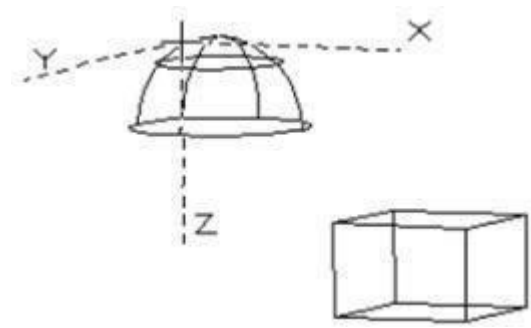
Click **By Entity**, and the cursor becomes a dashed representation of the current UCS. Click the desired object.



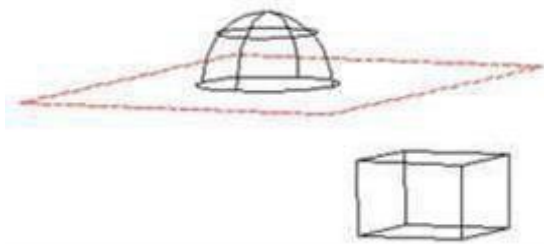
The workplane is placed along the plane on which the object was created.



To change the workplane to another object, use the tool again. Click the desired object.



The workplane is positioned according to the selected object.



Tip: If an object is listed in the Graphics section of the Design Director, you can set the view to an object's **Workplane by Entity**, and set this workplane as the current workplane. See [Design Director-Graphics](#)

Workplane by 3 Points



Default UI Menu: Tools/New UCS/By 3 Points

Ribbon UI Menu:

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Sets the workplane to fit three points. You can select these points, or enter their coordinates in the Coordinate Fields.

1. Select the UCS origin.

2. Select the second point to define the +X direction.

3. Select any point that lies on the desired X-Y plane.

The workplane is created.

Workplane by Z axis



Default UI Menu: Tools/New UCS/By Z Axis

Ribbon UI Menu:

Sets the workplane by defining the Z axis of the UCS. The X-Y plane is perpendicular to this axis.

Select the UCS origin, or enter the coordinates in the Coordinate Fields. In this example, the workplane is to be aligned with face shown.

2. Select the second point to define the +Z direction.

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The workplane is created.

In this example, because of the view, it may not appear that the workplane was aligned correctly. You can rotate the model to verify that the workplane is indeed aligned with the desired plane.

Workplane Origin

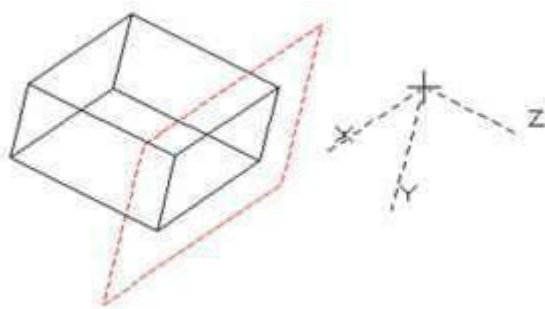


Default UI Menu: Tools/New UCS/Origin

Ribbon UI Menu:

Sets the workplane parallel to (offset from) the current workplane, by relocating the UCS origin.

When you select UCS Origin, the cursor becomes a set of dashed axes, with the X-Y plane parallel to the current X-Y workplane.



2. Select a new UCS origin, or enter the coordinates in the Coordinate Fields.

The parallel workplane is created.

Workplane by Facet



Default UI Menu: Tools/New UCS/By Facet

Ribbon UI Menu:



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

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Sets the workplane to be aligned along a facet of a solid.

Note: The workplane is set to a facet automatically by default, as long as **Workplane by Face mode** is checked in the **Drawing Aids** window. You can open this window by right-clicking on the SNAP or GEO field at the bottom of the screen. See [Snap Settings](#)

Move the cursor to the desired facet, which is highlighted in red, and the workplane axes are displayed.

Note: To select a facet behind or in front of the indicated facet, use the Page Up and Page Down keys.

2. Click to create the workplane.

Displaying the Workplane

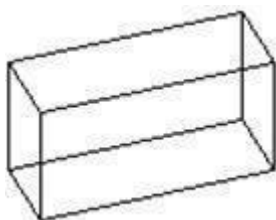


Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/New UCS/Display WorkPlane

Ribbon UI Menu:

Displays or hides the red indicator representing the current workplane.



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WorkPlane hidden

WorkPlane displayed

Note: If **Workplane by Face mode** is checked in the **Drawing Aids** window, the location of the workplane will change temporarily when you highlight a facet and create an object on it. As soon as the object on that facet workplane is finished, the workplane returns to the default location.

Initially the indicator is sized to fit the entire window, but you can change this, as well as the angle and location of the workplane. You can also display the origin of the workplane coordinate system by opening the **Preferences** (Options / Preference) and checking **Show User CS**.

Fit to Window

Default UI Menu: Tools/New UCS/Fit To Window

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Ribbon UI Menu:

Fits the workplane rectangle to the current window. This is useful after a view change such as zoom or pan.

Display Intersections with 3D Objects



Default UI Menu: View/Display/UCS/Show 3D Intersections

Ribbon UI Menu:

Enables you to visualize where the current workplane intersects with 3D objects. This is a toggle command; if selected, the interactions will always be displayed whenever the workplane is displayed.

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Note: You can snap to points along intersection lines and curves. See [Snap Modes](#)

Workplane without interactions

Workplane with interactions

Editing the Workplane



Default UI Menu: Tools/New UCS/Edit

Ribbon UI Menu:

Enables you to move or rotate the workplane, and to scale the workplane indicator.

Drag the blue handles to change the scale (only affects the indicator).

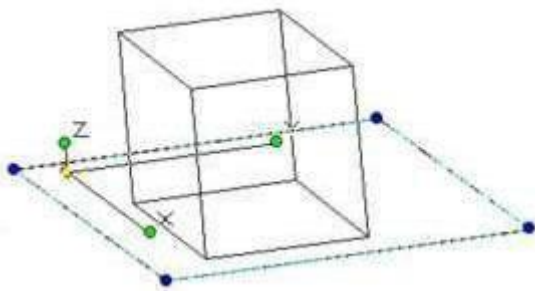
Use the green handles to rotate the workplane about a specific axis.

Use the yellow reference point to move the workplane.

Press D to select the workplane's reference point if you want to move it.

You can also use the scale, position, and rotation fields in the InspectorBar.

1. Click **Edit**, and the workplane is displayed with editing handles.



Click on a green rotation handle (Y in this case), and move the mouse to rotate the workplane.

3. Click anywhere outside the workplane to finish and accept the new position.

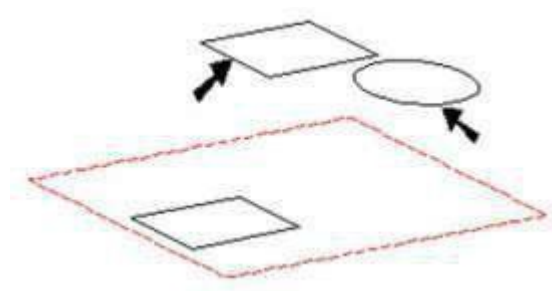
Place on WorkPlane

Default UI Menu: Modify/Place on Workplane

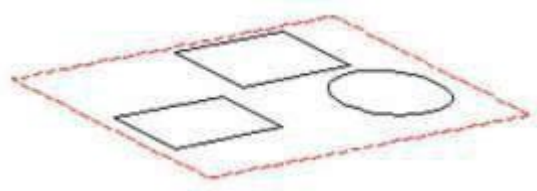
Ribbon UI Menu:

Enables you to place 2D objects on the current workplane.

1. Select the objects you want to move.



Select **Modify / Place on Workplane**. The objects are moved onto the current workplane.



This tool can be useful, for example, for objects that have been selected with the **3D Selector** (see [2D-3D Selector](#)) and moved from their original workplane. If you try using a 2D tool like **Fillet** on two lines that sit on different workplanes, the tool will not work; one of the lines must be moved to the workplane of the other.

Flatten on Workplane

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Flatten

Ribbon UI Menu:

Flatten tool is used for making 3D polyline and spline flat on the work plane.

Select the 3D polyline or spline by using selection tool.

Select Flatten tool from the Modify menu.

Demo Video:

<https://youtu.be/QzcYmiV44-E>

Saving and Recalling WorkPlanes



Default UI Menu: Tools/New UCS/Named

Ribbon UI Menu:

Saves workplanes so that you can recall them later.

Note: You can also create saved workplanes in the Design Director. See [Design Director-Workplanes](#)

1. Create the desired workplane.

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Click the **Set Named Workplane**, and assign a name for the workplane. Click **New Workplane** to add it to the list.

3. Type In a name for the workplane. Then click **OK**.



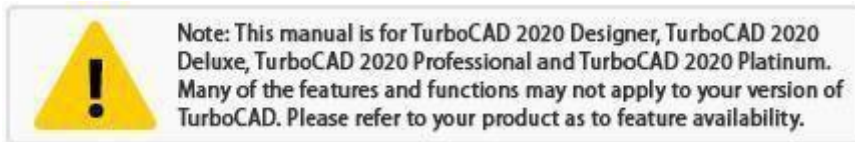
Now, when a different workplane is active, you can recall the one you saved. Click **Set Named Workplane** again, find the desired workplane on the list, and tick mark it in the **active** column.. (You can also double-click on the name.)

Previous



Recalls the previous workplane. Using this tool repeatedly will toggle between the current and previous workplanes; it will not scroll back through several workplanes.

8.3. 3D Views



(Available in Platinum, Professional and Deluxe)

While working in 3D, there are many ways to change how you view your drawing. Views are seen by the "camera" - equivalent to your eye. The camera is located in a specific place and faces a specific direction

There are several standard orthographic and isometric views, and if you need additional perspective you can change how the camera is oriented. You can move the camera using the **Move Camera** commands, and dynamically examine the model using the **Walk Through** tools. To save and display additional views, you can create camera objects.

Tip: If you use a wheel mouse, keep the middle mouse button pressed, and drag the cursor around the screen to dynamically rotate the model.

Saving 3D Views

You can save 3D views using the methods described in [Saving Views](#). In addition, you can use camera objects to save views with specific parameters and open them in their own windows.

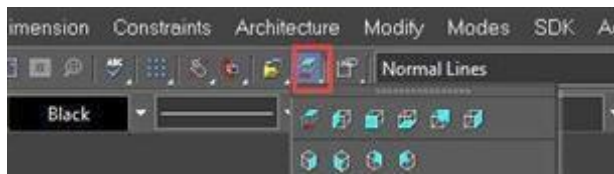
Standard Views

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Default UI Menu: View/3D Views

Ribbon UI Menu:

The ten standard views can be accessed from the **View / Camera** menu.



You can also display this toolbar separately by right-clicking in the toolbar area and selecting **Standard Views**.



And, by default, these tools are available on the local (right-click) menu.

Orthographic Views

The first six views on the toolbar are orthographic views - Top (World Plan), Left, Front, Bottom, Back, Right.

If your model was enclosed in a box, these views reflect how the model would appear by looking directly at each of the box faces. If you "unfold" the box, you can see the relationship between the views.

Front View: Looks at the XZ plane of the World Coordinate System, while facing +Y.

Top View: Looks at the XY plane while facing -Z (down).

Tip: The **Top** view can be also obtained from the **Plan** view by using the **Roll** walk through tool.

Left View: Looks at the YZ plane while facing +X.

Bottom View: Looks at the XY plane while facing +Z (up).

Back View: Looks at the XZ plane while facing -Y.

Right View: Looks at the YZ plane while facing -X.

Isometric Views

The last four views on the toolbar are isometric - angled views. The views are named for compass directions (Isometric_SE is the view looking from the southeast).

8.4. Examining the 3D Model

Once the model contains one or more 3D objects, there are several ways to change the way you see the model.

Camera Movements

(Available in Platinum, Professional and Deluxe)

Default UI Menu: View/Camera/Camera Turn, View/Camera/Camera Slide

Ribbon UI Menu:

Incremental controls for adjusting the camera, and therefore your view of the model. As the camera moves through space, the visible objects will seem to move in the opposite direction of the camera motion. These commands can be accessed via the **Camera** toolbar. You can also display the **Camera Tools** toolbar by right-clicking in any toolbar area and selecting **Camera Tools**.



Camera Turn Up: Turns the camera up.

Camera Turn Down: Turns the camera down.

Camera Turn Right: Turns the camera to the right.

Camera Turn Left: Turns the camera to the left.

Camera Rotate Right: Rolls the camera clockwise.

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Camera Rotate Left: Rolls the camera counter-clockwise.

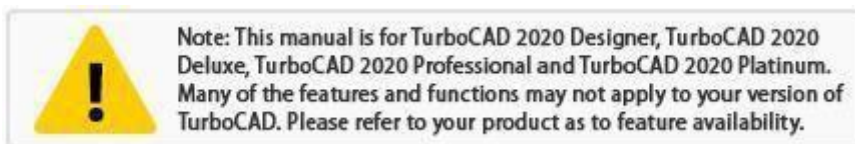
Camera Up: Moves the camera up.

Camera Down: Moves the camera down.

Camera Right: Moves the camera to the right.

Camera Left: Moves the camera to the left.

Camera Objects



(Available in Platinum, Professional and Deluxe)

Default UI Menu: View/Camera/Insert Camera

Ribbon UI Menu:

Camera objects are used to create and save views with specific parameters. The view for each camera object can be displayed in a separate window. Each new camera is assigned a name by default (Camera 1, Camera 2, etc.) but this name can be changed. You can display the **Camera** toolbar by right-clicking in any toolbar area and selecting **Camera**.



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The difference between camera objects and saved views (see [Saving Views](#)) is that saved views do not open in separate windows, and do not include any additional parameters, such as render mode.

Note: You can also create and manipulate camera objects, and group cameras into camera sets, via the Design Director. See [Design Director-Cameras](#)

Camera Object by View

Inserts the camera at the center of the view, so that the camera view is the same as the current model view. The properties of this camera are those of the default camera representing the current view, described in the **Camera** page of the **Camera Properties**. The camera object itself is not visible in the current window, and a separate view window will not be created, since you are already looking at the camera view.

Camera Object Normal to View



Inserts the camera perpendicular to the current model view. The camera will face the same way as **Camera by View**, but can be placed anywhere.

Camera Object Parallel to View



Inserts the camera on a plane parallel to the current model view. The first point places the camera, and the second point defines its direction.

Camera By 2 Points



Inserts the camera by picking two points. The first point places the camera and the second point defines its direction.

Camera Object Views

To display the view seen by the camera object, you can create a separate view window. To create a view window, click the



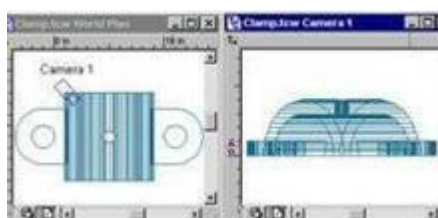
icon for the relevant camera. The icon is used to show or hide the camera indicator. You can also use this dialog to edit and delete camera objects.

Select: Selects the highlighted camera in the drawing. This option is useful when the drawing is large, and you are unable to find the camera in the screen.

Show All: Makes all camera objects visible.

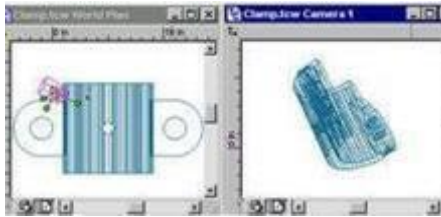
Hide All: Hide all camera objects.

To display all view windows, select **Window / Cascade** or **Window / Tile**.



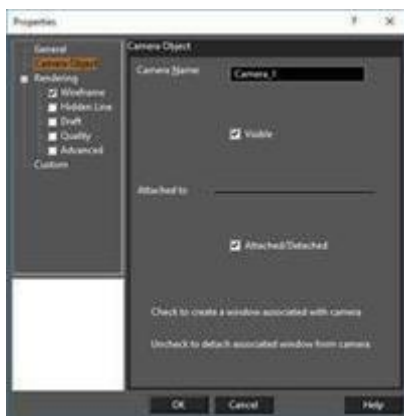
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If you change the direction of a camera in one window, the views in any attached windows will update accordingly. If you use other view controls in an attached window, the camera position will be changed.



Camera Object Properties

Double-click a camera object.



Visible - Check this box to display the camera object indicator in the view window.

Attached / Detached - Check this box to create a new drawing window containing the camera view. Updates to the camera will update the view. Unchecking the box will disconnect the window from the view, so that updates to the camera will not affect the view.

Camera Sets

A camera set is a group of cameras, which can be handy if you want to set rendering, perspective, and visibility parameters for multiple cameras. Camera sets are created and manipulated in the Design Director. See [Design Director-Cameras](#)

QuickTime Movies

(Available in Platinum, Professional and Deluxe)

Default UI Menu: View/Movies

Ribbon UI Menu:

If the model is rendered by LightWorks, you can export the scene to a QuickTime movie format. This format does not create an actual animated movie, rather it exports to a format in which you can rotate and view objects from different angles. The QuickTime player can be downloaded for free from www.apple.com. There are two types of QuickTime movies: **Panoramic**: Constructed from a series of images taken from a fixed viewpoint. You can rotate around the viewpoint and look in all directions. **Object**: Constructed from a series of images taken from different angles. You can rotate the objects in any direction, to view all sides. Before you can create a movie, the scene must be in **Perspective** mode, which is set in the **Camera Properties (View / Camera / Properties)**.

Also, the LightWorks rendering must be one of the following modes: **Flat** (Draft rendering) **Gouraud** (Draft rendering) **Phong** (Draft rendering) **Preview** (Quality rendering) **Full** (Quality rendering) See [Render \(Camera\) Properties](#)

Creating a Movie

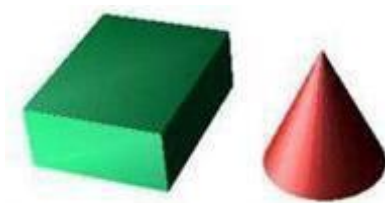
Panoramic QT Movie



Create Object QT Movie



Create the 3D model and view it in **Perspective** mode.



Create either the Panoramic or Object movie. You are asked for a file name, and you are notified when the movie is created.

Play the movie using the QuickTime Viewer. With either movie type, you can pan or rotate the scene by holding the left mouse button and moving the mouse.



Movie Setup



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Opens the **QuickTime Movie Setup** window.

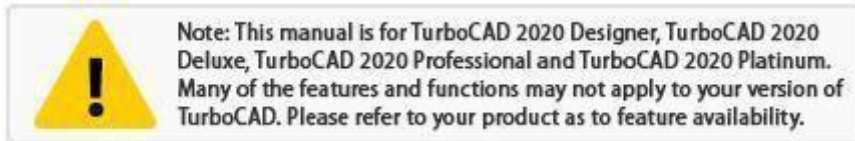


To display the lower section of the window, click the **More** button. **Movie Copyright** and **General Movie Info**: Identifying information about the movie. **Object Movie properties** **Pan Min**, **Pan Max**: Limits of horizontal pan position, in degrees. **Initial Pan**: The starting horizontal angle, in degrees, for when the movie is first opened. **Pan Frames**: The number of frames to generate between the **Pan Min** and **Pan Max** positions. With the default value of 36, frames are rendered every 10 degrees. **Tilt Min**, **Tilt Max**: Limits of vertical pan position, in degrees. **Initial Tilt**: The starting vertical tilt angle, in degrees, for when the movie is first opened. **Tilt Frames**: The number of frames to generate between the **Tilt Min** and **Tilt Max** positions. With the default value of 19, frames are rendered every 10 degrees. **Fixed Lights**: As the camera moves around an object, the lights will stay fixed relative to the camera. This gives the effect that the camera is static and the object is rotating, which may look more natural in some cases.

Note: Object movie files contain a large number of frames, and can therefore take a long time to create. With the default values of 36 pan frames and 19 tilt frames, 684 frames will be created.

Panoramic Movie properties **Initial Zoom**: Defined by a value between 0 and 100. **Slit Number**: The number of individual images that should be rendered. The default value should be used for all but advanced users; setting this value too low could result in distortion, and setting it too high can result in minimal quality gain. **Width Scale**: Movie width will be set as follows: $\text{Movie width} = (\text{Width Scale}) * (\text{Width} / 2) - \text{Width}$. When you increase the movie width, be aware that rendering time and movie file size also increases. **Width, Height**: The dimensions of the movie viewing window.

The Camera



(Available in Platinum, Professional and Deluxe)

Default UI Menu: View/Camera/Properties

Ribbon UI Menu:

In 3D space, you can view a model from any point. TurboCAD uses the concept of a "camera" - the camera represents your eye as you view the current scene. You can easily use the standard orthogonal and isometric views, but camera tools enable you to view from any angle. The current model view contains a default camera, placed at the center of the screen, facing into the model.

Note: While "camera" means the perspective of the current view, "camera object" is something different. Camera objects are created to save and display multiple views, or views with specific parameters.

Camera Properties

Controls the position and target of the camera, as well as rendering controls. For the rendering properties, see [Render \(Camera\) Properties](#)

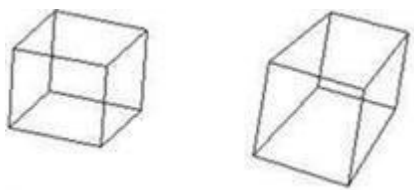
Camera Centered: Enables the camera to turn around its own center when you use the **Camera Turn** tools . When not checked, the camera will turn around the

Target Point. **Scroll Speed:** The speed of the camera movement.

Perspective View: The visual effect of a perspective projection is similar to that of photographic systems and of the human visual system. The size of the perspective projection of an object varies inversely with the distance of that object from the center of projection.

Note: When using **Perspective View**, objects cannot be located or moved beyond the horizon. Select **Display Horizon** to display the horizon, in order to better orient yourself to the space.

View Angle: The degree of perspective foreshortening. The view gets wider as the setting gets larger.



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Standard view and perspective view Position: The physical location of the camera.

Drawing Center: Assigns the target point coordinates as the drawing center.

The drawing center is always the geometric center of all the objects in the drawing.

Note: You can also access this view by selecting **View / Camera / Look to Drawing Center**.

Target Point: The point that the camera faces.

Drawing Center: Moves the camera to the current drawing center.

The UP Vector: Indicates which way is up for the camera. A positive Z value rotates the camera up; a negative Z value rotates the camera down.

Graphic in Front of Camera: Select the object closest to the desired view center, and the target point is attached to that object.

Note: You can also access this view by selecting **View / Camera / Look to in Front Graphic**.

Auto Targeting: Sets the center of rotation on the object closest to the center of the screen (the default camera position). This is relevant when moving objects using the keyboard and mouse (such as Shift + right mouse button).

Walk Through Tools

(Available in Platinum, Professional and Deluxe)

Default UI Menu: View/Walk

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Ribbon UI Menu:

Dynamic (moving) controls for moving the camera. These commands can be accessed through the **Walk Through** toolbar. You can display the **Walk Through** toolbar by right-clicking in any toolbar area and selecting **Walk Through**.



Examine

Ctrl + Right mouse button

Views the model from different angles. Drag the mouse to the right to see the left side of the object. Drag downward to see the top of the object.

Slide

Ctrl + Shift + Right mouse button Moves the camera along a plane parallel to the screen. You can move the camera up, down, left or right, but not forward or backward. Drag the mouse straight up to move the viewpoint upward, drag to the left to move it left. If you drag the mouse along an angle, the viewpoint will move along the corresponding angle. It's important to remember that the camera will move in the same direction as the cursor, which means that the Model Space will appear to move in the opposite direction.

Roll

Shift + Right mouse button or Ctrl with the arrow keys. Rotates the camera around the axis that passes through the camera. This tool will rotate the viewpoint either clockwise or counterclockwise. Drag the mouse to the left to rotate the viewpoint clockwise; drag

to the right to rotate counterclockwise. Vertical movement of the mouse will behave identically to the **Walk** control.

Walk

Simulates walking around on the current plane. This is a handy way to walk through a 3D house model, for example. Positions the camera on the current horizontal plane, also known as the viewplane. Moving the mouse forward or backward moves the viewpoint in the corresponding direction. Left or right movement turns the camera in the corresponding direction. To move the viewpoint forward, drag the cursor toward the top of the screen. To move backward, drag toward the bottom of the screen. If you drag diagonally upward toward the left or right, the viewpoint will move forward at an angle; if you drag diagonally downward to the left or right, the viewpoint will move backward at an angle.

Note: This tool will move forward or backward only when the **Perspective View** option is checked on the **Camera** page of the **Camera Properties**.

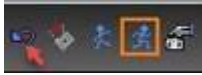
Turn

Rotates the camera around its center. This changes the angle of the viewpoint without relocating the camera. The viewpoint angle can be changed upward or downward. The viewpoint angle can also be changed to the left or right. To turn the viewpoint right, drag the mouse to the right of the screen. To turn downward, drag toward the bottom of the screen. As with **Slide**, the viewpoint will move in the same direction as the cursor. This will have the effect of moving the view of the model on the screen in the opposite direction of the cursor motion.

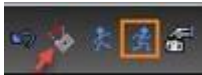
Local Menu Options

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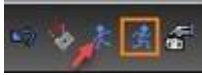
While using the **Walk Through** controls, there are several options available on the local menu and Inspector Bar. **Undo Move:** Returns the view to the position in which the current mode was invoked.



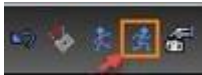
Straighten: Returns the view to a level orientation.



Discrete Movement: Limits camera motion. When on, the camera will only move when the mouse is moving.



Continuous Movement: Allows the camera to move as long as the left mouse button is held down. The camera movement will be in the direction the mouse was dragged. This is the default option.

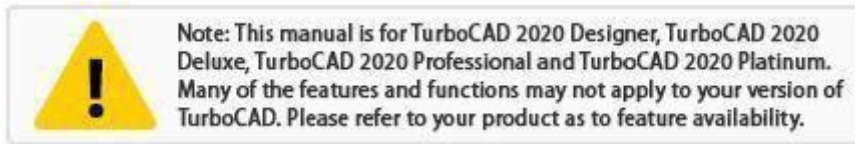


Camera properties: Opens the **Camera Properties** window.



8.5. Dynamic Cut Plane

Dynamic Cut Plane



(Available only in Platinum)

Default UI Menu: View/Dynamic Cut Plane

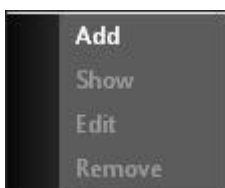
Ribbon UI Menu:

The Dynamic Cut Plane allows you to interactively create a cross section view of 3D objects.

The Dynamic Cut Plane menu is accessed via the View menu.

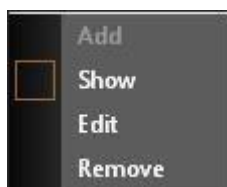
To Use the Dynamic Cutplane tool.

Go to View | Dynamic Cut Plane| Add



This will make the Cut Plane appear.

Go to View | Dynamic Cut Plane and make sure that Show is selected/highlighted.



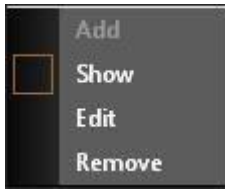
Go to View | Dynamic Cut Plane| Edit

Use the Selector to drag the Cutplane graphic laterally away from the object you want to view. This will give you a clear view of the object.

You can now use the selector to move and rotate the Cut Plane to generate dynamic cross-sections.

To Remove a Cutplane

1. Go to View | Dynamic Cut Plane.



2. Click Remove.

Options

You can manage the appearance of the Dynamic Cut Plane by going to **Options | Drawing Setup Dynamic Cut Plane** .

Cut plane

Active: If there is an active Cut Plane this is checked. If this is unchecked the active workplane will be removed.

Show: Sets whether the active Cut Plane is visible.

Cut section color: Sets the color of objects where they are cut.

Graphic

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Color: Sets the color of the cut plane.

Transparency: Sets the transparency level (0 - 1). 0 = No transparency, 1 = Totally transparent.

Show border: Sets whether the border of the cut plane is shown.

Auto-update in accordance with drawing extents: Sets the extent of the cutting plane to update as the drawing extents changes.

9. Creating 3D Objects

There are three types of 3D objects you can create:

Standard 3D objects: Objects created entirely within as 3D tool. See [Standard 3D Objects](#)

3D Profile objects: Objects created by performing functions on 2D profile objects. See [Profile Objects and Face to Face Modifications](#)

Modified 2D objects. 2D objects that are assigned a thickness. See [Creating 3D Objects by Editing 2D Objects](#)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

9.1. Standard 3D Objects

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Primitives

Ribbon UI Menu:

Standard 3D Objects, are created entirely within a 3D tool, with no reference to any other objects.

Box

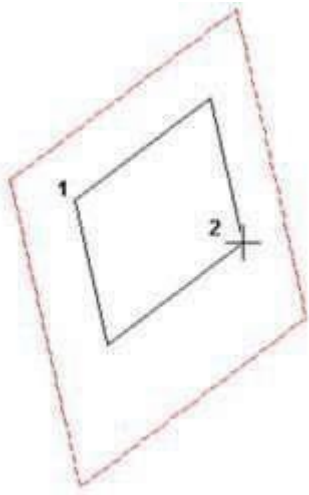


Default UI Menu: Draw/3D Object/3D Primitives/Box

Ribbon UI Menu:

Creates an orthogonal, rectangular prism.

Select two opposite corners for the rectangular base. The rectangle is created on the current workplane.



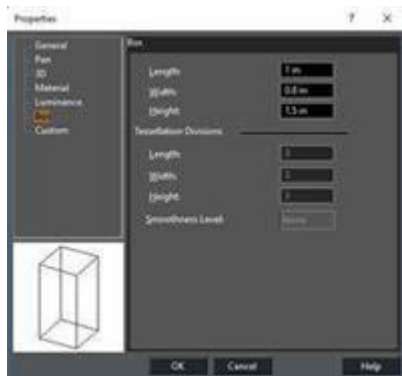
2. Select the third point to define the box height. The height is normal to the workplane.

You can also enter the length, width, and height in the Inspector Bar.

Box Properties

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The **Properties** window of a box contains a **Box** page, in which you can set the length, width, and height.



Cone

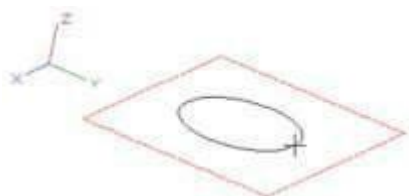


Default UI Menu: Draw/3D Object/3D Primitives/Cone

Ribbon UI Menu:

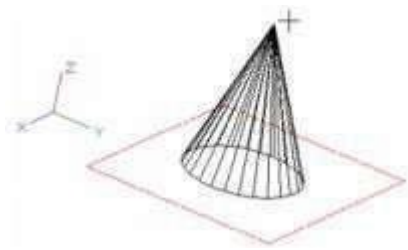
Creates a cone - by default, an object with a circular base that tapers to a point at the top. There are local menu options for creating non-standard cones.

Create the circular base by selecting the centerpoint and a point on the circumference. Or enter the radius, diameter, or circumference in the Inspector Bar. The circle is created on the current workplane.



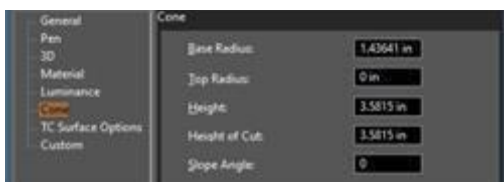
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Select a third point to specify the cone height, or enter the height in the Inspector Bar. You can create the cone on either side of the workplane. The top point is directly above the base centerpoint.



Cone Properties

The **Properties** window of a cone contains a **Cone** page, in which you can set the geometric parameters.



For **TC Surface Options**, see [TC Surface Properties](#)

Note: These options can be used separately or in combination. **Cut Cone byPlane** and **Cut Cone by Lofting** cannot be used together.

Local menu options:

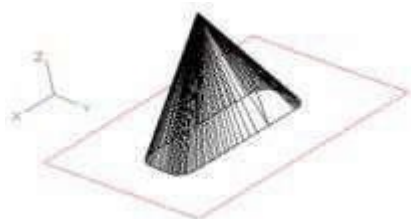
Specify 2D Base: Uses an existing 2D object as the base.

Select the 2D object. (You cannot use 2D objects that are contained within a group or block.)

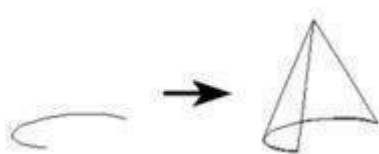


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Select a point to define the height, or enter the height in the Inspector Bar. The height is perpendicular to the workplane of the 2D object.



You can also select a 2D open object. In this case, a partial-cone surface object will be created.



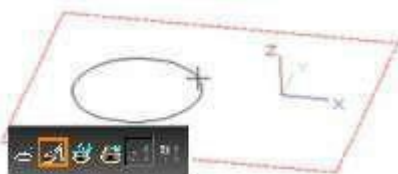
You can also use a compound profile, open or closed, for the 2D base. (A compound profile is a series of connected lines and/or arcs.) Make sure **Specify 2D Base** and **Use Compound Profile** are both active, and select the compound profile.



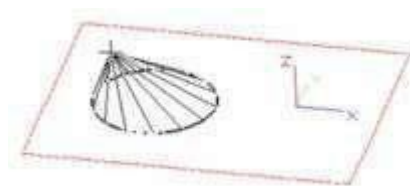
When the profile is selected, select **Finish Selection of Profile**.



Specify Object Height Base: Creates an offset cone by using a specified point for the cone tip.



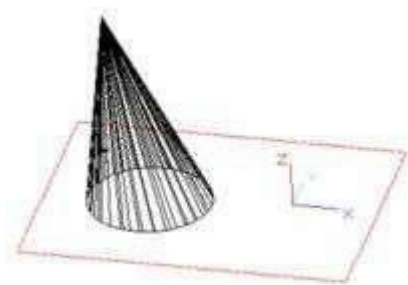
3. Create the circular base.



Select the point which is directly under the desired tip.

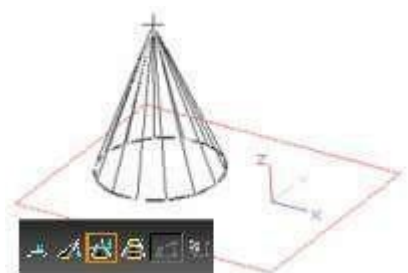
Select a third point to specify the cone height, or enter the height in the Inspector Bar.

The top point is directly above the selected point.



Cut Cone by Plane: Creates a truncated cone by cutting off the tip.

1. Create a standard cone.



Move the mouse back toward the base to create the cut. You can also enter the height of cut in the Inspector Bar.



Cut Cone as Lofting: Creates an inverted, truncated cone by using a scaling factor for the cone base.

1. Create a standard cone.



Move the mouse outward to create the top of the cone. You can also enter a scaling factor in the Inspector Bar.



Cylinder

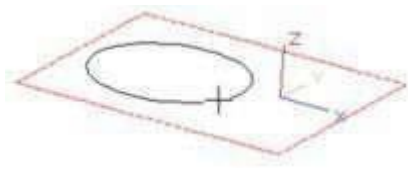


Default UI Menu: Draw/3D Object/3D Primitives/Cylinder

Ribbon UI Menu:

Creates a cylinder - by default, a circular prism.

Create the circular base by selecting the centerpoint and a point on the circumference. Or enter the radius, diameter, or circumference in the Inspector Bar. The circle is created on the current workplane.

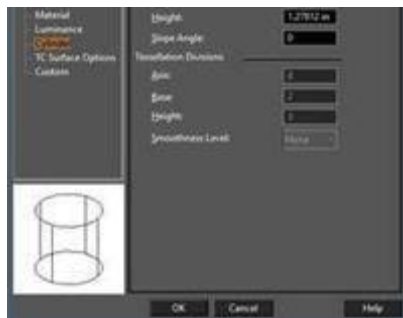


Select a third point to specify the cylinder height, or enter the height in the Inspector Bar. You can create the cylinder on either side of the workplane.



Cylinder Properties

The **Properties** window of a cylinder contains a **Cylinder** page, in which you can set the geometric parameters.

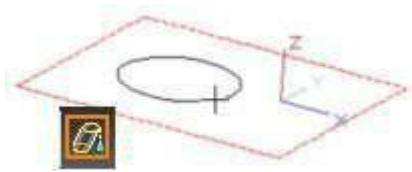


For TC Surface Options, see [TC Surface Properties](#)

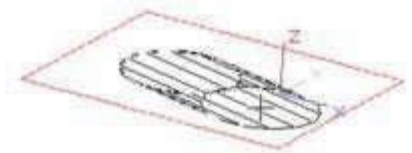
Local menu option:

Specify Object Height Base: Creates an offset cylinder by using a specified point for the top face.

1. Create the circular base.



Select the point which is directly under the desired center of the top face.



Select a third point to specify the cylinder height, or enter the height in the Inspector Bar.



Hemisphere



Default UI Menu: Draw/3D Object/3D Primitives/Hemisphere

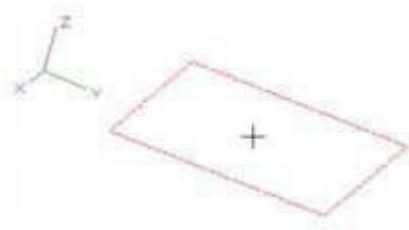
Ribbon UI Menu:

Creates a hemisphere (half of a sphere) with its circular base on the current workplane.

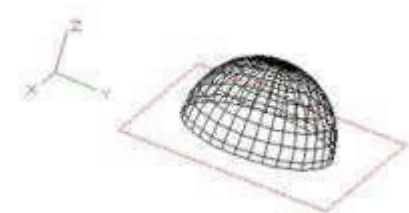
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Note: A hemisphere is considered to be a **Sphere** object, and its **Properties** contain a **Sphere** page.

Select the center point of the hemisphere. The point will be created on the current workplane.



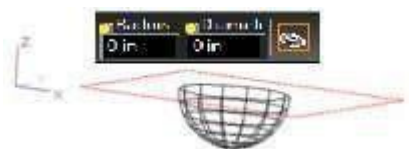
2. Click to size the hemisphere, or enter the radius in the Inspector Bar.



The hemisphere is created on the side of the positive Z axis.

Local menu option:

Downward: Creates a hemisphere in the negative Z direction, "below" the workplane.



Polygonal Prism

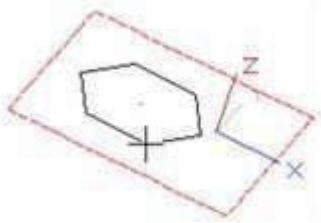


Default UI Menu: Draw/3D Object/3D Primitives/Polygonal Prism

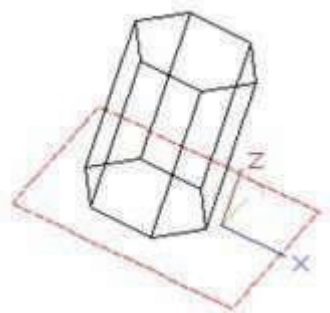
Ribbon UI Menu:

Creates a prism by defining a polygon (multi-sided, equal lengths, closed) as its base shape.

Create the base polygon by selecting the centerpoint and one of the vertices. You can enter the number of sides, angle, and radius or side length in the Inspector Bar. The polygon is created on the current workplane.



Select a third point to specify the prism height, or enter the height in the Inspector Bar. You can create the prism on either side of the workplane.



Prism Properties

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The **Properties** window of a prism contains a **Prism** page, in which you can set the geometric parameters.



For **TC Surface Options**, see [TC Surface Properties](#)

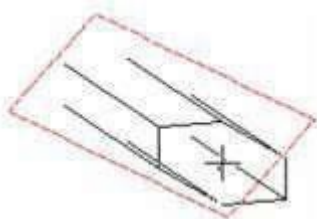
Local menu options:

SpecifyObject Height Base: Creates an offset prism by using a specified point for the top face.

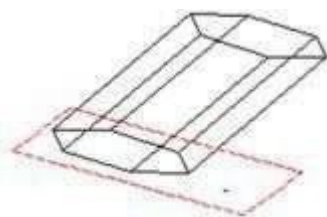
1. Create the base polygon.



2. Select the point which is directly under the desired center of the top face.



Select a third point to specify the prism height, or enter the height in the Inspector Bar.



Rotated Box

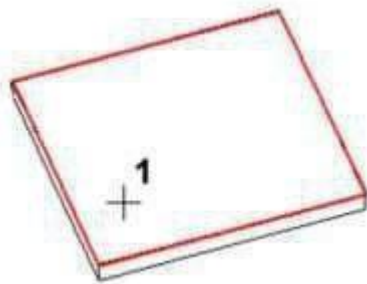


Default UI Menu: Draw/3D Object/3D Primitives/Rotated Box

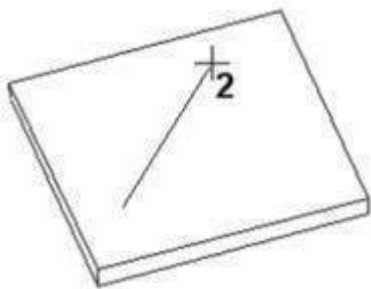
Ribbon UI Menu:

Creates a rectangular prism, based on a rectangle defined by three points.

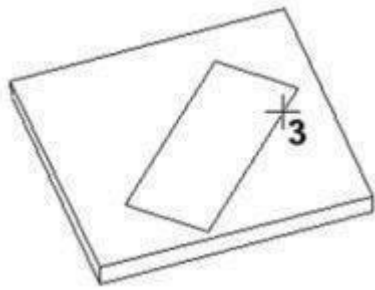
The rotated box will be placed on top of this shallow box. The first click defines one corner of the base rectangle.



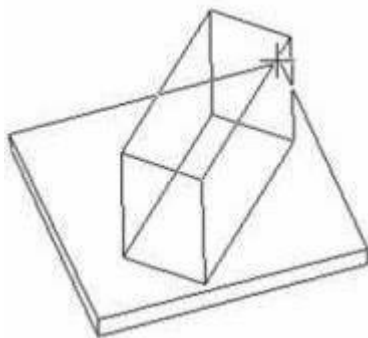
2. The second click defines one edge of the base rectangle.



3. The third click defines the width of the base rectangle.



4. The fourth and final click defines the height of the box.



You can also enter the length, width, and angle of the edge of the base rectangle, and the height, in the Inspector Bar. The **Properties** window of a rotated box contains a **Box** page, in which you can set the length, width, and height.

Sphere

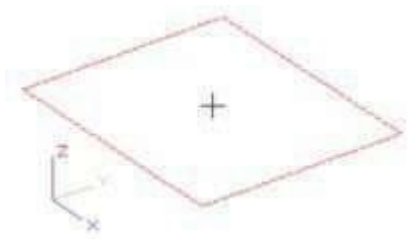


Default UI Menu: Draw/3D Object/3D Primitives/Sphere

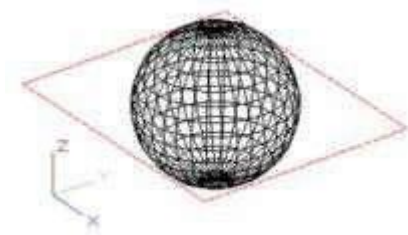
Ribbon UI Menu:

Creates a sphere centered on the current workplane.

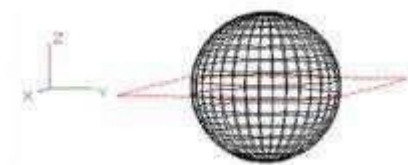
Select the center point of the sphere. The point will be projected onto the current workplane.



2. Click to size the sphere, or enter the radius in the Inspector Bar.

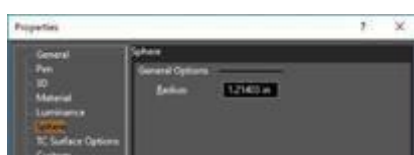


The sphere is created, centered on the workplane.



Sphere Properties

The **Properties** window of a sphere (and hemisphere) contains a **Sphere** page, in which you can set the radius.



For TC Surface Options, see [TC Surface Properties](#)

Torus

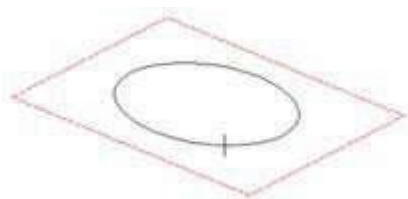


Default UI Menu: Draw/3D Object/3D Primitives/Torus

Ribbon UI Menu:

Creates a donut-shaped object by extruding a circle along a circular path.

Create the circular base by selecting the centerpoint and a point on the circumference. Or enter the base radius in the Inspector Bar. The circle is created on the current workplane.



The default option is **Center**, which means the cross-section of the torus uses the base circle as its center. Click to define the tube size, or enter the tube radius in the Inspector Bar.



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The torus is created, centered on the workplane.



Torus Properties

The **Properties** window of a torus contains a **Torus** page, in which you can set the geometric parameters.



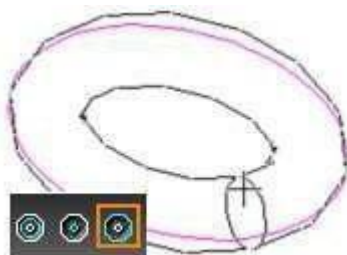
For TC Surface Options, see [TC Surface Properties](#)

Local menu options:

Inner Radius: The base circle is the inner radius; the torus proceeds outward.



Outer Radius: The base circle is the outer radius; the torus proceeds inward.



Wedge

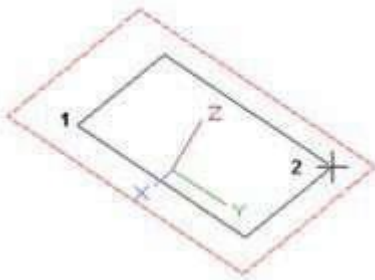


Default UI Menu: Draw/3D Object/3D Primitives/Wedge

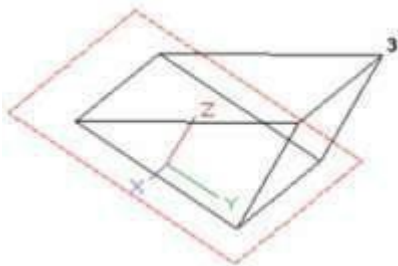
Ribbon UI Menu:

Creates a triangular wedge - a box cut diagonally in half.

Create the base rectangle by selecting its opposite corners. You can also enter the length and width in the Inspector Bar. The rectangle is created on the current workplane. The order of selecting points is important - the wedge will be extruded from the second point.



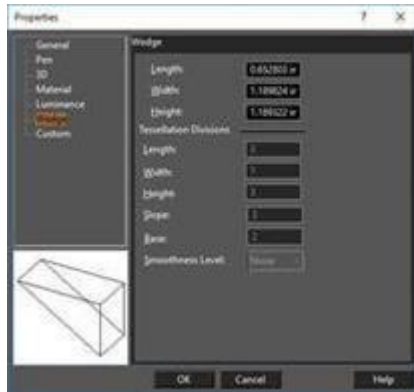
Select a third point to specify the wedge height, or enter the height in the Inspector Bar. You can create the wedge on either side of the workplane.



Wedge Properties

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The **Properties** window of a wedge contains a **Wedge** page, in which you can set the geometric parameters.



For TC Surface Options, see [TC Surface Properties](#)

9.2. Profile Objects and Face to Face Modifications

Profile objects are based on existing 2D objects. The original 2D profile remains unmodified by the 3D tool that references it. One advantage to profile objects is that you can change them by simply editing the profile (or profiles) on which they are based. See [Profile Editing](#)

Branched Lofting

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Profile Based/ Lofts/Branched Lofting

Ribbon UI Menu:

Creates a lofting object by defining profiles along a trunk, and profiles along two or more branches. The profiles lie on different planes, and the planes do not have to be parallel. The result is one 3D object.

Select the profiles that define the trunk. The branching will start at the last profile of the trunk.

Select **Finish selection of trunk** from the local menu or Inspector Bar.

Select the profiles that define the first branch, starting at the profile closest to the end of the trunk.

Select Finish selection of branch.

Select profiles for additional branches the same way.

6. When all branches are defined, select **Finish**. The lofting object is created.

Compound Profiles

(Available in Platinum)

A compound profile is an open or closed chain of connected curves or lines, or any set of closed curves or closed polylines. You can use compound profiles when you do not want to create a polyline or convert a chain into a polyline. Compound profiles can be used as paths for sweeps etc. and as profiles that are swept along paths.

You can use blocks as compound profiles, however, there are restrictions. You can only use one element in a block as a path for any given operation. Also, only closed sequences of curves, lines etc. (chains) or closed objects can be used to sweep, revolve etc. All such closed objects in the block selected to sweep, revolve, etc will be used in the resulting 3D object.

Extrude to Face

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/ Extrude to Face

Ribbon UI Menu:

Extends a 3D object by extruding selected faces to a selected 3D object and performing a boolean.

1. Select the face of a 3D object.

2. Select the 3D object to be the target of the extrusion.

3. The result is a new 3D object.

If the boolean option is set to Subtract, the extrusion will be subtracted from the target body, and the original 3D object from which the extrusion was made will be removed.

You can extrude multiple faces at one time, but you can extrude to only a single 3D object.

1. Hold down the SHIFT key to select multiple faces.

2. Release the SHIFT key and select the target object.

3. You will have a multiple extension.

There are some limitations that must be respected to create valid result. Resulting extrusions must intersect the target body when extended, or the operation will be invalid. You cannot extrude adjacent faces. Adjacent faces are extended along with the extrusion face. Adjacent curved faces and adjacent slanted faces will continue their curve or slant when the selected face it extruded .

You can extend to ACIS surfaces, but only if the target body completely intersects the resulting extrusion.

You cannot extrude a face to the same 3D object that the extrusion face belongs to.

When extruding to a ACIS surface you can remove the remnant "excess" of the target surface using the Facet Edit tool.

Select the facet.

Press the DELETE key.

Face2Face Lofting

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Profile Based/Lofts/Face2Face Lofting

Ribbon UI Menu:

Creates a 3D lofting object between two faces of existing 3D objects. The result is one 3D object.

Select the first face, in this example, the top of the cylinder. Selection order is important.

2. Select the second face, in this case, the sloped face of the wedge.

The result is a loft object that proceeds smoothly between the two faces. The two original objects and the new loft object are now merged into one object.

The resulting object takes the properties (such as Pen) of the object where you selected the first face.

Graphic On Path

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/3D Assemble/Graphic on Path

Ribbon UI Menu:

Places a 2D profile or 3D object along a 2D or 3D path.

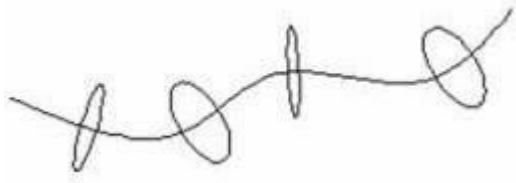
Select the profile or object you want to project, then select a point on the path where you want the projection.



Click more points where you want profile projections.



If you rotate the model, you can see the profiles in 3D.



3. Select **Finish** from the local menu or Inspector Bar to exit the function.

Local menu options:

Sets: Specifies a number of copies of the object that will be placed from the insertion point onward. The space between the insertions will be equal to the value in the Distance field.

Distance: Set the distance between the multiple insertions specified by the Sets field.

Delete Original: Deleted the original object.

Flip Original: Flips the inserted objects, creating a mirror of the original.

Set Graphic on Path Start Point: Sets the insertion at the start point of the path.

Set Graphic on Path End Point: Sets the insertion at the end point of the path. Sets will be ignored if this setting is on.

Lofting

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Profile Based/Lofts/Lofting

Ribbon UI Menu:

Creates a 3D object by connecting 2D profiles. The profiles lie on different planes, and the planes do not have to be parallel. The profiles are connected using NURBS (Non-

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Uniform Rational b-Spline) calculations.

The profiles can be closed or open, but they must be consistent - all open or all closed.

If you want to select only simple (single-object) curves, make sure **Use Compound Profile** is **not** selected.



Select the profiles, in the desired order. Selection order is important

Select **Finish** from the local menu, or double-click on the last profile, to create the loft.

The loft has no sharp corners at the profiles. To create corners, you can create two separate lofts.

If you use open profiles, a 3D surface will result.

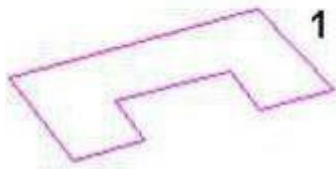
Lofting with Compound Profiles A compound profile is an open or closed chain of connected curves. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline.

Make sure **Use Compound Profile** is selected.

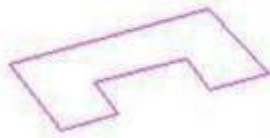


Select the compound profile, which is automatically identified as a chain. To select individual segments hold down the CTRL key and click it, to de-select click it again with the CTRL key. In this example, Profile 1 is a series of connected lines, not a polyline.

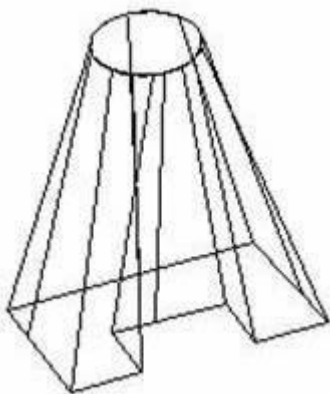
NOTE: if you select individual segments you will be prompted to **Finish the selection of the profile**.



Select the next profiles using the same steps. If subsequent profiles are simple (not compound), you can turn off **Use Compound Profile**.



4. When the last profile is selected, click **Finish** or select it from the local menu.



Note: Compound profiles can be modified, which updates the 3D objects upon which they are based. See [Profile Editing](#)

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Lofting with Guide Lines You can specify guides lines when creating the loft object. In this example, the lower circle and upper square are the loft profiles. The four curves between the profiles are guide lines.

6. Activate **Loft**, and select the loft profiles in the desired order.

Click **Select Guide Lines**, or select it from the local menu.



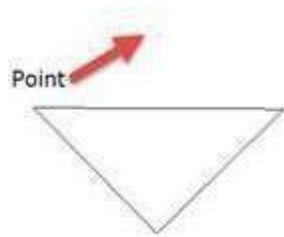
Select each of the guide lines.

Click **Finish** or select it from the local menu. The 3D object transitions between the loft profiles, following each of the guide lines.

Guide Lines: Guide lines can be arcs, 2D or 3D Splines or lines. Beziers, and polyline cannot be used as guide lines. Guide line must touch every profile used in the loft. One end of each guide line must terminate on the first profile used in the loft The other end of that guide line must terminate on the last profile used in the loft. If guide lines do not meet these requirements they will be ignored.

Lofting with Points

Points can be used in lofting. Note: Shaped points may be used but **not** the cross shape.



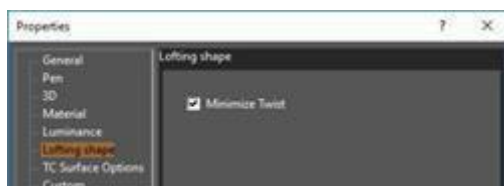
Result:



The shape of the point effects the shape of the resulting loft.

Lofting Shape Properties

The **Properties** window of a loft contains a **Lofting Shape** page, in which you can minimize twist.



Minimize Twist

Effect of Minimize Twist For TC Surface Options, see [TC Surface Properties](#)

Editing Profiles

If you used the compound profile option in the selection of the profiles and/or the guidelines you can post edit them via Edit node.

Prism



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Profile Based/Prism

Ribbon UI Menu:

Creates a 3D object between two 2D objects that lie on different planes. The planes do not have to be parallel.

Both profile objects must be the same type and must have the same number of vertices. For example, you can create a prism between two circles or two rectangles, but not between a circle and a rectangle

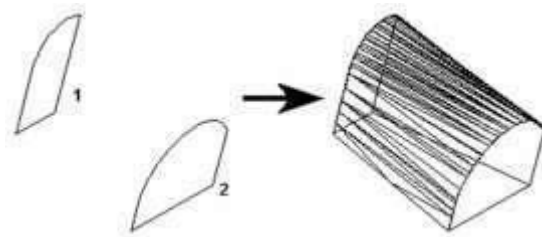
Splines and Bezier curves must have the same number of control points.

Note: A prism is considered to be a **Lofting** object, and its **Properties** contain a **Lofting Shape** page.

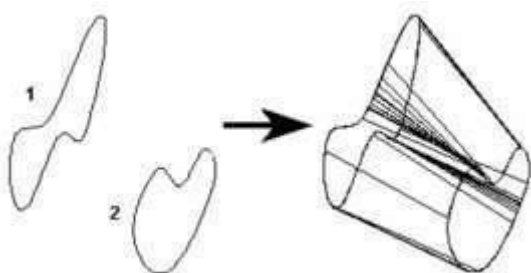
If you want to select only simple (single-object) curves, make sure **Use Compound Profile** is **not** selected.



Select the two 2D objects.

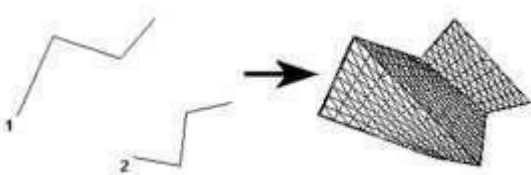


Polyline to polyline



Spline to spline

You do not have to select closed 2D objects. If you use open objects, a 3D surface will result.



Prism with Compound Profiles: A compound profile is an open or closed chain of connected curves or lines. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline.

Make sure **Use Compound Profile** is selected.

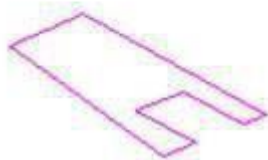
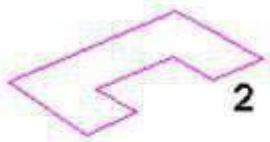
Select the first compound profile, which is automatically identified as a chain. To deselect any curve in the chain, select it again (it will turn green). In this example, Profile 1 is a series of connected lines, not a polyline.

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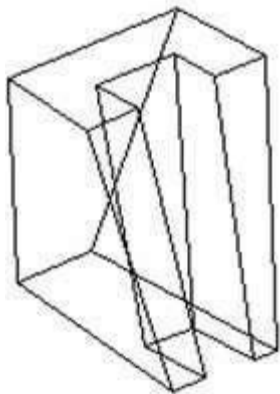
When the profile is selected, click **Finish Selection of Profile**, or select it from the local menu.



Select the second profile using the same steps.



7. When you finish the second profile selection, the prism is created.



Note: Compound profiles can be modified, which updates the 3D objects upon which they are based. See [Profile Editing](#)

Profile Along Path

(Available in Platinum and Professional)

Places a 2D profile along a 2D or 3D path.

Default UI Menu: Draw/Profile Along Path

Ribbon UI Menu:

Select the profile you want to project, then select a point on the path where you want the projection.



Click more points where you want profile projections.



If you rotate the model, you can see the profiles in 3D.



3. Select **Finish** from the local menu or Inspector Bar to exit the function.

Local menu options:

Make Copy Profile: Use this profile to make multiple copies along the path. When

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deselected, you can **Finish** or select another profile.



One Step Back: Removes profiles in the reverse order in which they were created on the path.



Unselect Profile: Unselects the current profile, enabling you to select another one.



Quick Pull

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Modify 3D Objects/Quick Pull

Ribbon UI Menu:

Pulls or pushes a 3D extension of a face (of either a TC Solid or a TC Surface) by extruding a 2D closed area on the face of a 3D object. The extrusion extends along a path normal to the face of the 3D object.

The closed area must be co-planar with the Flat face of a 3D object. It may consist of:

- Any closed 2D entity. E.g. a rectangle.

- Any area that can be hatched by picking a point (with zero gap tolerance).

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Areas enclosed by crossing co-planar, linear geometry, including edges and geometry in blocks E.g. intersecting lines that create a closure.

3D faces. E.g. The face of a cube

Any combination of the above.

Areas created by geometry (including edges on faces) drawn co-planar to any face of a 3D solid E.g. A circle is co-planar with the face of a cube. It overlaps the edge of the cube. The edge divides the circle into two parts, one is a valid area, or intersecting lines that create a closure on the face of a 3D object, with or without defining geometry of the 3D entity

1. Select the 2D closed area to extrude.

Move the mouse to extrude the face, or enter a value in the Height field of the Inspector Bar.

If you are specifying the height by using the mouse, click to finish the extrusion. Or If you are specifying the height by using the height field in the Inspector bar, press Enter to finish the extrusion.

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Note: If the Two Sided property is set, the solid extrusion will be created on either side of the profile.

Examples of areas that can be extruded:

Quick Pull Properties

The Properties window of an Quick Pull object contains a Simple Extrude Shape page, in which you can set parameters defining how the object is created. **Height:** The distance of the extrusion.

Direction: Switch between one-sided and two-sided.

Solid Options: For these parameters to be accessible, the extrusion must be created as a solid.

Draft Angle: Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.

Draft Start / End Distance: If Draft Angle = 0, you can specify a draft angle by entering the draft distances.

Offset: Creates a hole in the extrusion at a distance from the outside of the extrusion equal to the Offset value.

The Quick Pull and the Simple Extrude tool use the same Property Value Preset.

Therefore, changing the property settings in one tool will change the settings in the other.

Quick Pull and 2D Surfaces

You can use The Quick Pull tool to thicken a flat 2D surface. (This must be an ACIS solid / TC surface)

For example,

1. Take a 2D Polyline

2. Extrude it with the Simple Extrude tool

3. Fillet the corner.

4. Then use the Push mode of the Quick Pull tool to give it some thickness.

View the following video for more info on using the the Quick Pull tool on TC Surface.

Demo Video:

<https://youtu.be/Zoy-MlygXHI>

Pulling Curved Faces

Curved faces can also be pulled with the Quick Pull tool. However there are some limitations.

Only faces which have circular curves can be pulled. E.g. Cylinders, Fillets, and Bends.

Elliptical, spline, Bezier or spherical surfaces cannot be pulled.

A curved surface will normally disassociate tangency from adjacent planar surfaces when it is pulled.

Interlocked adjacent curved cases can prevent this from functioning.

To use the Quick Pull on a curve:

Select the tool.

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Hover over the desired surface until it highlights.

Click on the surface.

Move the cursor left or right to push or pull the face in or out. Alternately you can TAB into the Inspector bar, enter a new radius and press ENTER.

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Click to set the new radius.

Fillets

Click on the surface.

Move the cursor left or right to push or pull the face in or out. Alternately you can TAB into the Inspector bar, enter a new radius and press ENTER.

Click to set the new radius.

There are some limitations While it is possible to select the face below it cannot be

pulled because it is locked by the adjacent fillet.

Fillet - Detect Blend network



An additional option for pulling fillets is Detect Blend Network. When this option is **On** all adjacent fillets will be pulled simultaneously. Click on the surface.

Move the cursor left of right to push or pull the face in or out. Alternately you can TAB into the Inspector bar, enter a new radius and press ENTER.

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Click to set the new radius.

Bends

Below are some use cases for pulling bends.

Rail Sweep

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Profile Based/Rail Sweep

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Ribbon UI Menu:

Creates a 3D object by driving a 2D profile along a path. The profile and path do not have to touch or be in different workplanes; the profile section will be "brought" to the start of the path, made normal to the path, and swept along it.

Base Point

If you are going to use Compound profiles for both the path and the profile you will need a base point. The base point specifies the point through which the profile will follow the path.

To create a base point, use the Point tool and place a point at the location adjacent to the profile that you wish to be used. For example, at the end of a line, or at the center of an arc or circle.

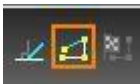
Creating a Rail Sweep

Start with one 2D profile. The profile can be open or closed. Add a 2D or 3D path. The path can lie anywhere, and in any workplane. In this example, profile and path lie in the same workplane.



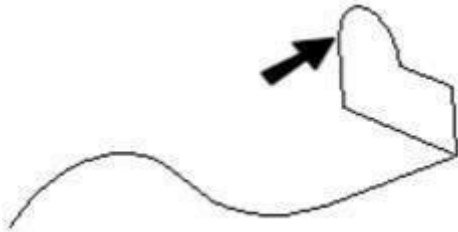
Activate Rail Sweep.

If the profiles consist of compound curves, make sure **Use Compound Profile** is active.



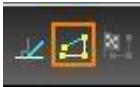
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Select the first 2D profile. If the profile is open, select **Finish Selection of Path** to end the selection. You can then select more profiles if necessary, by pressing the Shift key.

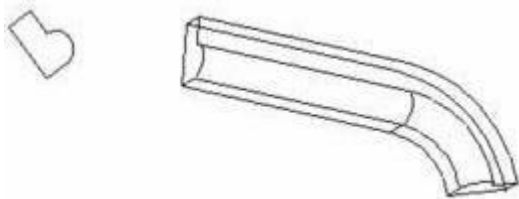


If you are using a Compound profile for the profile you must specify the base point to be used. At this step select the point you created for this purpose.

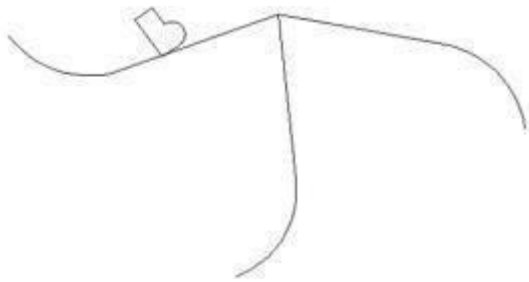
Then select the sweep path. If the sweep path consists of more than one curve, make sure **Use Compound Profile** is active.



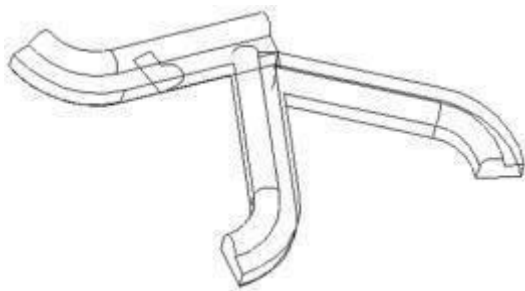
The profile is brought to the path (at the base point if one is being used) and swept along it, normal to the path. The profile intersects the path at its reference point (see [Components of Select Edit Mode](#)).



This tool is handy if you have a single profile that you want to sweep over multiple paths, or rails. In this example, there are three paths for the same profile:

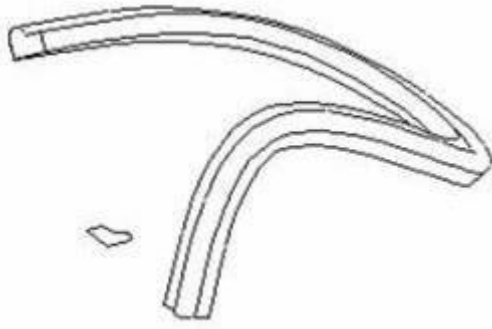


Here are the results:



If you used a Compound profile for the profile, you will be able to modify the resulting 3D object by modifying the original profile object or by moving the base point object. If you used a Compound profile for the path, you will be able to modify the resulting 3D object by modifying the original path object/s. The sweep path does not have to be 2D the path in this example was created using **3D Spline by Fit Points**.

This is the result:



Local menu option

Rigid Sweep: Keeps cross-sections of the solid parallel to one another along the entire path.

Compound Profiles as Paths

You can also use a compound profile as a path

Revolve

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Profile Based/Revolve

Ribbon UI Menu:

Creates a 3D object by revolving a 2D object about a revolution axis, or revolves a 2D object to make a solid imprint on an existing solid via Booleans.

By default, the profile will be revolved 360 degrees, but you can change this angle or create a spiral.

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If you want to select only a simple (single-object) curve, make sure **Use Compound Profile** is **not** selected.

If you want to select a line as the axis of revolution, make sure **Select Revolve Axis** is active



Select a 2D object to revolve.

Select two points of a revolution axis or if **Select Revolve Axis** is active, select the axis line..

The revolved shape is created.

Straight edges of solids can also be used as a revolve axis line.

You do not have to select a closed 2D object to revolve. If you use an open object, a 3D surface will result.

Revolve with Compound Profiles

A compound profile is an open or closed chain of connected curves. You can use compound profiles when you do not want to create a polyline, or convert a chain into a polyline.

Make sure **Use Compound Profile** is selected.



Select the compound profile, which is automatically identified as a chain. To deselect any curve in the chain, select it again (it will turn green). In this example, the profile is a series of connected lines, not a polyline.

You can select more profiles if necessary. When the profiles are selected, click **Finish Selection of Profile**, or select it from the local menu.



Select two points of a revolution axis, or, if **Select Revolve Axis** is active, select the axis line.

The revolved shape is created.

Note: Compound profiles can be modified, which updates the 3D objects upon which they are based. See [Profile Editing](#)

Revolve with Specified Angle

You can revolve with a specified angle and direction.

Select the tool.

Select the **Define revolve angle** option.

Select the profile.

Select two points of a revolution axis, or, if **Select Revolve Axis** is active, select the axis line.

5. Click to specify the angle or specify the angle in the inspector bar.

You can also reverse the direction by changing to the Revolve Clockwise or Revolve Counterclockwise option in the Inspector bar or the Local menu.

Revolve with Imprint

You can revolve a 2D profile to make a solid imprint on an existing solid via Booleans. The Profile must be on the face of the existing solid.

Select the tool.

Select the **Create Imprint** option in the Inspector bar or the Local menu.



Select the **Imprint type Subtract** option in the Inspector bar or the Local menu.



Make sure **Select Revolve Axis** is active.



Select the face that the 2D profile is on.

6. Select the 2D profile.

7. Select the edge of the solid.

Result:

If the Result if the **Imprint type Add** was selected.

Revolution Shape Properties

The **Properties** window of a revolve contains a **Revolution Shape** page, in which you can set geometric parameters.

For the Solid parameters to be accessible, the Revolve must be created as a solid. In the **3D** page of the **Properties** window, make sure **Solid** is selected under **Create Object As**.

Solid Options:

Spiral: The revolution will proceed outward in a spiral pattern

Taper Angle: The revolution will have a draft angle, with the section increasing or decreasing as it progresses. The remaining parameters (**Common Options**) appear for both surface and solid objects

Common Options:

Angle of rotations: Enter a value less than 360 for a partial revolve.

Spiral pitch: The lateral distance between the start and end of the revolve, enabling you to create a spiral. The angle of rotation must be 360.

Number of coils: If **Spiral pitch** is nonzero, you can specify the total number of revolutions.



Clockwise/Counterclockwise: Sets the direction of revolution. The direction depends on how the revolution axis was selected. For **TC Surface Options**, see [TC Surface Properties](#)

Simple Extrude

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Profile Based/Simple Extrude

Ribbon UI Menu:

TurboCAD 2023 User Guide Publication

Creates a 3D object by extruding (sweeping) a 2D open or closed profile along a path normal to the workplane of the 2D object. If you extrude an open profile, a surface will be created.

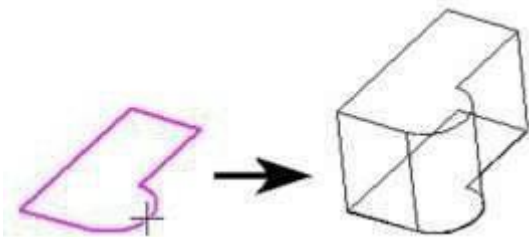
If you want to select only a simple (single-object) curve, make sure **Use Compound Profile** is **not** selected.

If you want to use a compound profile, which is a series of connected lines and/or arcs, select **Use Compound Profile**.

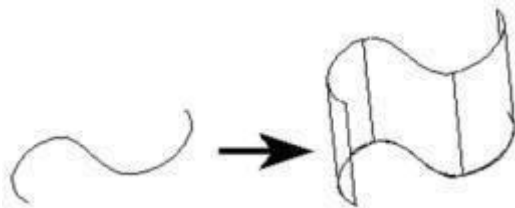


If necessary, select **Finish Selection of Profile** to continue.

Select the 2D open or closed profile to extrude. Move the mouse to extrude the profile, or enter a value in the Height field of the Inspector Bar.



If you select an open profile, the resulting 3D object will be a surface.

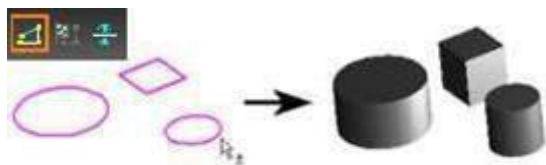


If you select , the solid extrusion will be created on either side of the profile.

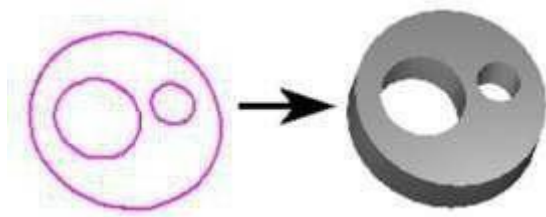
Extruding Multiple Profiles

You can also select multiple closed regions, or profiles, to extrude all at one time.

Make sure **Use Compound Profile** is selected. Select the first profile, then press Shift to select additional profiles. Each profile is extruded the same distance.



If you select nested regions, you can create islands, and regions within islands.

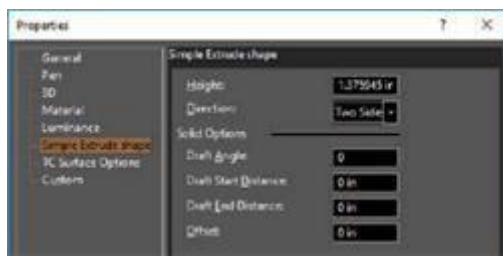


Note: Compound profiles can be modified, which updates the 3D objects upon which they are based. See [Profile Editing](#)

Simple Extrude Shape Properties

The **Properties** window of an extrude object contains a **Simple Extrude Shape** page, in which you can set parameters defining how the object is created.

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Height: The distance of the extrusion.

Direction: Switch between one-sided and two-sided.

Solid Options: For these parameters to be accessible, the extrude must be created as a solid.

Draft Angle: Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.

Draft Start / End Distance: If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.

Offset: Creates a hole in the extrusion at a distance from the outside of the extrusion equal to the Offset value.

For **TC Surface Options**, see [TC Surface Properties](#)

Note: Text can be used as part of a compound profile while extruding.

Now we can select flat 3d polylines or 3d curves for surface extrude(revolve) tools. Also, separated 3d curve and curve (2d curve) have been separated.

Demo Video: You can learn more about Surface Extrude improvements in TurboCAD from below video.

<https://youtu.be/Aqh18XqeYD4>

Sweep

(Available only in Platinum)

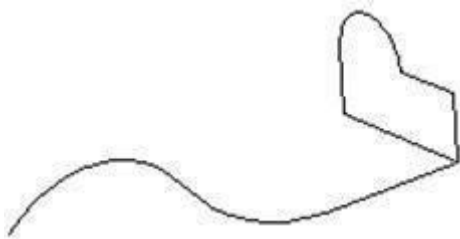
Default UI Menu: Draw/3D Object/3D Profile Based/Sweep

Ribbon UI Menu:

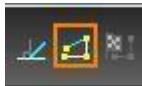
TurboCAD 2023 User Guide Publication

Creates a 3D object by driving a 2D profile along a path.

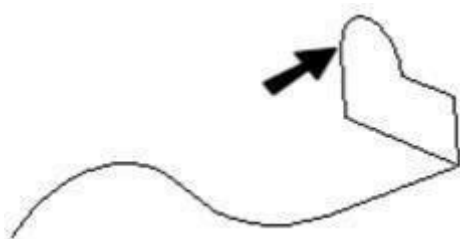
Start with one or more 2D profiles. The profiles can be open or closed. Add a 2D or 3D path. Typically the path intersects the profile and is approximately perpendicular to it, but these conditions are not required.



Activate **Sweep**. If the profiles consist of compound curves, make sure **Use Compound Profile** is active.



Select the first 2D profile. If the profile is open, select **Finish Selection of Path** to end the selection. You can then select more profiles if necessary, by pressing the Shift key.



Then select the 2D or 3D path over which you want to sweep the 2D profiles. If the sweep path consists of more than one curve, make sure **Use Compound Path** is active. When finished, click **Finish Selection of Path**.

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As stated before, the 2D profile and the sweep path do not have to intersect. However, the results will vary depending on how far apart the profiles are. Consider this comparison, in which the profiles on the left intersect and the profiles on the right do not.

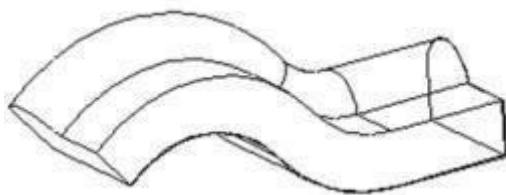


Here are the **Sweep** results: the solid on the right is swept over the offset of the sweep path, and is therefore larger.

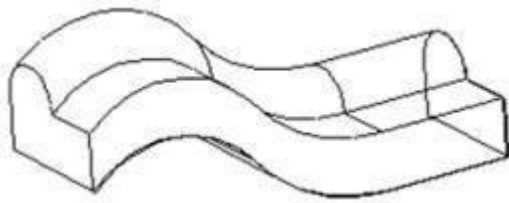


Local menu options

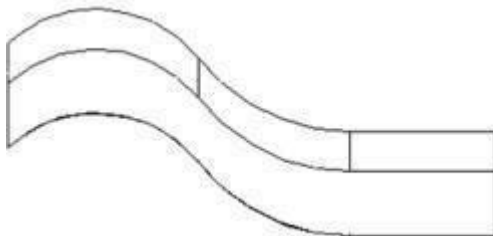
Rigid Sweep: Use this option if you want the cross-sections of the solid to remain parallel to the original swept profile. This solid does not use **Rigid Sweep**; the cross-sections of the solid are always normal to the sweep path.



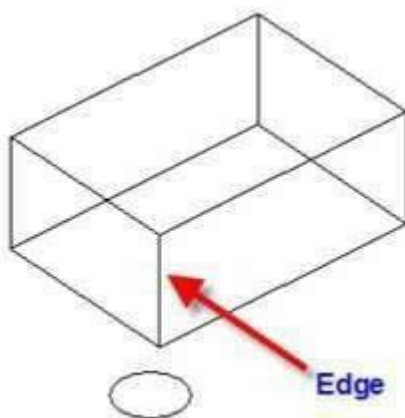
This is the same solid using **Rigid Sweep**; the sections are always parallel to one another, and to the original profile.



If you view the **Rigid Sweep** from this angle, you can see how the sections are parallel.



Use Edge as Path: When this option is **ON** you can use the edge of a 3D object as a path. You cannot use this option with a compound profile.

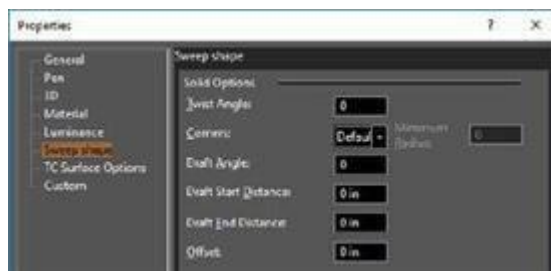


Use Loop as Path: When this option is on you can use the Loop of a 3D object as a path. A loop is all of the edges which enclose a single face of a 3D object. You have to select the 3D object first, then its face. You cannot use this option with a compound profile.

Note: If you are trying to sweep by using a block as the path, you must have the Compound Path option turned on.

Sweep Shape Properties

The **Properties** window of an sweep object contains a **Sweep Shape** page, in which you can set geometric parameters.



For these parameters to be accessible, the sweep must be created as a solid.

Twist Angle: The angle by which the sweep is twisted along the path.

Corners: Defines how the joints of multi-segmented extrusions will be formed.

Default: Sharp corners.

Bend: the rounding will be minimal.

If you want a larger rounding radius, specify a **Minimum Radius**.

Crimp: Keeps internal corners sharp, performs a minimal rounding on external corners.

Draft Angle: Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.

Draft Start / End Distance: If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances. Sweep does not allow you to create a **TC Surface**.

Tweak Face

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Tweak Face

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Ribbon UI Menu:

Extends a 3D object by extruding selected faces to a selected 3D destination object. The result is an ACIS Solid.

The destination object must be a sheet body (ACIS Surface). ACIS surfaces/sheet bodies result when non-closed objects, such as a line segment, are extruded, revolved or swept. They can also be created from ACIS Solids by using the [Create Solid from Surface](#) tool, or by using the [Create Surface from Profile](#) tool on a closed 2D object, and by using the [Extract Entity](#) tool.

Note: if the destination object is a closed volume, such as a sphere or closed revolve, then the destination object will be solidified and added to the resulting tweaked object.

To use the Tweak tool:

Select the Tweak tool.

Select the face of a 3D object.

3. Select the 3D object to be the target of the extrusion.

4. The result is a new 3D object.

Second Example:

There are two options available with the Tweak tool:

Leave source copy: If this option is on a copy of the original object being tweaked will be left in the model.

Leave destination copy: If this option is on a copy of the destination object will be left in the model.

Twisted Extrude

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Profile Based/Twisted Extrude

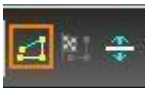
Ribbon UI Menu:

TurboCAD 2023 User Guide Publication

Creates a 3D object by extruding (sweeping) a 2D open or closed profile along a path normal to the workplane of the 2D object and imparting a twist. If you extrude an open profile, a surface will be created.

If you want to select only a simple (single-object) curve, make sure **Use Compound Profile** is **not** selected.

If you want to use a compound profile, which is a series of connected lines and/or arcs, select **Use Compound Profile**.



If necessary, select **Finish Selection of Profile** to continue.

Select the 2D open or closed profile to extrude. Move the mouse to extrude the profile, or enter a value in the Height field of the Inspector Bar.

If you select an open profile, the resultant object will be a surface.

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If you select Two Sided Extrude, the solid extrusion will be created on either side of the profile.

Extruding Multiple Profiles

You can also select multiple closed regions, or profiles, to extrude all at one time.

Make sure **Use Compound Profile** is selected. Select the first profile, then press Shift to select additional profiles. Each profile is extruded the same distance.

If you select nested regions, you can create islands, and regions within islands.

Note: Compound profiles can be modified, which updates the 3D objects upon which they are based. See [Profile Editing](#)

Twisted ExtrudeShape Properties

The **Properties** window of an extrude object contains a **Twisted Extrude Shape** page, in which you can set parameters defining how the object is created.

Height: The distance of the extrusion.

Direction: Switch between one-sided and two-sided.

For the following parameters to be accessible, the extrude must be created as a solid.

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Draft Angle: Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.

Draft Start / End Distance: If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.

Offset: Creates a hole in the extrusion at a distance from the outside of the extrusion equal to the Offset value.

Twist Options

Twist Angle: Specifies the total rotation of the twist.

Distance Type: Specifies where the twist will be applied to the extrusion. Normal applies the extrude between the Start and End distance. Full height applies the twist to the entire extrusion. Twist to Top applies the twist from the Start distance to the top of the extrusion.

Twist Start Distance: Where the twist starts along the extrusion.

Twist End Distance: Where the twist ends along the extrusion.

Continuity: Specifies the smoothness of the transition between the straight sections and the twisted sections. G0 is the most abrupt transition, G1 is a more moderate transition, and G2 is the smoothest transition.

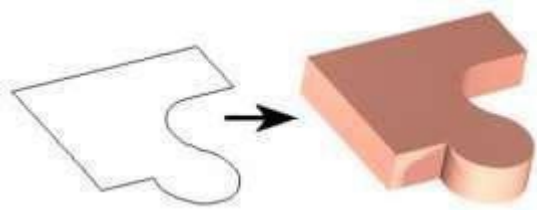
For **TC Surface Options**, see [TC Surface Properties](#)

9.3. Creating 3D Objects by Editing 2D Objects

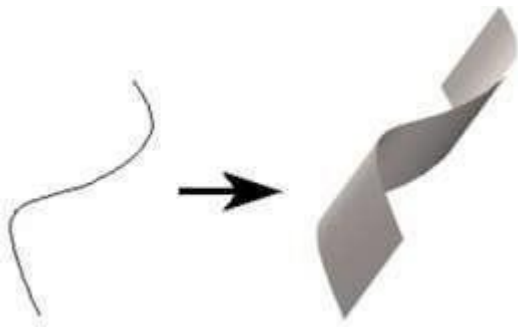
(Available in Platinum, Professional and Deluxe)

An easy way to create a 3D object is to give depth to a 2D object. This is done by opening the **Properties** window of a 2D object (see [Object Properties](#)) to the **3D** page, and entering a thickness.

For closed 2D objects, a solid will result. You can use the **Properties** window to create a surface object instead. See [3D Properties](#)



For an open 2D object, a surface will result.



Note: The original profile can be edited (**Select Edit** or **Edit Tool**) just like any 2D object. Whatever changes you make, the thickness will remain unchanged (unless you change it).

9.4. 3D Mesh



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Mesh

Ribbon UI Menu:

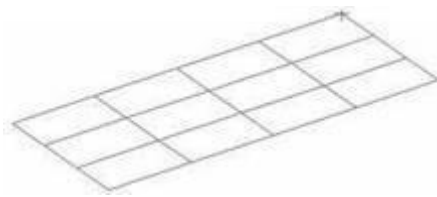
Creates a mesh that represents a set of facets. The base of a mesh is a rectangle with the sides parallel to the X and Y axes.

Note: The **Terrain** tools provide another way to create a meshed surface. See [Terrain](#)

To create this mesh, TurboCAD defines a matrix consisting of X and Y nodes, with each node separated by a **Step** value. The default number of nodes is 14 for each axis, but this can be changed in the Inspector Bar. In addition, you can specify **Interpolation nodes**, which are nodes created between the main X and Y nodes. If you use interpolation nodes, Z values are interpolated between main nodes, thus creating a smoother mesh.

In the Inspector Bar, enter the number of nodes and interpolation nodes. If you know the **Step** value, you can enter that as well.

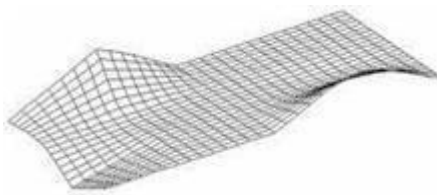
Place the rectangular mesh by selecting a corner point. If the **Step** values are not defined, sizing the rectangle will establish them. Notice that the interpolation nodes do not appear yet.



A coordinates table appears, in which you can enter Z values for each X-Y node. In this table, you can also modify the position of X and Y mesh lines, but you cannot edit X and Y coordinates of individual nodes at this stage of creating the mesh.

	Y	X1	X2	X3	X4
X		-1.75	0.33	2.41	4.5
Y1	-3.89	0	0	-1.0	
Y2	0.4	0	0	0	
Y3	4.7	0	0	0	
Y4	9	0	0	0	1.5
Y5	13.3	0	0	0	0

Click OK. The mesh now contains the interpolation nodes, and the Z values are linear interpolations between the main X-Y nodes.



Tip: Use the **EditTool** to move a single node or a group of nodes (see [Edit Tool in 3D](#)).



3D Mesh Properties

Enables you to change properties of the 3D mesh.



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Table of vertices: Displays the 3D Mesh Coordinates dialog, in which you can adjust the coordinates of each node. Use the **Row Number** spin control to choose the row of the mesh base.

This is different than the coordinates table you used to create the mesh. In this table you can edit coordinates of all points, including interpolation nodes. **Smooth:** Gives the mesh a smooth appearance in render mode.

9.5. 3D Polyline

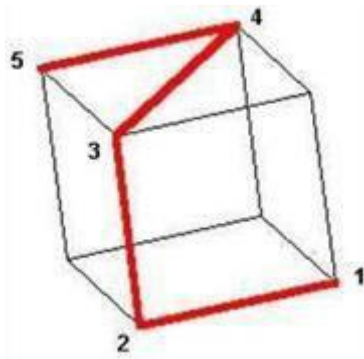


(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Polyline

Ribbon UI Menu:

3D polyline is similar to 2D Polyline (see Polyline), but the user can place line segments anywhere in 3D space. The user is not restricted to their current workplane.



You can enter coordinates in the Coordinate Fields, but there are no input fields in the Inspector Bar for this tool.

Unlike the 2D Polyline, you cannot add arc segments, nor can you manipulate line thicknesses.

Note: If you want to create a 3D polyline by joining existing linear and/or non-linear segments, use the **3D Polyline** option of **Join Polyline**. See [Join Polyline](#)

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3D Polylines can be used in two other tools, **3D Fillet** and **Pipe**.

Local menu option:

Create 2D:

A new option for "Create 2D" has been added for the 3D polyline tool which Enables you to create the 3D polyline as a flat polyline.

Clicking this option from the local menu creates a flat (to the Workplane) 2D object instead of 3D one.

Demo Video:

<https://youtu.be/FRYY-CQInHY>

Open/Closed Polyline

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You can now determine whether the polyline (2D/3D) is closed or not, you can choose to turn on and off without having to go through the node edited tool.

9.6. 3D Spline by Fit Points



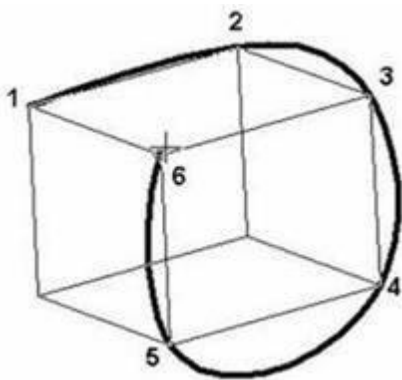
(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/3D Spline/By Fit Points

Ribbon UI Menu:

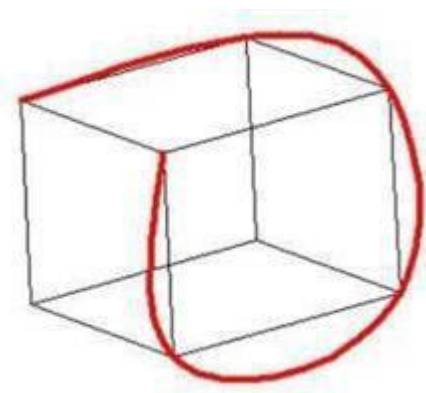
Creates a curve by selecting its control points, similar to the related 2D function (see [Splines and Bezier Curves](#)), but you can place line segments anywhere in 3D space. The curve passes through the points (unlike a spline, which uses control points as a guide).

1. Select the control points in the desired order.

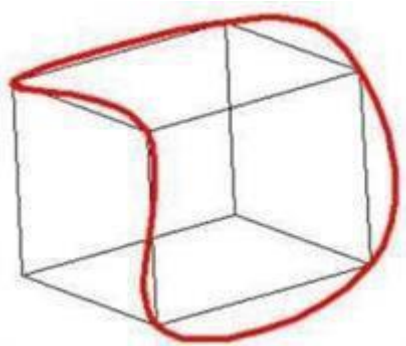


After selecting the last point, select **Finish** from the local menu or press Alt+F. You can also double-click the last point.

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If you want to close the spline, choose **Close** from the local menu instead of **Finish**. In this case, the spline will no longer touch the first and last control points.



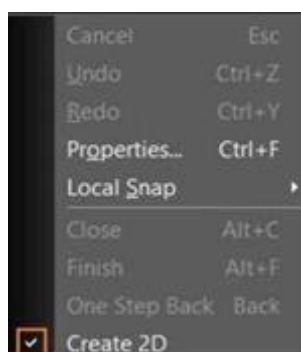
Once created, you can use the **Edit Tool** to change the shape of a spline and add knots.

See [Editing Splines and Bezier Curves](#)

Local menu option:

Create 2D:

Clicking this option from the local menu creates a flat (to the Workplane) 2D object instead of 3D one.



Demo Video:

TurboCAD 2023 User Guide Publication

<https://youtu.be/FRYY-CQInHY>

9.7. 3D Spline by Control Points



(Available in Platinum, Professional and Deluxe)

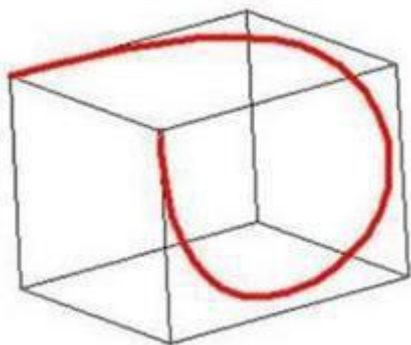
Default UI Menu: Draw/3D Object/3D Spline/By Control Points

Ribbon UI Menu:

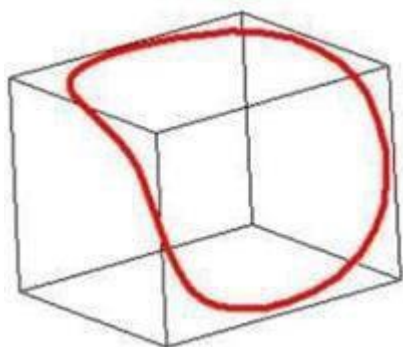
Creates a curve by selecting its control points, similar to the related 2D function (see [Splines and Bezier Curves](#)), but you can place line segments anywhere in 3D space. The curve does not pass through the points (unlike a Bezier curve), rather it uses them as a guide.

1. Select the control points in the desired order.

After selecting the last point, select **Finish** from the local menu or press Alt+F. You can also double-click the last point.



If you want to close the spline, choose **Close** from the local menu instead of **Finish**. In this case, the spline will no longer touch the first and last control points.



Once created, you can use the **Edit Tool** to change the shape of a spline and add knots. See [Editing Splines and Bezier Curves](#)

Local menu option:

Create 2D:

Clicking this option from the local menu creates a flat (to the Workplane) 2D object instead of 3D one.

TurboCAD 2023 User Guide Publication

Demo Video:

<https://youtu.be/FRYY-CQInHY>

9.8. Patterns

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Pattern

Ribbon UI Menu:

Patterns are arrays of objects copied in specific arrangements, and controlled parametrically. Patterns of solid objects act as solid and they and/or their elements can be manipulated by 3D tools, like Bend, and patterns can be Booleaned with other 3D objects. There are six pattern tools: Array, On curve, Radial, Spherical, Cylindrical, Pattern on Polyline.

Associative Arrays

You can use the pattern tools to create 2D and 3D arrays of 2D objects, 3D objects and blocks. Update the arrays in arrangement, number, columns, levels, and rows.

Applying Solid Operations to the Patterns Simultaneously:

Apply to Pattern: This local menu option Apply to Pattern allows users to simultaneously apply solid operation for all Pattern elements This option is available for the following operations which can be applied on the pattern objects simultaneously: Fillet Edges, Chamfer Edges, Face offset, Shell Solid.

Example:

Here's an example using Fillet Edges using Apply to Pattern.

1. Activate Fillet Edges
2. Select an object in the Pattern
3. Select "Apply to Pattern" from the local menu/Inspector Bar
4. The Fillet Object previews for all the pattern elements simultaneously.

5. Click Finish from the local menu and changes get applied to all pattern elements.

Array Pattern

Default UI Menu: Draw/3D Object/3D Pattern/Array Pattern

Ribbon UI Menu:

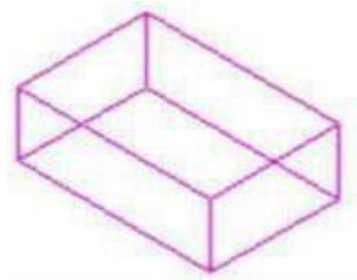
Creates a rectilinear array composed of columns, rows and levels.

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Select the Array Pattern tool.

In the Inspector bar set the Number of columns, rows and levels.

Select a 3D solid.



4. Click to specify the origin of the array.

5. Move the cursor and click to set the X axis of the array.

6. Move the cursor and click to set the Y axis of the array.

7. Move the cursor and click to set the Z axis of the array.

8. The array is created.

The axis values set the distance from center to center of the source object.

Warning: You must have at least two rows to have more than one levels in this array. Reducing rows to one will eliminate the level option until more rows are added.

Local menu options:

Create Association: Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for

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associations to be available.

Leave Source: If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

Leave Destination: If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

By Entity: If this option is on you can select an existing box to set the axis size of the array. With this option on you select the box instead of specifying the origin and axis values. The axis values will be the axis values of the box.

Fit Pattern: If this option is on the axis lengths will define the total length between the centers of the outermost elements in the array along each axis. In other words "fit" within the lengths. If the option is off the axis lengths specify the distance between centers of adjacent objects.

Rectangular Base: Forces the the Y-axis to be 90 degrees to the X-axis

Hexagonal: Creates a hexagonal grid array. If this option is on you will only be allowed to enter an X-axis, unless you use the By Entity setting. **Checker Board:** Creates an alternating checker board array.

Note: Array Properties are available only through the Selection Info Palette..

Array Properties:

The properties available will vary depending upon how you created your pattern.

Fit: If on the array will be fit within the specified axes lengths. Otherwise the axis lengths

specify the distance between the centers of objects.

Cols: Sets the number of columns

Rows: Sets the number of rows

Levels: Sets the number of levels.

X-axis Length: Sets the X axis length.

Y-Axis Length: Sets the Y axis length.

Height: Sets the Z axis length.

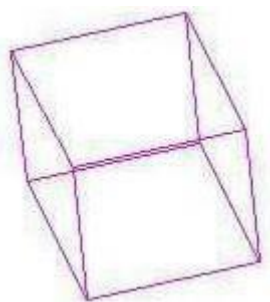
Cylindrical Pattern

Default UI Menu: Draw/3D Object/3D Pattern/Cylindrical Pattern

Ribbon UI Menu:

Select the Cylindrical Pattern tool.

Select a 3D solid.



3. Click to set the center of radial pattern.



4. Click to set the radius of the pattern.

5. Click to set the height of the pattern.

Local menu options:

Leave Source: If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

Create Association: Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available

By Entity: If this option is on you can select an existing circle or an arc or a circular curve on a 3D object to set the size of the array. With this option on you select the circle instead of specifying the center and radius values.

Fit Pattern: If this option is on the height will define the total distance between the top and bottom of the array. In other words "fit" within the height. If the option is off the height specifies the distance between each of the axial sets. **Hexagonal:** Creates a hexagonal alternating cylindrical array.

Radius: Sets the radius for the pattern.

Hole Patterns

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

You can use the pattern tool to layout holes.

1. Create a cube and a point.

2. Start the Radial Pattern tool

Click the Point as the source object, define the pattern path on the top face of cube. A pattern of points will be created.

Start the Hole Tool.

Select the plane face, select the pattern of points, a PartTree with hole operation will be created.

Select the pattern object in the drawing, and change count of elements, for example. The count of holes in Part Tree will be changed accordingly to copies of elements of Pattern Of Points.

Pattern by Curve

Default UI Menu: Draw/3D Object/3D Pattern/Pattern On Curve

Ribbon UI Menu:

Creates an array of objects along an existing curve. Beziers can be used as the curve, so can 2D or 3D splines. Polylines, circles, arcs and lines cannot be used.

Select the Pattern by Curve tool.

In the Inspector bar specify the number of Sets, then press Enter.

Select a 3D solid.

4. Click on a curve to finish the pattern.

Local menu options:

Create Association: Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination option must be active for association to be available.

Leave Source: If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

Leave Destination: If this option is on the curve will remain. If it is off the curve will be deleted.

Non Rotate: If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

Array Properties: The properties available will vary depending upon how you created your pattern.

Fit: If on he array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects.

Count of Elements: Sets the number of items in the pattern.

Non Rotate: If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

Pattern by Polyline

Default UI Menu: Draw/3D Object/3D Pattern/Pattern On Polyline

Ribbon UI Menu:

Creates an array of objects along and 2D or 3D polyline.

Select the Pattern by Polyline tool.

In the Inspector bar specify the number of Sets, then press Enter.

Select a 3D solid.

4. Click on a polyline to finish the pattern.

Local menu options:

Create Association: Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination option must be active for association to be available.

Leave Source: If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

Leave Destination: If this option is on the curve will remain. If it is off the curve will be deleted.

Non Rotate: If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

Array Properties:

The properties available will vary depending upon how you created your pattern.

Fit: If on he array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects.

Count of Elements: Sets the number of items in the pattern.

Non Rotate: If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position.

Radial Pattern

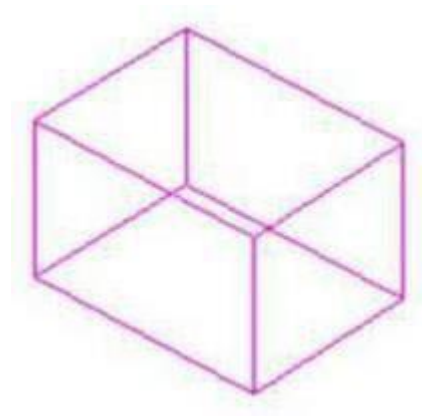
Default UI Menu: Draw/3D Object/3D Pattern/Radial Pattern

Ribbon UI Menu:

Creates a variety of radial array patterns.

Select the Radial Pattern tool.

Select a 3D solid.



3. Click to set the center of radial pattern.

4. Click to set the radius of the pattern.

Local menu options:

Create Association: Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available.

Leave Source: If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

Leave Destination: If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

By Entity: If this option is on you can select an existing circle or an arc or a circular curve on a 3D object to set the size of the array. With this option on you select the circle instead of specifying the center and radius values.

Elliptical Pattern: If this option is selected the tool creates a circular pattern. The number of rings is ignored. This is the default option.

Polar Grid Pattern: If this option is selected the tool creates a polar grid pattern, similar to the polar longitude lines on a globe. The angular number is ignored.

Radial Pattern: If this option is selected the tool creates a radial pattern, similar to the spokes of a wheel.

Non-Rotate: If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position. This option is not available for if Polar Grid is selected.

Fit Pattern: If this option is on the radius will define the total distance between the center of the outermost elements in the array and the center of the array. In other words "fit" within the radius. If the option is off the radius specifies the distance between centers of

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adjacent rings. This option is not available for elliptical patterns.

Hex Symmetry: If this option is on the resulting Polar grid pattern will use a hexagonal arrangement for the spokes instead of a quadrant arrangement. This option is only available if Polar grid is selected..

On Arc: If this option is selected the tool will prompt you to specify the desired arc after you specify the radius. The arc is specified by selecting two points in sequence. This option will be ignored if By Entity is selected.

Array Properties

The properties available will vary depending upon how you created your pattern.

Fit: If this option is on the array will be fit within the specified axes lengths. Otherwise the axis lengths specify the distance between the centers of objects. **Count of**

Elements: Sets the number of items in the pattern.

Start Angle: Sets the angle at which the pattern starts.

End Angle: Sets the angel at which the pattern ends. **Non-Rotate:** If this option is selected the tool creates the designated pattern but the arrayed objects retain the orientation of the source object and are not rotated at each position. This option is not available for Polar Grid is selected.

Fit Pattern: If this option is on the radius will define the total distance between the center of the outermost elements in the array and the center of the array. In other words "fit" within the radius. If the option is off the radius specifies the distance between centers of adjacent rings. This option is not available for elliptical patterns.

Hex Symmetry: If this option is on the resulting Polar grid pattern will use a hexagonal arrangement for the spokes instead of a quadrant arrangement. This option is only available if Polar grid is selected..

Radius: Sets the radius for the pattern.

Array Properties

The properties available will vary depending upon how you created your pattern.

Count of Latitudes: Sets the number of latitudes in the pattern.

Radius: Sets the radius for the pattern.

Elliptical Patterns have the following options:

Count of Rings: set the number of concentric rings.

Radius Step: specifies the distance between each concentric ring. **Z-offset:** sets the height offset for concentric rings. Only available if the value of Count of Rings > 0.

Snaps and Dimensions in 3D

Snaps work in 3D, but are most are projected onto the current workplane.

Therefore, to apply a dimension to a 3D object, you must set the workplane to the plane where you want the dimension to appear. In other words, you can display projected measurements in 3D.

For ACIS solid objects, you can apply **Radius** and **Diameter** dimensions to arc-based objects. You must turn on **Degenerative Faceting** in the **ACIS** page (**Options/ACIS**).

These dimensions are non-associative.

Spherical Pattern

Default UI Menu: Draw/3D Object/3D Pattern/Spherical Pattern

Ribbon UI Menu:

Select the Spherical Pattern tool.

In the Inspector bar set the Number of latitudes.

Select a 3D solid.

4. Click to set the center of radial pattern.



5. Click to set the radius of the pattern.

Create Association: Creates an association between the pattern and the destination object. If subsequently the destination object is edited the pattern will update to reflect the changes made. The Leave Destination and By Entity options must be active for associations to be available.

Leave Source: If this option is on the original 3D object (source) will remain. If it is off the object will be deleted.

Leave Destination: If this option is on the 3D object used as for By Entity will remain. If it is off the object will be deleted.

Use Patterns as Compound Profiles

You can use patterns of 2D objects as a compound profile.

1. Draw a Polyline

2. Create an Array pattern of Polylines.

Start the Simple Extrude tool, Select the Use Compound Profile option. Select the pattern of polylines as the profile for the Simple Extrude operation. The result is Simple Extrude object.

Select the Pattern in the drawing and change Rows value, for example. Simple Extrude will be changed accordingly to pattern properties.

A Simple Extrude can be added to something using Boolean Add. Short Part Tree for such a complex object.

9.9. Smooth Mesh-SMesh

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh

Ribbon UI Menu:

A smooth mesh or "smesh" is a surface object which can be form organic shapes more easily than traditional solids and surfaces. This can be accomplished by specifying a smoothing value to the smesh.

3D tools for SMesh Creation

Most Standard 3D tools can be used to create primitive 3D objects can be used to create a basic smesh. In the 3D properties of most 3D tools there is an option to create the object as a smesh.

In addition some of the 3D tools also allow for the defining of the tessellation and smoothness of the resulting smesh.

Smoothness Level

The smoothness level for a smesh sets how smooth the transition will be on the smesh for each face and between faces. Smoothness works by creating hidden subdivisions of the surface. The higher the level is set the higher the number of subdivisions.

The Smoothness level can be set from the minimum of one to the maximum of 4. A cube set to one tessellation in all values will look like this when the Smoothness level is set to 1:

And Like this when the smoothness level is set to 4:

Warning: Setting a high value for the Smoothness level on object with a large number of faces will slow the application down significantly.

Tessellation Divisions

These divisions refer to the number of surface segments that will be created on the resulting object. The number and type of inputs will vary depending upon the object type. For example, for a Box there are three values (Length, Width, Height), for a Sphere the inputs are called "segments" and are latitudinal and longitudinal.

The minimum value for a division/segment is 1, while the practical maximum value is Above that value deformation may begin to occur, and the application slows down significantly.

SMeshes and Exploding the Primitive

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While it is possible to create a smesh using a 3D primitive tool like Box, there are limitations on what you can do with that smesh. Most smesh editing is done with the Edit tool, but smesh objects created with standard 3D tools will function as that primitive type when you use the Edit tool.

In addition many 3D tools like loft and prism do not allow you to create a smesh directly. To get the maximum utility out of the 3D objects you need to explode the original 3D object. Exploding a box smesh will result in the following when using the Edit tool. When you do this you will lose the ability to set the number of tessellations, because this is no longer a "box", but you will gain access to much more.

To get a smesh for something like a loft you must first explode the loft, and then you must set the object 3D property to the SMesh option.



After you have a smesh you will still be to set the smoothness level, but this option will now be located in the smesh section of the objects properties.

Edit Tool and SMeshes

The full power of smeshes is exposed via the Edit tool. The Edit is used to refine, simplify, move, and transform smeshes by editing their nodes, edges, and faces.

Moving SMesh Nodes, Edges and Faces

To move selected nodes, edges or faces you can drag or specify delta values.

To drag your selection, click and hold the left mouse button on one of the selected nodes, and then drag-and-drop. You must use the nodes to drag the edges and faces as well.

Alternately you can type values into the Delta X, Delta Y, and Delta Z fields, and then press Enter.



Selecting SMesh Edges

To select the edge of a smesh, hold down the Shift key and click on that edge. The selected edge will be highlighted. You can select as many edges as you want while holding the Shift key, and you can deselect as well. There is no way to select multiple edges with a fence, only direct selection is available. When you select an edge both nodes associated with that edge will become selected.

Ignore Back facing

Selecting the Ignore Back facing option will limit the selection face, nodes, and edges to those on the side of the object facing the user.

Selecting SMesh Faces

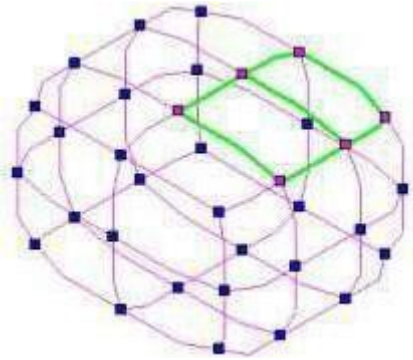
To select the face on a smesh, hold down the Shift key and click in the white space of that face, being careful to avoid edges and nodes.

Sometime there may be multiple faces at a position, stacked one behind the other, however only the closest face will be selected.

The selected face will be highlighted. You can select as many faces as you want while holding the Shift key, and you can deselect in the same manner.

There is no way to select multiple faces with a fence, only direct selection is available.

When you select a face all of the edges and nodes associated with that face will become selected.



Ignore Backfacing

Selecting the Ignore Back facing option will limit the selection face, nodes, and edges to those on the side of the object facing the user.

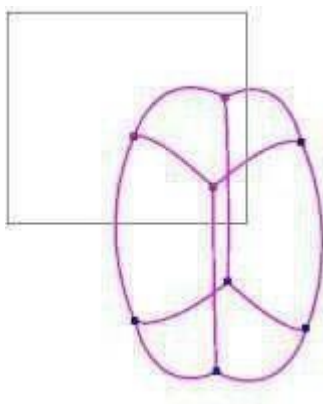
Selecting SMesh Nodes

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With the Edit tool active select the object you wish to edit.

A node can be selected just by clicking on that node. Selected nodes will be highlighted.

You can select multiple nodes by holding down the Shift key and selecting each node in turn. You can deselect any highlighted node by holding down the Shift key and selecting that highlighted node.



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You can also use fence selection to select multiple nodes. To use fence selection, click and hold the left mouse button and drag a rectangular fence around the nodes you wish to select. It is important to note that you cannot begin a selection fence while over the object; you must start the fence away from the object. You can use the Shift key with the selection fence to add/subtract multiple nodes. If you hold down the Shift key while using the selection fence any node within the fenced area which are not already selected will be added to the selection, while any nodes that are already selected will be de-selected.

Ignore Backfacing

Selecting the Ignore Backfacing option will limit the selection face, nodes, and edges to those on the side of the object facing the user.

Extruding SMesh Faces

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Extrude Face

Ribbon UI Menu:

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Faces can be extruded from/into the body of an object to create extensions or depressions. This moves the original face and adds new faces along the axis of movement. You can extrude multiple faces simultaneously. You cannot extrude if the Edit tool is in Selector Mode.

To extrude select the face/s you want to extrude using the Edit tool. [Selecting SMesh Faces](#)

Hold down the Shift key and click on the face you want to select.

Then, select the Extrude Face option from the local menu or the Inspector bar.



A Height field will appear. Type a value into the field, and then press the Enter key.



Positive values will extrude away from the body of the object , negative values will extrude toward the body of the object.

Along Facet Normal

Once the **Ext rude** option is active another option called **Along Facet Normal** will appear. This option is ON by default. **Along Facet Normal** forces the extrusion to move parallel to the normal of the face/facet.

The normal is the direction to which the face is oriented. It is the imaginary line, perpendicular to the center of the face extending out into space.

When **Along Facet Normal** if OFF you can extrude the face by dragging it anywhere in space, by dragging the associated nodes or you can use the **Delta X**, **Delta Y** , and **Delta Z** fields to specify a location for the extrusion.



SMesh Merging

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Merge Smesh

Ribbon UI Menu:

Merging is the opposite of refining. It takes all of the selected faces and creates one larger face. All of the faces must be contiguous (neighboring) and share edges, or they cannot be merged. The Edit tool Selector mode must be OFF or the Merge option will be unavailable.

To merge faces, first select the desired faces.



Once all of the faces are selected, choose the Merge Facets option from the local menu or Inspector bar.

Refining

Sometimes it is desirable to increase the number of faces in a given area. This is called refining and can be done to any selected face. You cannot refine if the Edit tool is in Selector mode, or if the Smoothness Level of the object is set to none.

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To refine a face or set of faces, first select them with the Edit tool. Then select the Refine Face option from the local menu or from the Inspector bar.



The face will be refined based on the Smoothness Level. In essence the invisible subdivisions are converted into new faces for the object. The degree of refinement is as follows:

If the Smoothness Level = 1 the then face (with four edges) will be divided into 2 x 2 (4 new faces) If the Smoothness Level = 2 the then face (with four edges) will be divided into 4 x 4 (16 new faces) If the Smoothness Level = 3 the then face (with four edges) will be divided into 8 x 8 (64 new faces) If the Smoothness Level = 4 the then face (with four edges) will be divided into 16 x 16 (256 new faces)

At a minimum, one new face will be made for each edge of the face. The general rule of thumb for the algorithm is:

Where F is final number of faces, N is the Number of initial edges, S is the Smoothness Level.

$$F=N(2^S)$$

So if the smoothness Level is 3, and the number of edges is 6, the resulting number of

faces will be 96.

SMesh Crease

A crease is a sharp edge in an otherwise smooth surface. Creasing allows you to combine nice smooth curves with hard edges to make more realistic models. Any edge can be set to be creased. You cannot Crease if the Edit tool is in Selector Mode. The Crease value can be: -1, 0, 1, 2 or 3. A Crease value of 0 means No Crease. A

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Crease value of -1 means crease with no smoothing. Crease values from 1 to 3 interact with the Smoothness level. The values define the highest Smoothing Level at which the crease is retained. If the Smoothing Level exceeds this value of the crease, the crease is also smoothed. The degree of smoothing is one degree of for each level of difference between the Crease value and the Smoothing Level. For example:

If the smooth level of the object is 4, and the Crease value is set to 1, the creased edge will have a effective Smoothness Level of 3.

To crease edges, first select all of the edges you wish to crease.

Once you have your edges selected specify the desired Crease value in the Inspector bar and press Enter.



Since selecting faces also selects their associated edges you can also begin your creasing by selecting faces for the desired edges. This can expedite matters in some cases.

Re-patching

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You can use a combination of merging, and refining to re-patch an area with an arrangement of faces that is more conducive to you desired results.

An important point to understand is that while they do opposite functions Merge and Refine are not direct inverses. In other words one does not undo the other. This example is going from merging faces to refining faces.

Another example is going from refining faces to merging faces.

Converting a SMesh

SMeshes can be converted from and to other forms, specifically Solids and TC Surfaces.

Booleans with SMesh

Boolean tools (Add, Subtract, Intersect, Slice) can be used on smeshes, so long as they are combined only with other smeshes or TC Surfaces. The result of any Boolean involving a smesh is a TC Surface. All of the facets of the smeshes and all of their Smoothing level subdivisions are converted to Facets on the resulting TC Surface. This can radically increase the number of apparent facets on the object.

Converting From a SMesh to a Solid

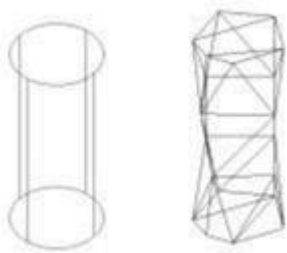
To convert a smesh into a Solid, just toggle the option in the 3D properties of the object. When converting from a TC Surface into a smesh, a set of facets will be generated from all of the facets of the smesh and all of its Smoothing level subdivisions. These are converted to Facets on the Solid. This can radically increase the number of apparent facets on the object, and generate some unpredictability of results. In many cases converting to a TC Surface first, then to a Solid will improve predictability.

Converting from a SMesh to a TC Surface

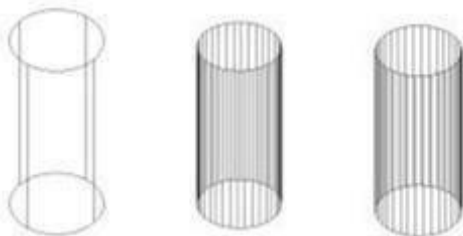
To convert a smesh into a TC Surface, just toggle the option in the 3D properties of the object. When converting from a TC Surface into a smesh all of the facets of the smesh and all of its Smoothing level subdivisions are converted to Facets on the TC Surface. This can radically increase the number of apparent facets on the object.

Converting Solid to SMesh

To convert a Solid into a smesh, just toggle the option in the 3D properties of the object. When converting from a Solid into a smesh, a set of facets will be generated to become facets on the smesh. This can result in unpredictable results. The following is an example of going directly from Solid to smesh.



You can improve the predictability by first converting the Solid to a TC Surface then to a smesh. The following is an example of going from Solid to TC Surface to smesh.



Changing between Solid and smesh for 3D primitives like a box will generally produces more predictable results as well.

Converting TC Surface to SMesh

To convert a TC Surface into a SMesh, just toggle the option in the 3D properties of the object. When converting from a TC Surface into a SMesh all of the facets of the TC Surface are converted to Facets on the SMesh.

Smesh Symmetry Tools



There are three Symmetry tools: Set Symmetry, Mirror Symmetry, and Remove Symmetry. The **Set Symmetry** and **Mirror Symmetry** tools are used to create a Mirror Symmetrical object from a Smesh. This means that only half of all points are available for editing. The points on the opposite side of the mirror are calculated automatically in accordance with mirror (plane of symmetry).

Set Symmetry

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Set Symmetry

Ribbon UI Menu:

The tool looks for the "plane of symmetry" of the object. To use the Set Symmetry tool you must start with a symmetric object. This means there must be a natural division, equal on either side, and passing through edit nodes. For Example You can have a plane through the middle of a 4 wide mesh, but not through a three wide mesh because there are no edit nodes at the center.

1. Draw Smesh box

Explode it to create a Smesh object

Click the **Set Symmetry** tool. As you move the mouse over the object acceptable planes will be highlighted. A series of points that are closed like this are sometimes referred to as loops.

4. Click to set the result .

Mirror Symmetry

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Mirror Symmetry

Ribbon UI Menu:

The **Mirror Symmetry** tool uses Smeshs which have an "open" boundary. An "Open boundary" is a hole in the closed object. To make an open boundary use the 3D Slice tool.

Draw a smesh box with the smooth level set to: None (This is important)

Explode the box

Select the 3D Slice tool.

4. Slice the box.

5. The result should look like this.

Delete one of the resulting parts. Now

Proceed with Mirror Symmetry tool.

Select the "Mirror Symmetry" tool.

Hover over the sliced face.

Select the highlighted boundary.

4. The result should look something like this.

Remove Symmetry

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Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Remove Symmetry

Ribbon UI Menu:

Delete Faces

Deletes selected faces from a Smesh.

To delete faces:

Start the Edit tool.

Select the object.

3. Select the faces you wish to delete.

Disable "Selector mode" if it is enabled from local menu and then right click and choose Delete Facets from the local menu.

Copy Faces

Copies selected faces into a new smesh object.

To Copy faces:

Start the Edit tool.

Select the object.

3. Select the faces you wish to copy.

4. Right click and choose Copy Facets from the local menu.

Add Face

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Add Face

Ribbon UI Menu:

Adds a face to a smesh. The smesh must have a gap or hole in order to be modified.

To add a face:

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1. Select the tool.

2. Select the object.

3. Draw the new face by selecting a series of points.

4. Click Finish in the local menu or inspectorbar.

Split Face



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Split face

Ribbon UI Menu:

Divide any face on a SMesh at any angle.

1. Select a facet on the SMesh object.

Select the first point to define the dividing line.

Select the second point to define the dividing line.

4. The face is split.

Extrude Face



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Extrude face

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Ribbon UI Menu:

Extrude any face of a SMesh along a path defined by a Line, Arc, Polyline or Curve.

1. Select a facet on the SMesh object.

2. Select the path defining object.

3. The extrusion is created.

Split Face by Isoline

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Split Face by Isoline

Ribbon UI Menu:

This tool splits all normal faces along a common isoline (a plane cutting through the object).

To Split Faces:

Select the tool.

Hover over an edge of a face, the isoline will be drawn perpendicular to the edge.

3. Click to split the faces along the isoline.

Note: Isolines do not cross merged faces.

Join Coplanar Faces

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Join Coplanar Faces

Ribbon UI Menu:

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Combines selected faces which share a common plane into a single face.

To Combine faces:

Select the tool.

Select Faces.

3. Click finish.

Options:

Use Neighbor Faces: When this option is on the resulting face will be formed from all faces on the object which are co-planar, and connected on that plane to the selected faces.

Merge Smesh

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Merge Smesh

Ribbon UI Menu:

This tool combine two Smeshs into a single Smesh. The original Smeshs do not have to be touching or overlapping. Nodes that are coincident will be combined into a single node.

To Merge Smeshs:

Select the Merge Smesh tool.

Click on the first smesh.

3. Click on the second smesh.

Solidify

Default UI Menu: Modify/Modify 3D Objects/Surface Mesh/Solidify

Ribbon UI Menu:

This tool takes the surface of any smesh and adds depth to it.

To solidify a smesh:

1. Select the tool

Specify the thickness desired.



Click on the object.

4. Click Finish

Keep Original: The original object is retained and the tool creates a new object.

9.10. Thread 3D

(Available only in Platinum)

Default UI Menu: AddOns/Special Tools/Insert/Thread 3D

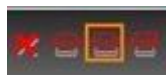
Ribbon UI Menu:

It can also be accessed from Special Tools toolbar.

Creates a threaded bolt with a cylindrical, hexagonal, or square head. Hexagonal and square heads are automatically filleted.

Note: You can display the **Special Tools** toolbar by right-clicking in any toolbar area and selecting **Special Tools**.

Use the Inspector Bar or local menu to select the type of bolt to be created: circular, hexagonal, or square head.

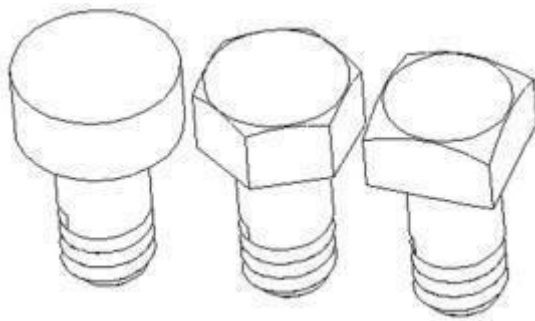


Select the center point of the bolt's base (bottom of threads). You can then specify dimensions using the mouse or by entering them in the Inspector Bar fields.

Length1	ThrDiam	ThrLen	Pitch	BodyDiam	FullLen
-3.665	2.186	1	0.1	0.255	-8.837

The bolt parameters are shown below:

After the last click, or after pressing Enter if you use the Inspector Bar, the bolt is created. The three types are shown below.



9.11. Thread

Thread

(Available only in Platinum)

Default Menu UI: Modify/Modify 3D Objects/Thread

Ribbon Menu UI:

Thread adds a threaded section to any cylindrical surface, including the outside of a cylinder, or the inside of a cylindrical hole. The hole can be the result of using a Boolean to subtract a cylinder from a solid, or created with the hole tool if the hole has no chamfer.

Local menu options:



Length: Sets the total length of the thread.

Pitch: Set the distance between ridges on the thread.

Schematic: Sets the thread display and generation of the thread to a simpler form appropriate for schematics. This form takes the least computation.

Simple: Sets the thread display and generation to a middle form appropriate in which the threads are shown as parallel ridges. This form takes the moderate computation.

Genuine: Sets the thread display and generation to an exact form with true helical threads. This form takes the most computation.

Specify the **Pitch** and **Thread** type in the Inspector bar

Click on a planar face at one end of the cylinder (or hole). The thread length will be measured from this face.

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3. Select the side of the cylinder (or hole).

Move the mouse to specify the **Length**, or enter a value in the **Length** field of the Inspector Bar.

If you are specifying the Length by using the mouse, click to finish the thread. Or If you are specifying the Length by using the Inspector bar, press Enter to set the Length.

9.12. Helix

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Draw/3D Object/Helix

Ribbon UI Menu:

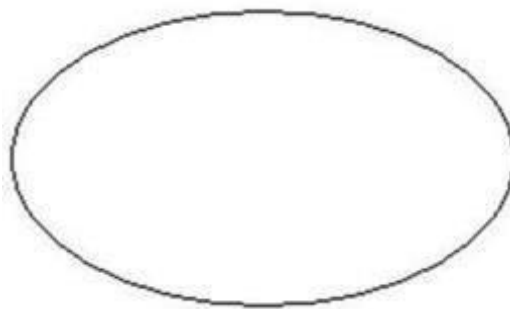
This tool creates a curvilinear helix that can be used for creating springs and other helical elements.

Click to place the Base Point of the helix.

Move the mouse to specify the Base Radius, or enter a value in the Radius or Diameter fields of the Inspector Bar.

If you are specifying the radius by using the mouse, click to finish the extrusion.

Or If you are specifying the radius/diameter by using the Inspector bar, press Enter to



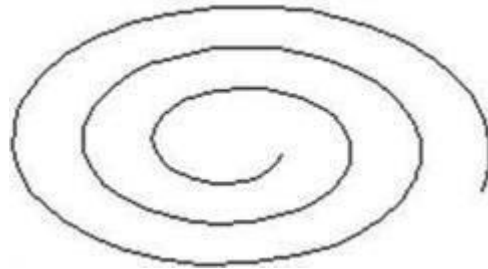
set the radius/diameter.

Move the mouse to specify the Top Radius, or enter a value in the Radius or Diameter fields of the Inspector Bar.

If you are specifying the radius by using the mouse, click to finish the set the Top Radius.

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Or If you are specifying the radius/diameter by using the Inspector bar, press Enter to

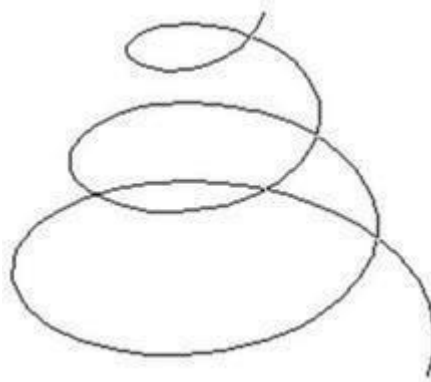


set the radius/diameter.

Move the mouse to specify the height of the helix, or enter values in the Turns and Turn Height fields of the Inspector Bar.

If you are specifying the height by using the mouse, click to finish the helix.

Or If you are specifying the Turns and Turn Height by using the Inspector bar, press



Enter to set the finish the helix.

Helix Properties

The Properties window of a Helix object contains a Helix page, in which you can set parameters defining how the object is created.

Constrain Type: This controls the dynamic for changing the properties by semi-locking the selected property. When one of the other properties is changed, the designated constrained property remains constant, and the third property is altered so that the constrained property can remain constant. The constrained properties work in the following pattern:

If the Turn Height property is constrained, changing the Height property will also change the Turns property, and changing the Turns property will change the Height property. If the Turn Height value is changed the Height property will also change.

If the Height property is constrained, changing the Turn Height property will also change the Turns property, and changing the Turns property will change the Turn Height property. If the Height value is changed the Turn Height property will also change.

If the Turns property is constrained, changing the Height property will also change the Turn Height property, and changing the Turn Height property will change the Height property. If the Turns value is changed the Height property will also change.

General Properties

Height: Sets the total height of the helix.

Turns: Sets the number of turns in the helix.

Turn Height: Sets the height of each complete turn in the helix.

Base Radius: Sets the radius of the first turn of the helix.

Top Radius: sets the radius of the final turn of the helix.

Twist Orientation: these options specify the direction that the helix will turn, clockwise, or counter-clockwise.

9.13. Intersection and Projection

(Available only in Platinum)

These two tool create curves, one by finding the intersection of two solid objects, and the other by projecting a 2D object onto a 3D solid.

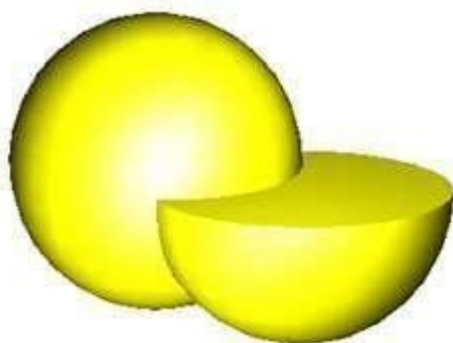
Intersection



Default UI Menu: Modify/Intersection

Ribbon UI Menu:

Creates curves along the intersection of two solids (not surfaces). This tool does not work with surface objects. Select the first and second solid objects. This example shows a sphere and a hemisphere.



The intersection curves are created.

Projection

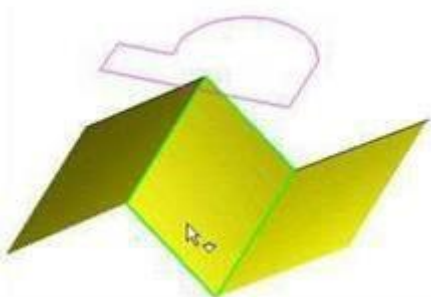


Default UI Menu: Modify/Projection

Ribbon UI Menu:

Projects a 2D curve onto a solid (not surface) face. The 2D curve can be open or closed.

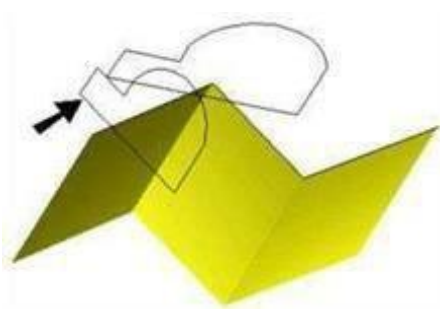
This example shows a 2D polyline and a **Simple Extrude** generated from a line-segmented polyline. First select the 2D curve. Then select the face. Even though the **Simple Extrude** is one object, its faces are still considered separate for this tool.



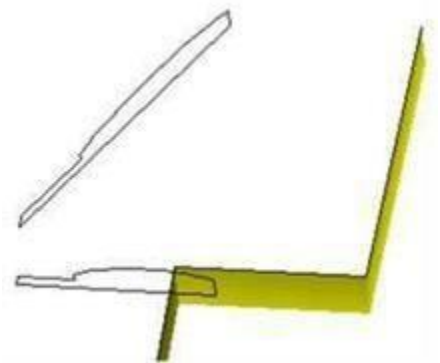
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Note: Even when you are projecting onto a surface such as this one, the object type must be a solid, as set in the **3D** page of the object's **Properties**.

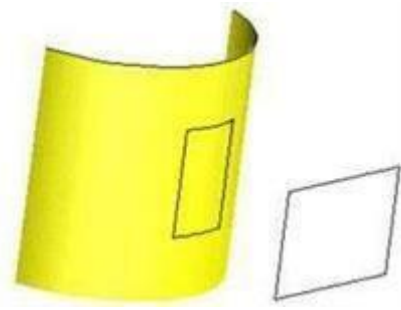
The projection curves are created. For a planar face, the curves can be placed on the face's theoretical extension.



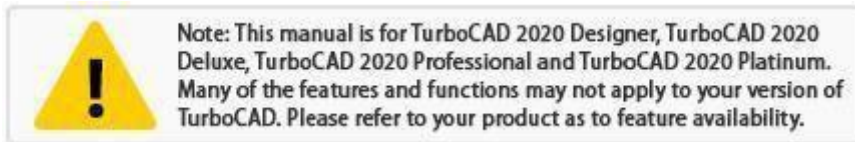
In this view, the projection can be seen more clearly.



If you are projecting onto a curved face, the projection will be scaled toward the center of the curved face.



9.14. Wire Wrap



(Available only in Platinum)

Default Menu UI: Modify/Wire Wrap

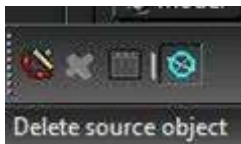
Ribbon Menu UI:

The Wire Wrap tool wraps a 2D object (line, polyline, arc, curve) over 3D Solid object. This create useful paths for extrusions and sweeps.

For wrapping, at least one vertex of the 2D graphic must be on the initial face of the 3D object.

Options:

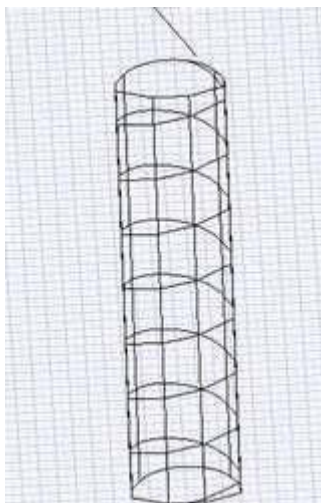
Delete Source: Allows yo to leave the original object in the drawing after the wrap operation.



To Create a Wire Wrap:

1. Define initial target face on body

2. Define planar 2D graphic which is supported by tool to get the result



At least one point of 2D graphic should be on initial face for wrapping. If both the start and end points are on initial face, you will be prompted to select the start point.

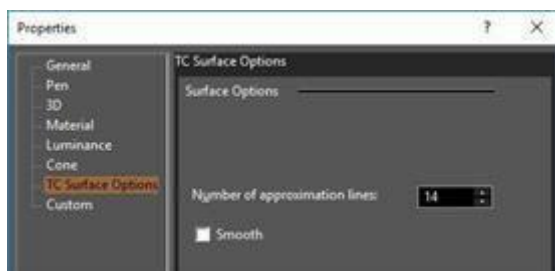
Result:

Another Example:

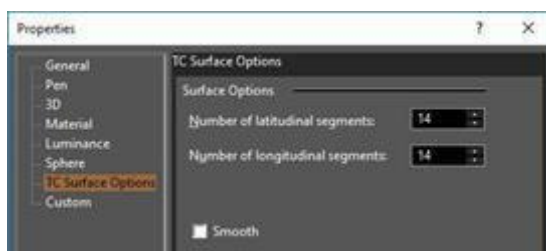
The last image shows the wire used as a path.

9.15. TC Surface Properties

Parameters on this page depend on the type of 3D object. For example, these are a cone's parameters:



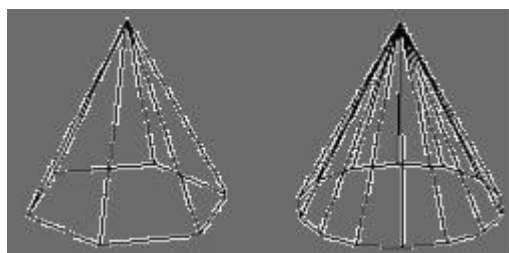
These are the parameters for a sphere:



Most 3D objects have a **TC Surface Options** page resembling one of the above examples.

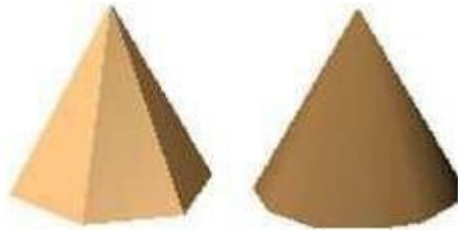
Number of Latitudinal Segments

Number of Longitudinal Segments



Low vs. high number of approximation lines

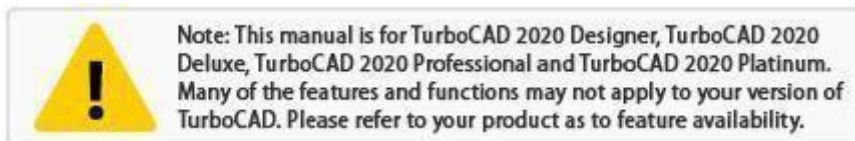
Smooth: Smooths an object when rendering.



9.16. 3D Properties

Controls how objects are created, and their materials.

Note: For options that control how 3D objects are displayed, see [ACIS](#)



Create 3D Object As: Choose between **Solid** (ACIS representation) and **TC Surface** (TurboCAD representation), or SMesh, (Smooth Surface Mesh). 3D objects are created as solids by default, but you can change this **Properties** setting for all 3D tools, or for individual objects.

Solid objects are created using the ACIS solid modeling engine. Solids are more realistic than surfaces, because objects have volume as well as shape. The solid model assumes that a 3D object comprises a framework, a "skin" (set of surfaces) encasing the framework, and inner "body."

TC Surface objects are created using the TurboCAD internal graphics engine. The "inner body" concept does not apply to surfaces, but when an object is trimmed or cut, the resultant object is entirely covered by surface elements. You can create surfaces that are ACIS objects by converting existing objects. See [Surface and Solid Conversion Operations](#)

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Smooth Mesh - SMesh A smooth mesh or "smesh" is a surface object which can be form organic shapes more easily than traditional solids and surfaces. This can be accomplished by specifying a smoothing value to the smesh.

Tip: You can explode a solid object twice to turn it into an exploded (node-editable) surface object.

Thickness: Relevant for 2D objects. Assigning a 2D object a thickness makes it a 3D object. Closed 2D objects become solid or surface volumes; open objects (lines, arcs) become surfaces. See [Creating 3D Objects by Editing 2D Objects](#)

Note: The thickness is assigned perpendicular to the workplane in which the object was drawn, regardless of the current workplane.

Object Specific Faceting

Faceting can be controlled on an individual object basis via the Properties of the object. Object specified faceting overrides the global setting in the ACIS settings.

For details on the Faceter settings see: [ACIS](#)

Two entities with new faceter options:

Render Properties

These properties are used to speed up rendering and have more efficient memory use. These properties work only in render modes and do not affect wireframe or hidden line modes.

Faceting

A set of options that define the faceter mode for the object. Faceting options can be enabled only for ACIS (solid) objects.

Use Faceting: This option turns the local faceter mode on/off. When it is off (by default), this section does not affect the render. If it is on, then render will use the local faceter mode instead of the one from the ACIS dialog to control rendering of this object.

Edit Faceting Properties: This button opens the dialog with local faceter mode settings. Dialog parameters fully match the parameters available in the ACIS Custom parameters.

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This set of additional options define the behavior of the graphic during rendering.

Use Options: This option turns the use of this set of options on/off. If this option is off (by default), then these options do not affect the render. If it is on, then rendering will use the parameters from this section for this object.

Double-Sided: (Off by default) When On this option indicates to the rendering engine that the primitive should be treated as double-sided. By default a primitive is single-sided.

Reversed Normals: (Off by default) When On this option indicates to the rendering engine that orientation of normals will be reversed.

Shadow Receive: (On by default) When Off this option indicates to the rendering engine that the primitive will no longer receive any shadows.

Shadow Cast: (On by default) When Off this option indicates to the rendering engine that the primitive will no longer cast any shadows.

Skylight Portals

If the option **Use as Skylight Portal** is on, the entity will be used as a portal geometry for skylight luminances. The option is 'Off' by default.

Portals are useful for interior scenes for example, where sky light is entering by small openings which are difficult to sample efficiently. By marking explicitly the openings using portals, the engine can render difficult situations at a much higher quality in less time.

For all Luminance: If On the portal will be used for all luminances with skylight in scene, no matter how this sky light was created (drawing luminance or graphic luminance). If Off the portal will be used for this objects luminance only. The option is enabled if option only if **Use as Skylight Portal** is On. The option is 'On' by default.

Using portals

Portals are objects (surfaces) that control the flow of photons from the light source. More precisely: the spatial configuration of the photon's flow.

The simplified algorithm is the following: if the photon has not passed through the portals, it is not considered in the calculation of illumination and the CPU time is not spent on it. Instead of it the next photon is launched (generally at random). This continues until the number of photons passing through the portals will not be equal to the target parameters of the light source. In our case it is a sky light (RedSDK supports only sky lights for the portals). In the case of absence of portals all launched photons are involved in the calculation of the illumination.

So, portals allow you to concentrate the quality and the intensity of the light in the right places of the scene. This facilitates the creation of a scene with proper lighting and

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improves the speed of calculation.

The main purpose of the portals is using them for the openings through which the outer sky light penetrates. Portals need not to be rectangular necessarily. The portal may consist of any set of surfaces. At the same time you should not make the portal very complicated (for example as an ADT window), because of the additional time is required to clarify whether the photon passes through a portal or not. Ideally it should be 1 rectangular facet.

In TurboCAD any 3D primitive can be a portal. This simplifies creation of portals. It should be noted that in this case the geometry of this primitive is not loaded into the scene. In other words an option 'Use as skylight portal' is similar to the option "Load to render luminance only".

Below is shown the effect of the portal's size on the lighting quality and intensity. By increasing the size of the portal 2 times the quality worsens 4 times (at the square).

Below is shown how it is possible to highlight the places in the scene by using the portal's position.

Below is shown how it is possible to highlight the places in the scene by turning the option **For all luminance** Off.

9.17. Curve from Law

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Law Objects/Curve from Law

Ribbon UI Menu:

This tool creates an 2D,3D Curve using Laws. The resulting curve can be used by the Graphic on Path tool and in Sweeps etc.

Laws are mathematical functions in the form of a formula. They are used to define how resulting objects are modified.

They are composed and selected in the Laws Editor which is specific to each tool that uses laws and can be opened from that tool.

To use a the Curve f rom Law tool:

1. Select the tool. The Laws Editor opens.

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Select the desired law in the Laws table by double-clicking and press OK. The law that is applied is what is in the formula field.

Set the U-min and U-max either by entering a values in the Inspector Bar or by selecting two points. (U-min defaults to zero (0))

To create a law:

1. Type the desired formulas into the formula fields.

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$x(u) =$	cos(u)	+ -
$y(u) =$	sin(u)	
$z(u) =$	u	

2. Press the + (plus) button.

3. Type in a title for the law and press OK. The new law will appear in the Laws table

Law title	Law expression
Waveform	$\text{vec}(\cos(u), \sin(u), u)$
Second Wave	$\text{vec}(\cos(u), u, u)$

To Delete a law:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Delete, or press the - (minus) button.

Laws cannot be edited but double click a law will load it into the Formula fields where it can be altered and then a new altered law added.

To Delete all laws:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Clear List.

To Export laws:

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Open the Laws Editor.

Click Export.

Type a file name and click Save.

To Import laws:

Open the Laws Editor.

Click Import.

Select a file and click Open.

Laws cannot be edited but double click a law will load it into the Formula fields where it can be altered and then a new altered law added.

Editing a Curve by Law

The values for a Curve by law can be edited via the Selection Info palette or by the object properties.

$x(u)$ $y(u)$ $z(u)$

In the Offset tool the function are defined as a function of U. Where U is the distance along the object.

Examples:

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Setting $x(u)=\cos(u)$, $y(u)=u$, $z(u)=u$, $U \text{ min} = 0$, $U \text{ max} = 9$

Setting $x(u)=\cos(u)$, $y(u)=\sin(u)$, $z(u)=u$, $U \text{ min} = 0$, $U \text{ max} = 9$

Setting $x(u)=(5+\cos(u))\cos(u/48)$, $y(u)=(5+\cos(u))\sin(u/48)$, $z(u)=\sin(u)$, $U \text{ min} = 0$,
 $U \text{ max} = 301.579987$

9.18. Surface with Law

(Available only in Platinum)

Default UI Menu: Draw/3D Object/3D Law Objects/Surface with Law

Ribbon UI Menu:

This tool creates an open surface using Laws.

Laws are mathematical functions in the form of a formula. They are used to define how resulting objects are modified.

They are composed and selected in the Laws Editor which is specific to each tool that uses laws and can be opened from that tool.

To use a the Surface from Laws tool:

1. Select the tool. The Laws Editor opens.

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Select the desired law in the Laws table by double-clicking and press OK. The law that is applied is what is in the formula field.

Set the U-min, U-max and V-min, V-max either by entering a values in the Inspector Bar or by selecting two points. (U-min and V-min default to zero (0))

To create a law:

1. Type the desired formulas into the formula fields.

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2. Press the + (plus) button.

3. Type in a title for the law and press OK. The new law will appear in the Laws table

Law title	Law expression
base	$\text{vec}(u,v,\sin(u))$
Waveform	$\text{vec}(u,v,\sin(u)*\sin(v))$

To Delete a law:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Delete, or press the - (minus) button.

Laws cannot be edited but double click a law will load it into the Formula fields where it can be altered and then a new altered law added.

Editing a Surface by Law

The values for a Surface by law can be edited via the Selection Info palette or by the object properties.

To Delete a law:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Delete, or press the - (minus) button.

Laws cannot be edited but double click a law will load it into the Formula fields where it can be altered and then a new altered law added.

To Delete all laws:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Clear List.

To Export laws:

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Open the Laws Editor.

Click Export.

Type a file name and click Save.

To Import laws:

Open the Laws Editor.

Click Import.

Select a file and click Open.

$x(u,v)$ $y(u,v)$ $z(u,v)$

In the Offset tool the functions are defined as a function of U and V. Where U is the distance along the x-axis of the object and V is the distance along the y-axis of the object.

Setting $x(u,v)=u$, $y(u,v)=v$, $z(u,v)=\sin(u)*\sin(v)$, $U_{\min}=0$, $U_{\max}=9$, $V_{\min}=0$, $V_{\max}=9$

Setting $x(u,v)=u$, $y(u,v)=v$, $z(u,v)=\sin(u)*\cos(v)$, $U_{\min}=0$, $U_{\max}=9$, $V_{\min}=0$, $V_{\max}=9$

Setting:

$$x(u)=2(1-\exp(u/(6*\pi)))\cos(u)*\cos(v/2)^2$$

$$y(u)=2(-1+\exp(u/(6*\pi)))\sin(u)*\cos(v/2)^2$$

$$z(u)=1-\exp(u/(3*\pi))-\sin(v)+\exp(u/(6*\pi))*\sin(v)$$

$$U \text{ min} = 0, U \text{ max} = 20, V \text{ min} = 0, V \text{ max} = 6.283185$$

Setting:

$$x(u)=(1+v/2*\cos(u/2))*\cos(u)$$

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$$y(u)=(1+v/2*\cos(u/2))*\sin(u)$$

$$z(u)=v/2*\sin(u/2)$$

$$U \text{ min} = 0, U \text{ max} = 12, V \text{ min} = 0, V \text{ max} = 3$$

9.19. Warp by Law

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Bending/Warp Entity by Law

Ribbon UI Menu:

With this tool a Solid entity can be warped using Laws. The result is ACIS Solid.

Laws are mathematical functions in the form of a formula. They are used to define how resulting objects are modified.

They are composed and selected in the Laws Editor which is specific to each tool that uses laws and can be opened from that tool.

To use Warp by Law tool:

1. Select the tool. The Laws Editor opens.

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Select the desired law in the Laws table by double-clicking and press OK. The law that is applied is what is in the formula field.

3. Click on the object to be warped.

Options:

Leave source copy: If this option is selected the tool leaves the original source object in place and creates wrap by law object.

To create a law:

1. Type the desired formulas into the formula fields.

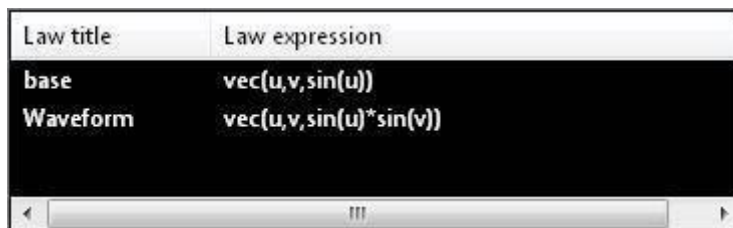


The screenshot shows a dark-themed interface with three rows of formula input fields. The first row is labeled $x(u,v) =$ and contains the text 'u'. The second row is labeled $y(u,v) =$ and contains the text 'v'. The third row is labeled $z(u,v) =$ and contains the text 'sin(u)*sin(v)'. To the right of the input fields is a vertical button with a '+' sign on top and a '-' sign on the bottom.

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2. Press the + (plus) button.

3. Type in a title for the law and press OK. The new law will appear in the Laws table



Law title	Law expression
base	<code>vec(u,v,sin(u))</code>
Waveform	<code>vec(u,v,sin(u)*sin(v))</code>

To Delete a law:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Delete, or press the - (minus) button.

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Laws cannot be edited but double click a law will load it into the Formula fields where it can be altered and then a new altered law added.

To Delete all laws:

Open the Laws Editor.

Click on the Law in the Laws table to select it.

Right click and select Clear List.

To Export laws:

Open the Laws Editor.

Click Export.

Type a file name and click Save.

To Import laws:

Open the Laws Editor.

Click Import.

Select a file and click Open.

$x(u,v)$ $y(u,v)$ $z(u,v)$

In the Offset tool the function are defined as a function of U and V. Where U is the distance along the x-axis of the object and V is the distance along the y-axis of the object.

Examples:

All of the follwowing examples start with this base:

Setting

$$x(u,v)=u$$

$$y(u,v)=v$$

$$z(u,v)=z+\sin(u)$$

Setting

$$x(u,v)=u$$

$$y(u,v)=v$$

$$z(u,v)=z+\sin(u)*\sin(v)$$

Setting:

$$x(u)=u$$

$$y(u)=v+\sin(u)$$

$$z(u)=z$$

Setting

$$x(u,v)=u$$

$$y(u,v)=v+\sin(u)$$

$$z(u,v)=z+\sin(u)$$

Setting

$x(u,v)=u$

$y(u,v)=v$

$z(u,v)=z+(u^2)/5$

$x(u,v)=u$

$y(u,v)=v$

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$$z(u,v)=z+(u^2)/10+(v^2)/5$$

9.20. Covering

Covering



(Available in Platinum only)

Default UI Menu: Draw/3D Objects/3D Profile Based/Covering

Ribbon UI Menu:

Covering is the process of fitting a NURB surface to a set of outer curve boundaries. Covering compliments existing skinning and lofting tools with a technique often used to fill or close openings in surface models. For controlling the interior surface shape, covering also includes the option to specify guide curves.

To create TurboCAD Cover object:

Define the circuit profile. A circuit is a collection of curves, polylines, arcs (2D or 3D) , that form a closed loop. The loop of the profile must not be self -intersecting

Define guidelines. These can be points, curves, polylines, or arcs. Guidelines are an optional parameter in the Cover creation process.

Select the closed profile circuit

Enable 'Use Compound Profile' option (if required) from the Local Menu or Inspector Bar

Select Finish from the Local Menu or from the Inspector Bar.

In addition, when you select and then display the information about the Covering Object in the Selection Info Palette, there is a "MaxSpans" option which when modified, changes the cover stretch according to the value inserted by the user.

10. Editing in 3D

This section covers ways to change 3D object geometry, including node editing and Boolean and blending operations. You can use most 3D editing tools (and all 3D object tools) in both wireframe and render modes. See [Creating a Rendered View](#).



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

10.1. 3D Boolean Operations

3D Boolean Operations

Default UI Menu: Modify/Modify 3D Objects/3D Boolean Operations

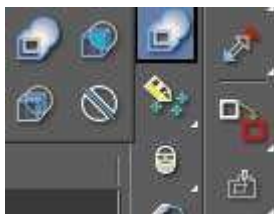
Ribbon UI Menu:

Boolean operations use two existing 3D objects to create a new object. Objects can be combined, subtracted, intersected, or sliced. The objects must both be of the same type - surfaces or solids.

You can display the **Boolean & Facet** toolbar by right-clicking in any toolbar area and selecting **Boolean & Facet**.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.



When Boolean operations are completed, you can edit the shape and location of the objects used to create them. See [Editing 3D Objects using Selection Info](#) . If you plan to assign materials to your 3D objects, it is recommended to do this after the Boolean operations are performed.

Note: The **Imprint** tool also uses Boolean operations, combining a solid with the extrusion of a 2D closed profile. See [Imprint](#)

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Boolean operations can use multi-select modes to select objects as components for operations.



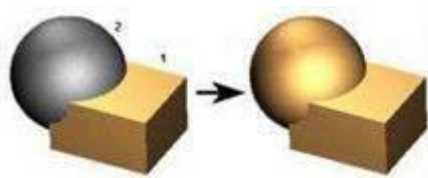
3D Add

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Modify 3D Objects/3D Boolean Operations/3D Add

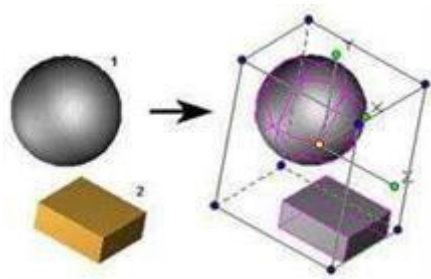
Ribbon UI Menu:

Combines two or more 3D objects into one object. Any overlapping volume is removed. Select two objects to combine, and the second object is joined to the first object. Selection order is important, because the new object will have all the properties of the **f**irst object selected, such as layer and material.



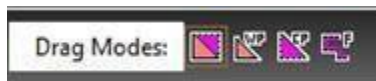
The resulting object is one object. You can verify this by selecting the object.

The selected objects do not have to overlap. If you combine non-overlapping objects, they will still be combined into one object.



To select multiple objects hold down the shift key while selecting, then select **Finish Selection** from the local menu.

Note: Boolean operations can use multi-select modes to select objects as components for operations.



3D Intersect



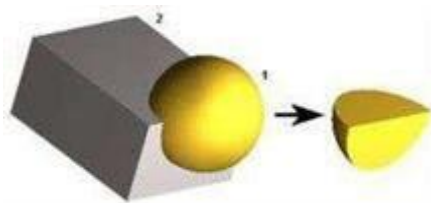
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(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Modify 3D Objects/3D Boolean Operations/3D Intersect

Ribbon UI Menu:

Creates one 3D object that is the overlap (intersection) between two objects. Select two objects that overlap. Selection order is important, because the new object will have all the properties of the **first** object selected, including layer, and material.



Local menu option:

Keep Original: When this option is selected the original objects used in the intersection will be retained.

3D Subtract

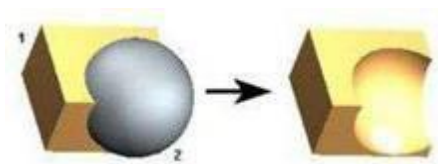


(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Modify 3D Objects/3D Boolean Operations/3D Subtract

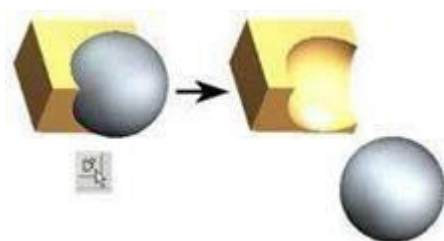
Ribbon UI Menu:

Subtracts one or many 3D objects from one 3D object. First select the object to subtract from, then select the object to subtract. By default, the second object is deleted.



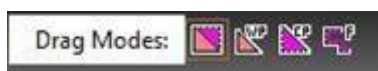
Local menu option:

Don't remove the subtrahend: Keeps the second (subtracted) object. The resultant objects appear the same, but if you move the second object you can see that it was subtracted from the first.



To select multiple objects hold down the shift key while selecting, then select **Finish Selection** from the local menu. The first object will act as the minuend, and all of the subsequent objects will be subtrahends.

Note: Boolean operations can use multi-select modes to select objects as components for operations.



3D Slice



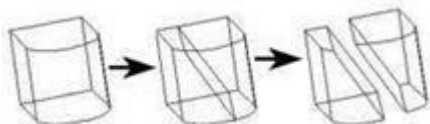
Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/3D Boolean Operations/3D Slice

Ribbon UI Menu:

Slices, or divides, one or more objects into new objects, by specifying the slicing plane.



Note: The **Section** tool is similar, but it creates a section of the object on the specified plane, while leaving the original object intact. See [Section](#)

Slice by Line

Slices objects by a plane perpendicular to the current view, defined by two points. This is the default option.

1. Select **Slice by Line (2 Points)** from the local menu or Inspector Bar.

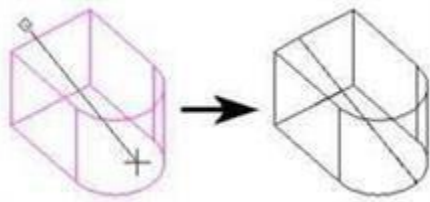


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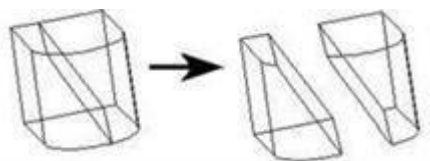
Select the objects to slice. You can use Shift to select more than one object.

Select two points The slicing plane will pass through the line defined by these points, in the direction normal to the current view (into the screen).

The slice is created, though in the current view the dividing plane appears as a line.



You can rotate the view and move one of the new objects to see how the original object was sliced.



Slice by Plane

Slices objects by defining the slicing plane. The plane is defined by three points.

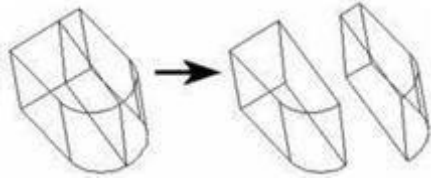
Select **Slice by Plane (3 Points)** from the local menu or Inspector Bar.

Select the object to section. You can use Shift to select more than one object.

Select three points to define the slicing plane.

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The slice is created. You can move one of the new objects to see how the original object was sliced.



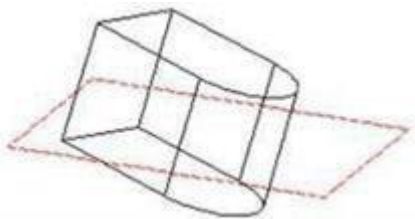
Slice by Workplane

Slices objects by using the current workplane as the slicing plane. For details on workplanes, see [Workplanes](#)

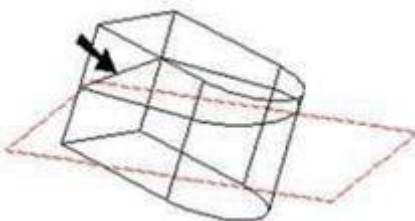
Select **Slice by Workplane** from the local menu or Inspector Bar.



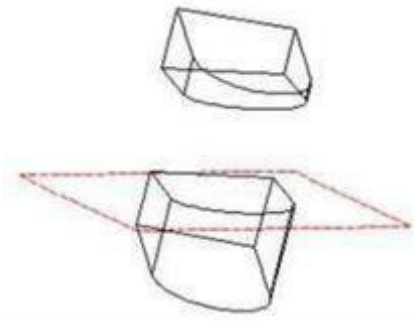
Select the object to slice.



Note: To slice multiple objects, start out using a different option - **Slice by Line** or **Slice by Plane**. Use Shift to select the objects to slice, then click the **Slice by Workplane** icon. The object is sliced.



You can move one of the new objects to see how the original object was sliced.



Slice by Surface

Slices the object by using a surface. The surface must be a ACIS solid surface.

1. Select **Slice by Surface** from the local menu or Inspector Bar.

2. Select the object to be sliced.

Note: To slice multiple objects, start out using a different option - **Slice by Line** or **Slice by Plane**. Use Shift to select the objects to slice, then click the **Slice by Workplane** icon. The object is sliced.

3. Select the the object to do the slicing.

You can move one of the new objects to see how the original object was sliced.

Slice by Facet

Slices objects by a plane perpendicular to the current view, defined by two points. This is the default option.

Select **Slice by Facet** from the local menu or Inspector Bar.



Select the objects to slice. You can use Shift to select more than one object.

3. Select the facet that will be used to slice the object.

You can rotate the view and move one of the new objects to see how the original object was sliced.

Delete a Sliced Part

Removes one of the new objects created by slicing objects. This option is used in tandem with one of the other options.

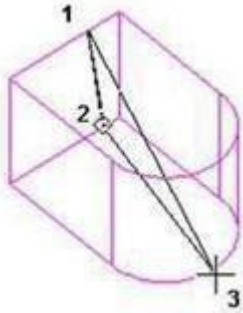
Select the desired slicing option (in this case, **Slice by Plane**) and **Delete a**

Sliced Part from the local menu or Inspector Bar.

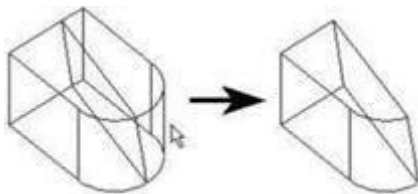


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Select the object to section. You can use Shift to select more than one object.
Slice the object by the selected method, in this case, defining a plane by three points.

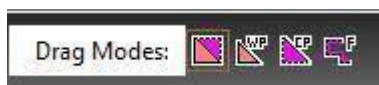


3. The slice is created. Select the object you want to delete.



Tip: Objects created as a result of slicing can also be deleted by selecting them and pressing the Delete key.

Note: Boolean operations can use multi-select modes to select objects as components for operations.



Leave Graphic Associations



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If this option is on graphic association to the sliced part will be retained. It is on by default.

Delete Graphic Associations



If this option is on graphic association to the sliced part will be deleted.

Exploding Solids



Note: This manual is for TurboCAD 2019 Designer, TurboCAD 2019 Deluxe, TurboCAD 2019 Professional and TurboCAD 2019 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum, Professional and Deluxe)

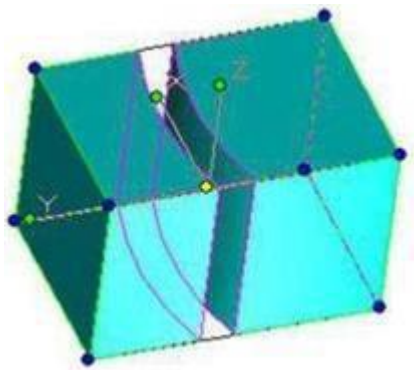
Default UI Menu: Modify/Explode

Ribbon UI Menu:

If you use Boolean functions such as **3D Add** or **3D Subtract**, it is possible to get a solid comprised of two or more separate objects that are still considered to be one object. It can be difficult to split this solid using 3D editing tools, but the **Explode** tool can be easily used for this.

1. Start with two solids that overlap.

Use **Boolean Subtract** to remove one solid from the other. In **Select** mode the two parts are considered to be one object.



3. With the object still selected, activate **Explode** (**Format / Explode**).



Note: For more information on exploding, see [Exploding Objects](#)

The two parts are now considered to be separate objects.

MultiAdd_VB6

(Available in Platinum, Professional, and Deluxe)

Default UI Menu: SDK/Boolean/MultiAdd_VB6

Ribbon UI Menu:

Combines two or more 3D objects into one object. Any overlapping volume is removed. Select two or more objects to combine, then click the MultiAdd_VB6 tool. All of the selected objects will be combined into one object. The resulting object is one object. You can verify this by selecting the object.

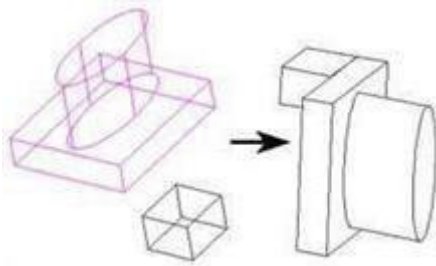
Note: Not Currently Available in 64-bit editions

10.2. Assembling

Default UI Menu: Modify/Modify 3D Objects/3D Assemble

Ribbon UI Menu:

The **Assemble** tools are used to position a 2D or 3D object relative to another object.



The **Assemble** tools can be used on 2D and 3D objects. For the objects you want to use, be sure the Selector is set correctly (2D, 3D, or both). See [2D-3D Selector](#). These tools are available on the **3D Modify** toolbar, which you can display by right-clicking in any toolbar area and selecting **3D Modify**.



Note: The **Transform** tools also enable you to move objects, as well as scale and copy, and do not require information as accurate as that required by the **Assemble** tools. In addition, you can record **Transform** operations for use on other objects. See [Transforming](#)

Assemble by 3 Points



(Available in Platinum, Professional and Deluxe)

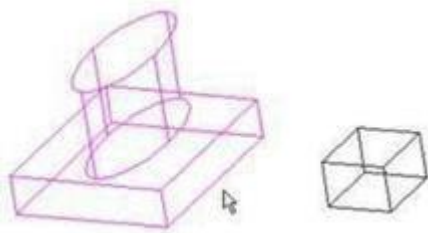
Default UI Menu: Modify/Modify 3D Objects/3D Assemble/Assemble by 3 Points

Ribbon UI Menu:

Changes the position of an object by changing the location of one point, two points (line), or three points (plane). The source points typically lie on the object to be moved, although this is not required. Destination points can lie on destination objects, or they can be specified in the Coordinate Fields.

To assemble by one point:

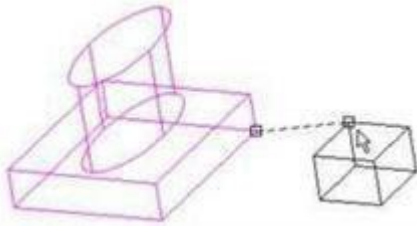
1. Select the object to be repositioned.



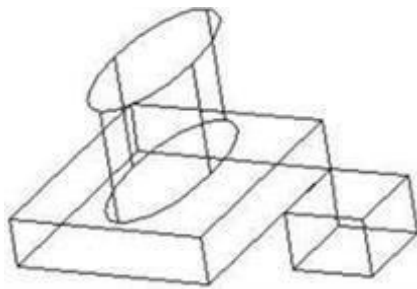
2. Select the source point.

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3. Select the destination point.

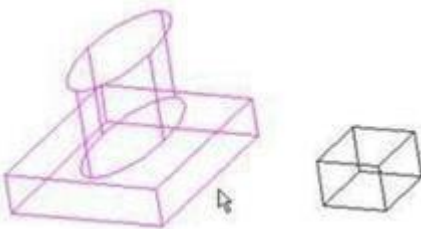


Select **Finish** from the local menu or Inspector Bar. The object is moved so that its source point meets the destination point, keeping its orientation.

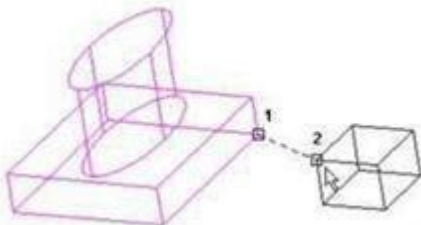


To assemble by two points (line):

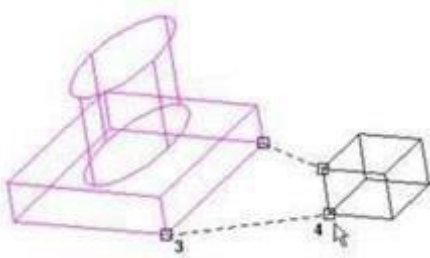
5. Select the object to be repositioned.



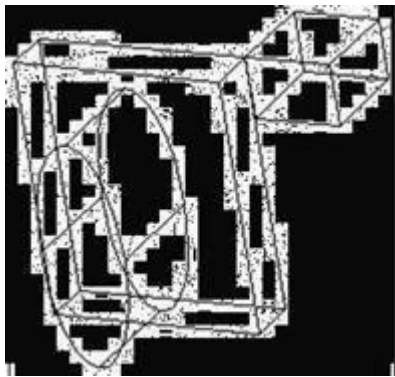
Select the first source point and its destination. This first set of points determines the actual object position; the remaining points set the orientation.



7. Select the second source point and its destination.



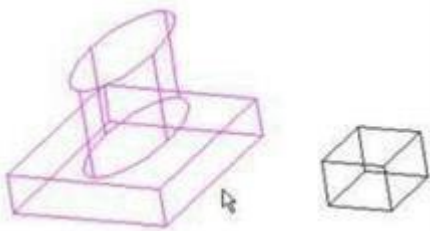
Select **Finish** from the local menu or Inspector Bar. The object is moved so that its first source point meets the first destination point, and the line between source points becomes aligned with the line between destination points.



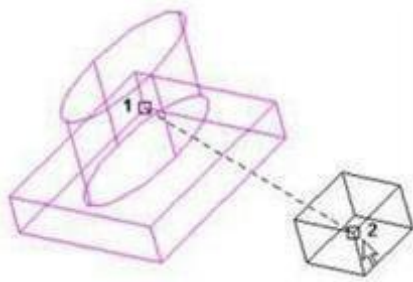
Note: For another way to assemble by edges, see [Assemble by Edge and Point](#).

To assemble by three points (plane):

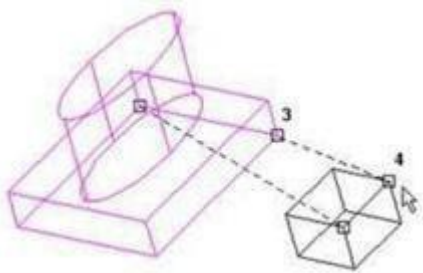
9. Select the object to be repositioned.



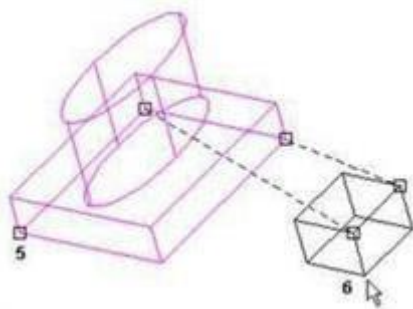
Select the first source point and its destination. This first set of points determines the actual object position; the remaining points set the orientation.



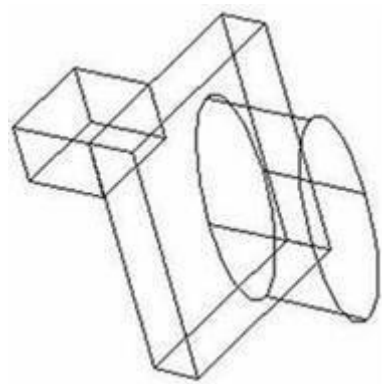
11. Select the second source point and its destination.



12. Select the third source point and its destination.



The object is moved so that its first source point meets the first destination point, and the plane defined by the source points becomes aligned with the plane defined by the destination points.



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Note: In a particular case, just after two clicks "finish" is enabled in local menu. But in common case you can repeat 2 point selection 3 times: 6 clicks. So you can achieve any position of the object in space

Assemble by Edge and Point



(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Modify 3D Objects/3D Assemble/Assemble by Edge and Point

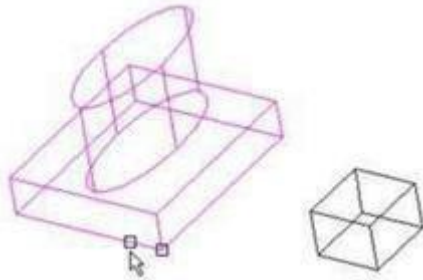
Ribbon UI Menu:

Changes the position of an object by changing the location and alignment of an edge. You can also add a source point and destination point to change the rotation as well. **To assemble by a pair of edges:**

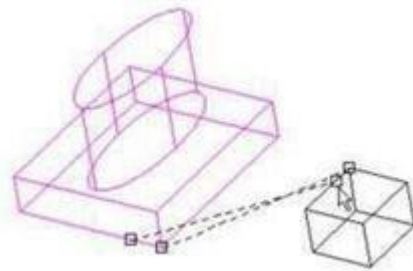
1. Select the object to be repositioned.

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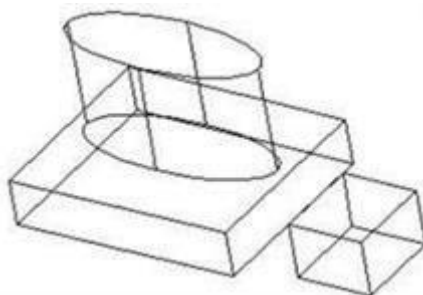
Select the source edge. The new alignment of the edge depends on where you select the edge; the point you select will be moved to the point you select on the destination edge.



3. Select the destination edge. The dotted lines indicate how the object will be aligned.

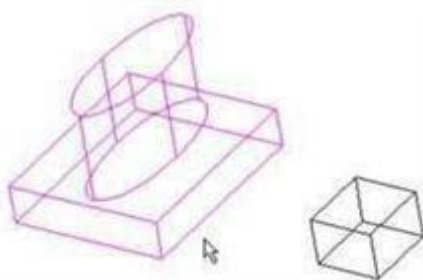


Select **Finish** from the local menu or Inspector Bar. The object is moved so that its source edge meets the destination edge, connected at the two selected points.

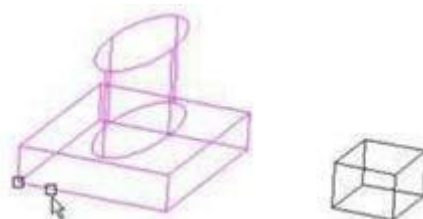


To assemble by edges and a plane:

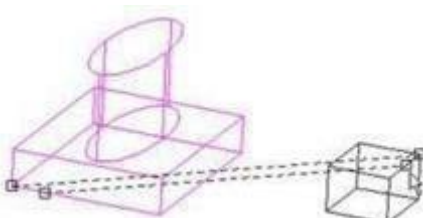
1. Select the object to be repositioned.



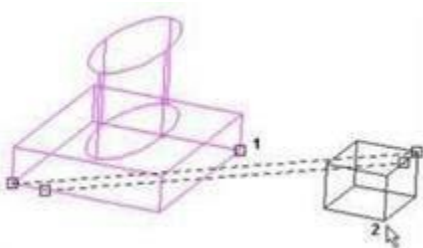
Select the source edge. The new alignment of the edge depends on where you select the edge; the point you select will be moved to the point you select on the destination edge.



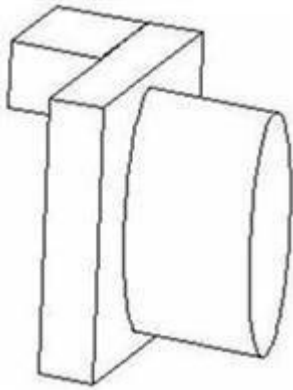
3. Select the destination edge. The dotted lines indicate how the object will be aligned.



Select a point on the source plane (not on the selected edge) and a point on the destination plane.



The object is moved so that its source edge meets the destination edge, connected at the two selected points. The rotation is set by the points on the source and destination planes. The results are shown here in Hidden Line render mode.



Assemble by Facet

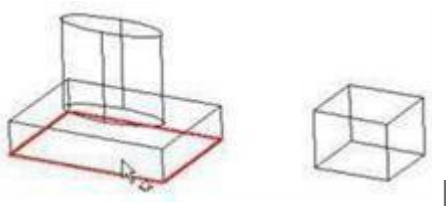
(Available in Platinum, Professional and Deluxe)

Default UI Menu: Modify/Modify 3D Objects/3D Assemble/Assemble by Facet

Ribbon UI Menu:

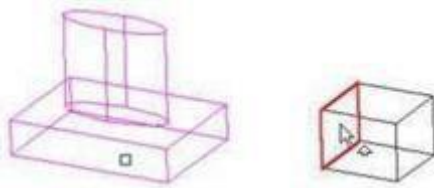
Changes the position of an object by aligning facets.

Select the source facet of the object to be repositioned. To select a facet behind or in front of the indicated facet, you can use the Page Up and Page Down keys.

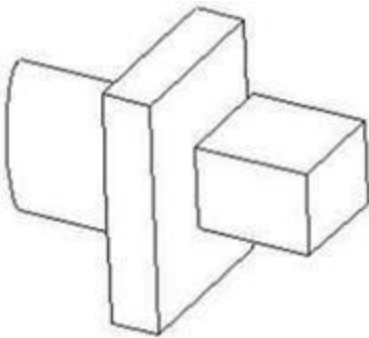


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2. Select the destination facet.



The object is moved so that the source facet meets the destination facet. The results are shown here in **Hidden Line** render mode.



Assemble by Axis

(Available only in Platinum)

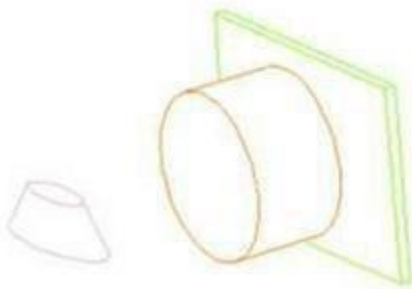
Default UI Menu: Modify/Modify 3D Objects/3D Assemble/Assemble by Axis

Ribbon UI Menu:

Changes the position of an object by aligning axes.

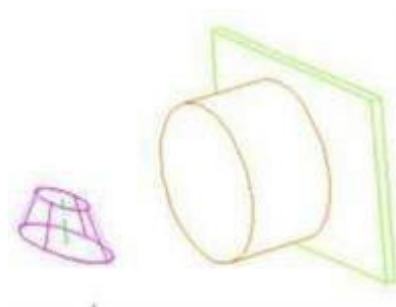
Assembling Circular or Rotated Objects

For round objects that have a rotational axis (such as **Cylinder**, **Cone**, **Polygonal Prism**, **Revolve**, or **Simple Extrude** based on a circular 2D object), you need to set the assembly axis before the objects can be assembled. This example will assemble a truncated cone based on an ellipse, aligning it with the axis of a cylinder.

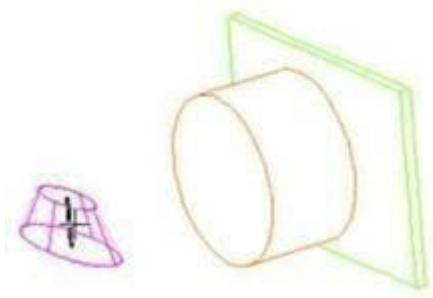


Activate Set Assembly Axis.

Click one of the round objects (the truncated cone, in this case). Its axis is displayed as a dashed line.

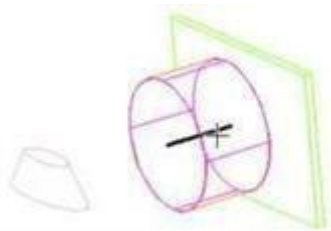


3. Click the axis to define it for the assembly.



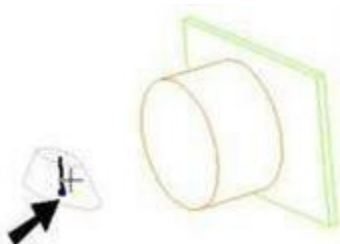
Select **Finish** from the Inspector Bar or local menu.

Do the same to define the axis for the other round object (the cylinder).

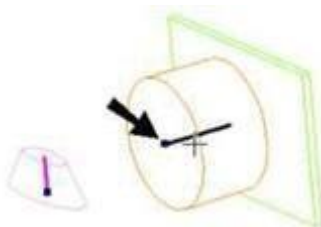


Select **Finish** from the Inspector Bar or local menu.

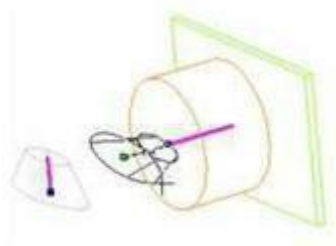
Activate **Assemble by Axis**. Click the axis of the object you want to move. Be sure to select the axis near the endpoint that will meet the endpoint of the other axis.



8. Click the destination axis, near the endpoint where the other axis will meet it.



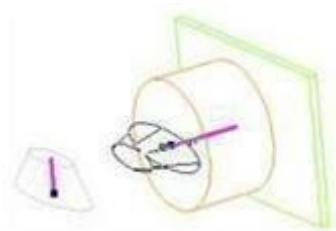
A preview of the moved object appears. You can move it in either direction along the axis.



Use the mouse or enter the move distance. If you want to rotate the object, you can enter an Angle. If the moved object is oriented backward, select **Turn Over** from the local menu or Inspector Bar.



To rotate it with the mouse, select **Rotate**. If you turned or rotated the object, the preview will update.



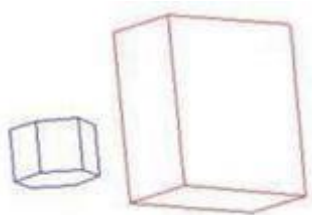
When the distance and angle are defined, the object moves so that the two axes are aligned.



Assembling Planar or Linear Objects

For objects that have no assembly axis defined, you can define the axis during the assembly process. This example uses a polygonal prism and a box.

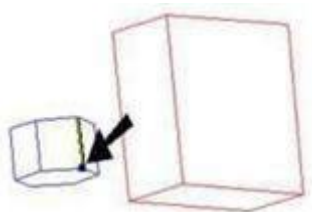
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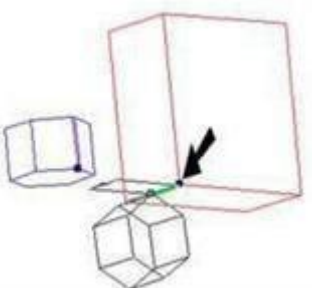
Activate **Assemble By Axis**, and select **Use Any Line** from the local menu or Inspector Bar.



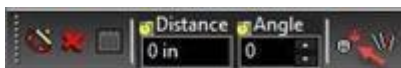
Select the object you want to move. Then click a line or edge you want to use as the assembly axis. Be sure to click near the endpoint of the axis which will meet the other axis.



Click an edge or line as the destination axis. Select it near the endpoint that will meet the other axis. A preview of the moved object appears. You can move it in either direction along the axis.

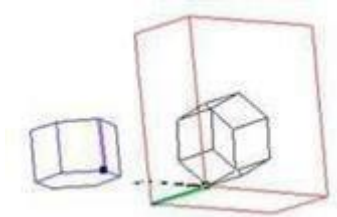


Use the mouse or enter the move distance. If you want to rotate the object, you can enter an Angle. If the moved object is oriented backward, select **Turn Over** from the local menu or Inspector Bar.

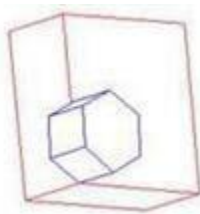


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To rotate it with the mouse, select **Rotate**. If you turned or rotated the object, the preview will update.



When the distance and angle are defined, the object moves so that the two axes are aligned.



Distance by faces, Angle by faces and Angle by axes

Make sure to create sufficient axes to setup for these modes.

Make two axes of two bodies coincident, then select one of modes by using hot keys or local menu

Distance by Faces

Select the Distance by faces mode.

Define the first face on the body which will be moved.

Define destination face

The body will be moved by the distance between selected faces along the common axis and two faces will become coincident.

Angle by Faces

1. Select face on body you want to rotate.

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Select the second face (on any body) you wish the first face to be parallel to.

If the rotation around the common axis is available, then the normals of the two defined faces will become coincident/parallel.

Angle by Axes

Mode 1:

Select the second axis on first body; the one you are moving.

Select the second axis on the body you are aligning to.

If the rotation around the common axis is available, the directions of selected axes will become coincident.

Mode 2:

Select the second axis on first body; the one you are moving.

Select the second axis on the body you are aligning to.

If the rotation around the common axis is available, the directions of selected axes will become coincident.

Assemble by Tangents

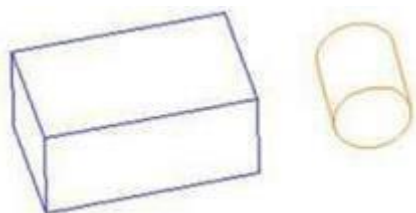
(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/3D Assemble/Assemble by Tangents

Ribbon UI Menu:

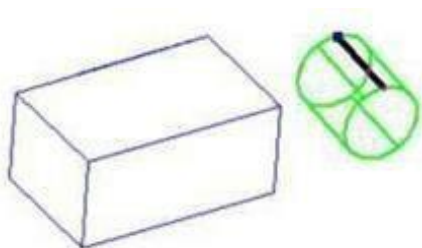
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Changes the position of an object by aligning a cylindrical face tangent to another cylindrical face, or to a flat face. This example will assemble a cylinder and a box.

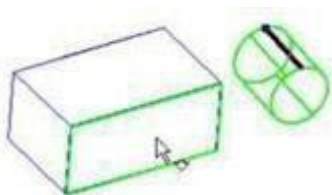


Aligning to a Face

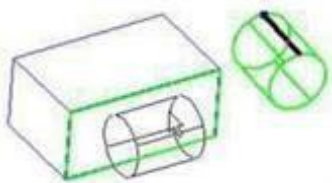
Activate **Assemble by Tangents** . Select the object that will move (the cylinder). Be sure to select the axis where you want the object to be placed on the second object.



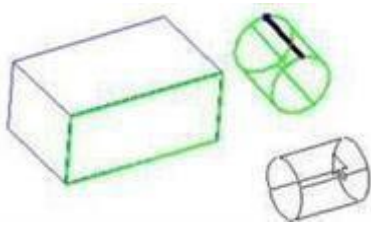
To align the object to a face, and not to a specific axis or point, click the face of the second object. This face can be planar or cylindrical. Where you click defines the initial location of the first object.



A preview of the moved object appears.



3. You can move the preview anywhere, and it remains tangent to the face.

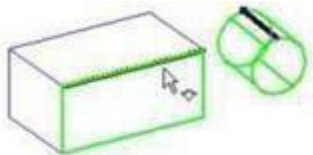


When you place the object, and view it in **Side** view, you can see that the cylinder is tangent to the face of the box.

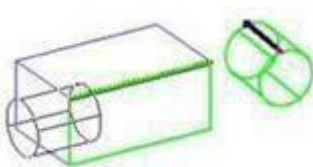


Aligning to a Face and Edge

You can align a cylindrical face to both a face and edge. When selecting the second face, click on one of its edges.

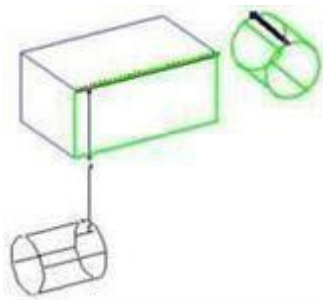


The preview shows the cylinder aligned to both the face and edge.

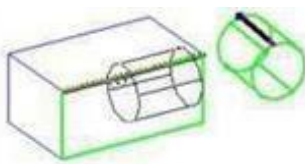


A **Distance to Edge** value will move the cylinder away from the edge, in an offset direction while staying tangent to the face.





3. A **Distance to Vertex** value will move the cylinder along the edge.



When you place the object, and view it in **Side** view, you can see that the cylinder is tangent to the face and edge of the box.



10.3. Bending and Unbending

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Bending

Ribbon UI Menu:

These tools are used for bending existing sheets and tubes (pipes) or for adding flanges on sheets and tubes. There are also tools for unbending.

These tools are available on the **Modify** toolbar, which you can display by right-clicking in any toolbar area and selecting **3D Modify**.



Bend Sheet

Default UI Menu: Modify/Modify 3D Objects/Bending/Bend Sheet

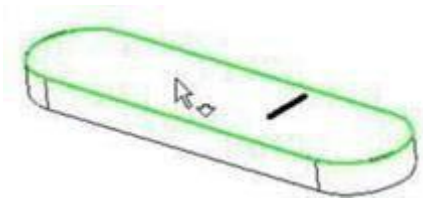
Ribbon UI Menu:

Bends a solid ACIS object, most commonly a sheet with thickness (Box).

Note: You can unbend the object using **Unbend Sheet**. See [Unbend Sheet](#)

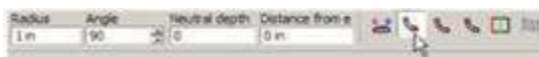
Start with a shallow **Box** or a **Simple Extrude** based on a polyline or rectangle. Add a line to the top.

Activate **Bend**. Select the face you want to bend.



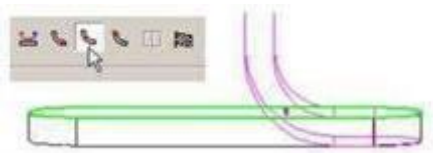
Then select the line about which you want to bend the solid. The line must lie on the solid face you selected.

Set the bending **Radius** and **Angle**. The default method is **Start**, in which the bend starts where the top face meets the selected line.

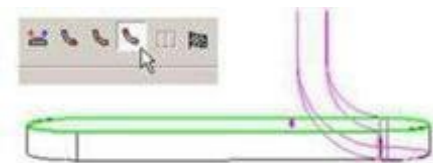


The preview shows how the final result will look.

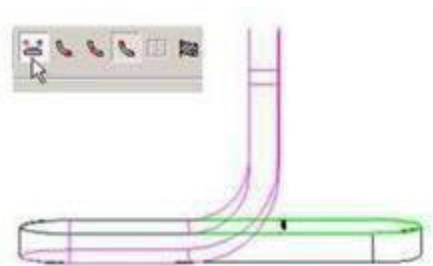
Switch to **Center**. In this case, the center of the bent solid meets the selected line (or its projection into the solid).



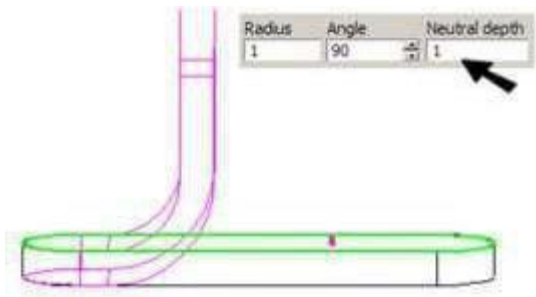
Switch to **End**. In this case, the bottom face of the bent solid meets the selected line (or its projection).



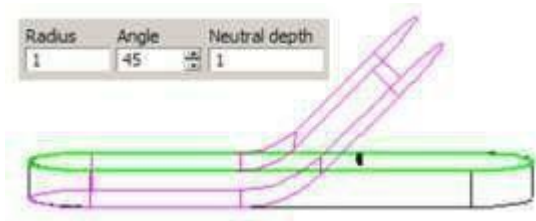
Select **Left Side**. The bend now starts from the other end of the solid.



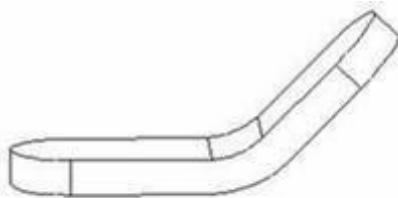
The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression.



9. The **Angle** is measured from the plane of the bending face.



When the parameters are set, select **Finish** from the local menu or Inspector Bar to complete the bend.

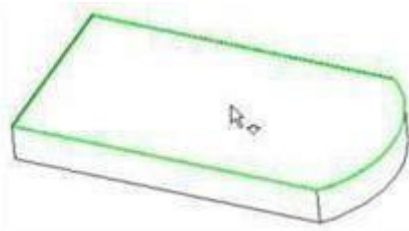


By Distance from Edge

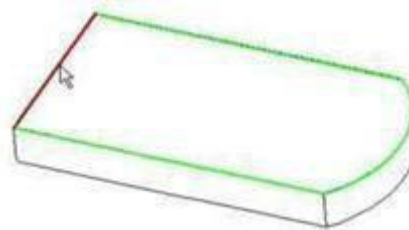
This option of the **Bend** tool enables you to define where the bend starts from a linear edge of the planar face.



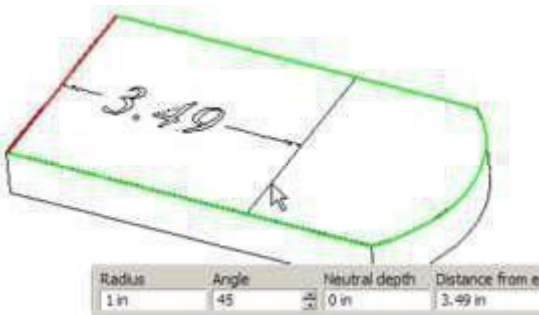
Start with a solid that has at least one linear edge. The first step is to select the face that will bend.



2. Then select the linear edge from which the bend distance will be defined.



Pull the cursor from this edge to set the bend distance. Once you click to set the distance, you can change it by clicking again. You can also set the distance in the Inspector Bar.



4. The preview shows how the final result will look.

Bend to Path

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(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Bending/Bend to Path

Ribbon UI Menu:

This tool bends a ACIS solid to conform to a 2D path created from a polyline, arc, or curve. The 2D entity must not be closed.

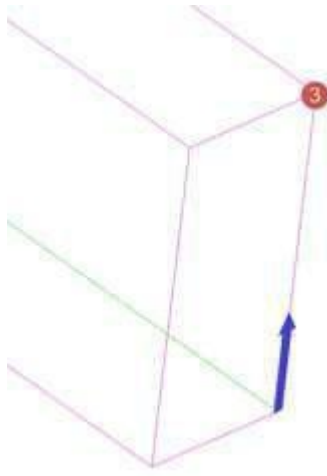
To use the Bend to Path tool:

Select the Bend to Path tool.

Select the solid to be bent.

Specify the Inside/Outside axis by selecting a first point and then a second point. This axis defines the line along the object that will be bent around the path.

Specify the initial direction of bending by selecting a point or by entering an angle value into the Inspector bar. The direction is indicated by a blue arrow. This point and the initial point of the inside/outside axis combine to create a bend axis which defines the orientation of the object as it is bent around the path. After creation this value can be changed via the Properties in the selection info palette if the Part Tree is turned on (Highly Recommended).



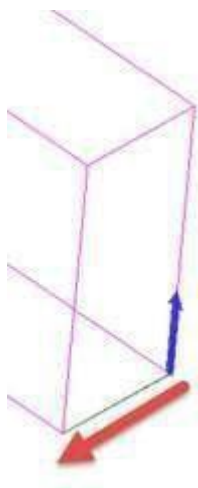
5. Select the path to that the object will be bent around.

The result:

If the inside/outside axis had been diagonal...

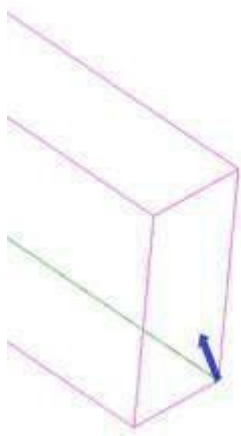
This would have been the result:

If the inside/outside axis had be drawn along the base...



This would have been the result:

If the bend axis had been defined diagonally...



This would have been the result:

Options:

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Leave source copy: When this option is selected a copy of the original object will be left in the model. It can be toggled via the Local menu or the Inspector bar.

Bend by Sketch

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Bending/Bend by Sketch

Ribbon UI Menu:

Extends and bends a sheet body (a flat object) along a selected sketch defined by a polyline.

Note: The polyline must begin on the initial edge of the sheet body. In addition the sketch must be on the plane which is perpendicular to the initial edge. Finally, the polyline must be comprised of only line segments. To Bend by Sketch

Select the tool.

Select the sheet body.

3. Select the Sketch.

Hold down the SHIFT key and click on the additional edges which should be included.

Specify the values for the tool options.



Click Finish.

Options

Bend Radius: Set the internal bend radius at each vertex.

Bend Offset: Sets the value for how far the edges of the bend will be offset from the end of the initial edges. This value is ignored when mitering is in use.

Miter Gap Width: Set the width of the gap between mitered sections when mitering is used.

Cut Co rners: Clips the corner back to the offset.

Auto Miter: Causes adjacent bending to meet at a miter

Flange

Default UI Menu: Modify/Modify 3D Objects/Bending/Flange Sheet

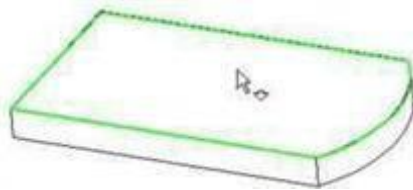
Ribbon UI Menu:

Adds a bend onto the edge of a solid (ACIS) object, most commonly a sheet with thickness (Box).

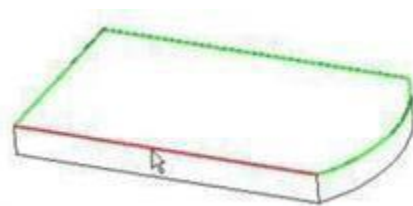
Note: You can unbend a flange using **Unbend Sheet**. See [Unbend Sheet](#)

Start with a shallow **Box**, or a **Simple Extrude** based on a polyline or rectangle.

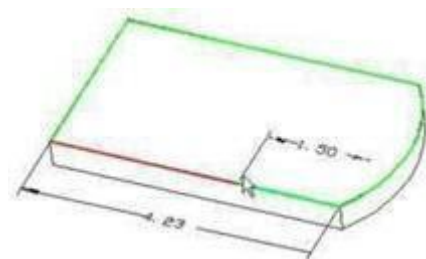
Activate **Flange**. Select the face to which you want to add the flange.



3. The next click defines the linear edge about which the flange will bend.

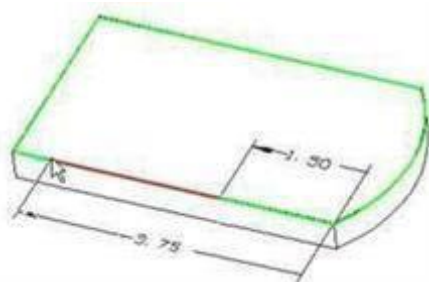


The next click defines how far from one end the flange will start. This distance can be zero. The total length of the edge is also indicated.



5. The next click defines how far from the other end the flange will end.

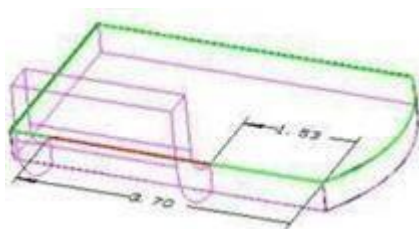
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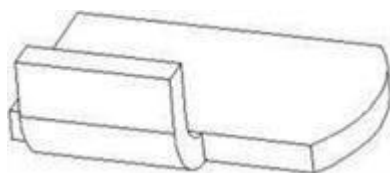
These distances are **Start flange position** and **End flange position**, which can be set in the Inspector Bar as well. You can also define the **Radius** and **Angle** here.

Radius	Angle	Neutral depth	Flange height	Start flange pos.	End flange pos.
0.5 in	90	0.25 in	2 in	1.53 in	3.7 in

The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression. **Flange Length** is the length of the new material beyond the edge. The preview shows how the final result will look. You can still make changes to all values in the Inspector Bar.



7. Select **Finish** from the local menu or Inspector Bar to complete the flange.



You can use this tool multiple times on the same sheet.

Sheet Metal

Following options are now available in TurboCAD

Taper Angle: Allows to create a flange with one or two tapered side faces

Offset distance dStart and dEnd.

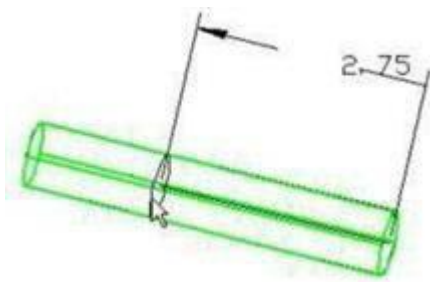
Tube Bend

Default UI Menu: Modify/Modify 3D Objects/Bending/Tube Bend

Ribbon UI Menu:

Bends a solid (ACIS) cylinder.

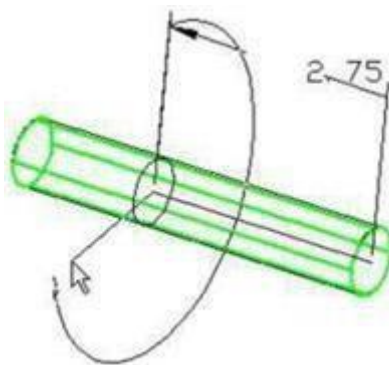
Start with a cylinder, and activate **Tube Bend**. Select the cylinder you want to bend.
The next click defines the distance where the bend will start.



This distance is the **Axial distance**, which can be set in the Inspector Bar as well. You can also define the **Radius** and **Angle** here.

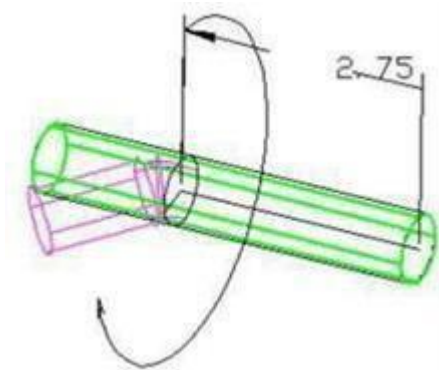
Radius	Angle	Neutral depth	Axial distance
0.1 in	45	0.37 in	2.75 in

The next click defines the direction of the bend, relative to the axis of the cylinder.



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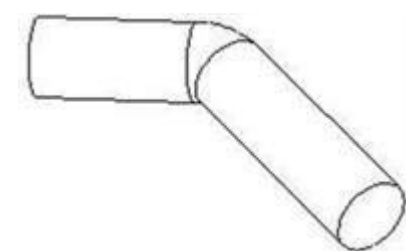
The preview shows how the final result will look. You can still make changes to all values in the Inspector Bar.



Click **Left Side** if you want to bend the cylinder from the other end.



Select **Finish** from the local menu or Inspector Bar to complete the bend.



You can use this tool multiple times on the same tube.



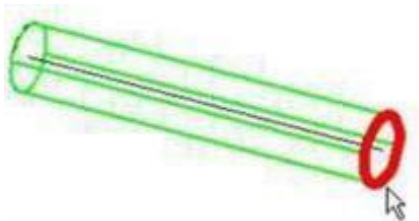
Tube Flange

Default UI Menu: Modify/Modify 3D Objects/Bending/Tube Flange

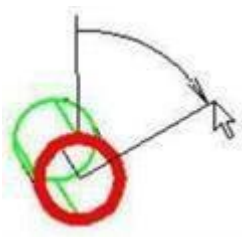
Ribbon UI Menu:

Adds a bend onto a solid (ACIS) cylinder.

Start with a cylinder, and activate **Tube Flange**. Select the cylinder you want to bend. The next click defines the end where the flange will be added.



The next click defines the, which defines the direction of the bend, relative to the axis of the cylinder.

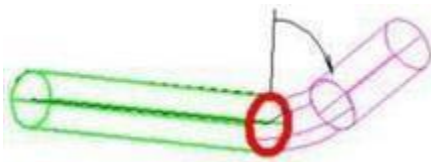


This angle can be set in the Inspector Bar as well. You can also define the **Radius** and **Angle** here.

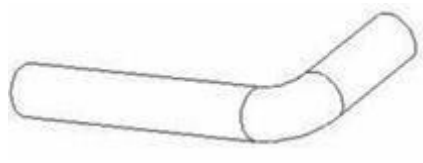
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Radius	Angle	Neutral depth	Flange height	Arzimuth angle
0.25 in	90	0.37 in	2 in	60.97

The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression. **Flange Length** is the length of the new material beyond the current end of the cylinder. The preview shows how the final result will look. You can still make changes to any value in the Inspector Bar.



Select **Finish** from the local menu or Inspector Bar to complete the bend.



You can use this tool multiple times on the same cylinder.



Unfold Face

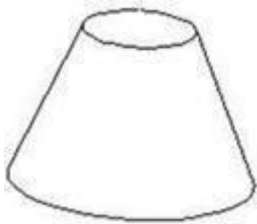
Default UI Menu: Modify/Modify 3D Objects/Bending/Unfold Face

Ribbon UI Menu:

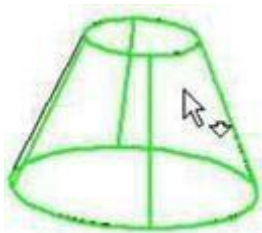
Unfolds faces of an ACIS object. This is also known as obtaining the involute of a surface.

Example 1

For a simple example, start with a truncated cone, using the **Cut Cone by Plane** option.



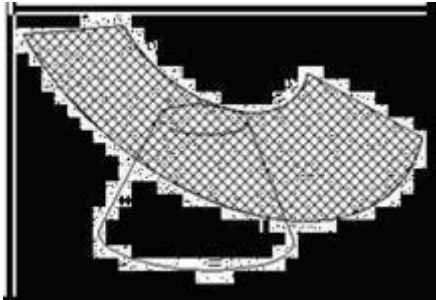
Activate **Unfold Face**. Click when the curved face is highlighted. Only faces that can be unfolded will be highlighted.



3. A preview of the unfolded face will appear.

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4. To create the face, select **Finish** from the Inspector Bar or local menu.



Example 2

5. For a more complex example, start with a **Rectangle**, filleted using the **Fillet** tool.



6. Create a cone from this polyline, using the **Specify 2D Base** option.



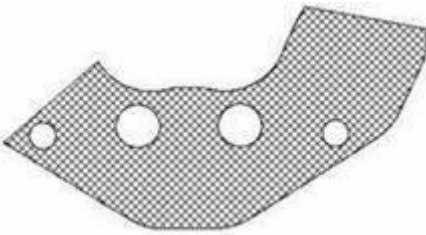
7. Create some cylinders and use **3D Subtract** to remove them from the cone.



8. Activate **Unfold Face** and click the cone.



9. Select **Finish**. If you move or delete the cone, you can see the unfolded face clearly



Unbend Sheet

Default UI Menu: Modify/Modify 3D Objects/Bending/Unbend Sheet

Ribbon UI Menu:

This tool unbends a sheet that has a bend or flange.

It is most commonly used on objects that have a Bend or Flange added (see [Bend Sheet](#) and [Flange](#)) or on objects imported into TurboCAD.

It may be used on bends created via filleted corners if the difference in the Radii of the inner and outer fillet equals the depth (thickness) of the material.

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Start with a sheet that has a bend and/or flange. This example has a sheet with a cylindrical imprint, a bend with two holes, and a flange with one hole.

Activate **Unbend Sheet**. The **Neutral Depth** is the distance into the depth of material, along which there will be no tension or compression. Changing this parameter will very slightly affect the resulting sheet size.

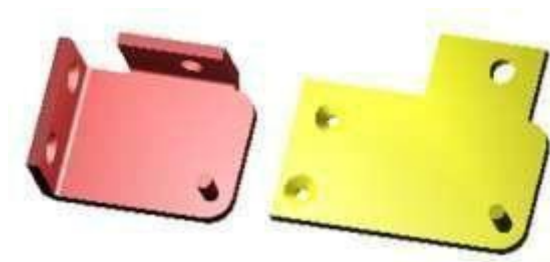


Select the face that will be used as the base for unbending. The resulting flat sheet will be flushed with this face.

4. A preview of the resulting sheet will appear.

Select **Finish** from the local menu or Inspector Bar. The new, unbent sheet is created, overlapping with the base face.

6. To see the results, you can move the unbent sheet, or delete the original sheet.



There are several options available for the Unbend Sheet tool.

Leave Source Copy: if this option is on, the tool leaves the original bent object in place and creates a new unbent object.

Get Bend Lines: If this option is on, the tool creates the bend lines that were necessary to create the object. This allows for re-bending if required.

Get Bend Angle: If this option is on, the tool creates the text representing the bend angle that was, necessary to create the object. This allows for re-bending if required.

Unbend along edge: If this option is on, you can individually select edges to be unbent. If this option is off the Unbend tool will unbend everything on the object at once.

Show Labels: Shows labels designating the order in which edges have been selected for unbending.

Progress Indicator

You can trace the progress of Unbending by progress indicator.

Press "esc" to interrupt the process while building a preview.

Greater accuracy in the "Unbend" results due to using ACIS® unbend at each step.

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Demo Video: Please see below detailed video about the improvements made in TurboCAD 2023 regarding Unbend functionality.

https://youtu.be/k8X2074_yAw

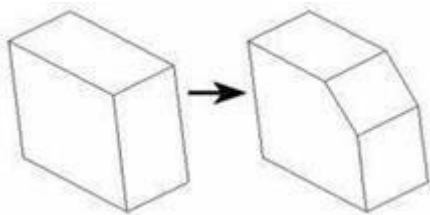
10.4. Chamfer Edges

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Modify Edges/Chamfer Edges

Ribbon UI Menu:

Chamfers (bevels) edges of solid 3D objects.



Only solid objects (not surfaces) can be selected for chamfering. Chamfering can either add or remove material; material is added to a concave edge, and it is removed from a convex edge. You can choose whether to create rounded or mitered vertices.

Note: You can edit a **Chamfer Edges** operation in the Selection Info palette. See [Editing 3D Objects using Selection Info](#)

The chamfer size is set by the **Offset** and **Chamfer Angle** values. Offsets are the distances from the edge. If the **Offset** values are equal, the **Chamfer Angle** is 45 degrees.

Offset 1	Offset 2	Chamfer Angle
3 in	3 in	45

If you change one **Offset** value, the **Chamfer Angle** updates accordingly.

Offset 1	Offset 2	Chamfer Angle
3 in	2	56.31

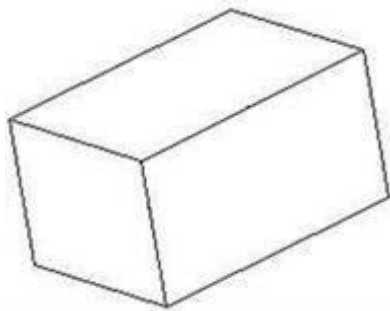
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Similarly, if you change the **Chamfer Angle** , the **Offset 2** value will update accordingly.



Chamfer with Miter Vertices

Mitering the vertices produces chamfered edges that meet at an edge at the corners. This example will use the following model.

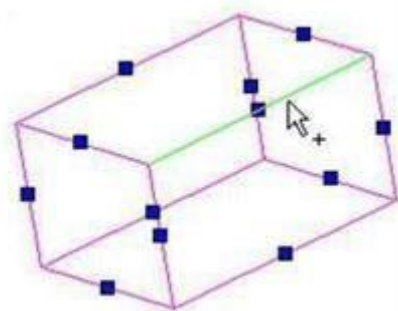


Make sure **Round vertex** is disabled.

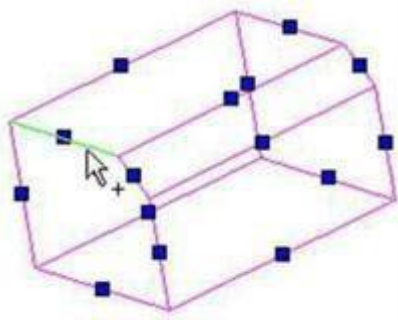


Select the object whose edges are to be chamfered. Each edge is marked with a blue square.

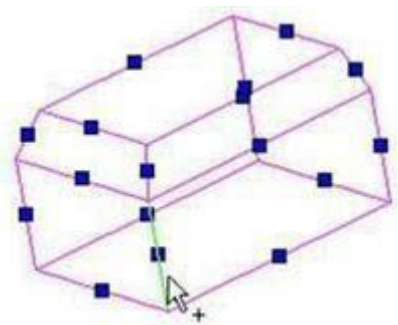
Set the chamfer offset values in the Inspector Bar, and select the edge to chamfer.



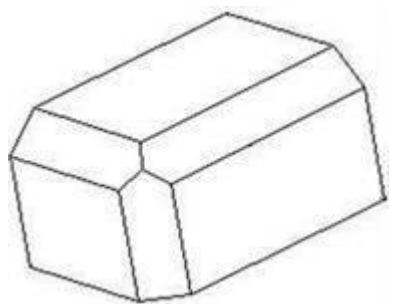
The edge is chamfered. To chamfer another edge, select the object again and select the next edge.



5. Continue to select edges to chamfer.



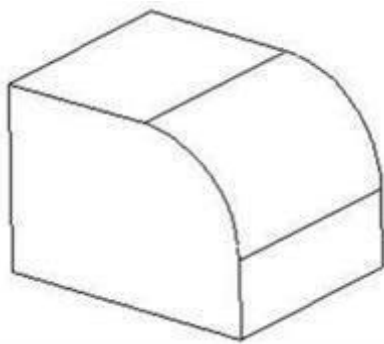
Any common corners will meet at a sharp edge.



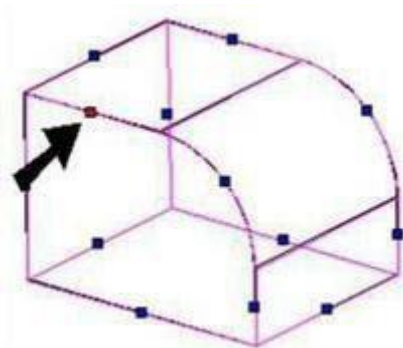
Pick Smooth Sequence

Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex. When working with **Round vertex**, smoothly connected edges are automatically selected. But this is not the case when **Round vertex** is turned off.

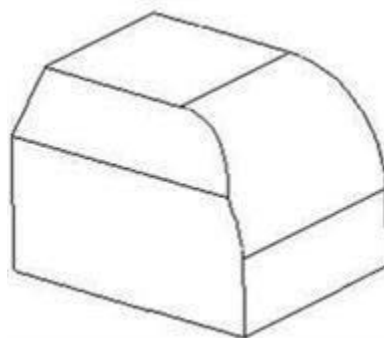
1. Start with a box with one filleted edge, using a large radius.



Activate **Chamfer** without **Round vertex**, and chamfer one edge, using a smaller offset.



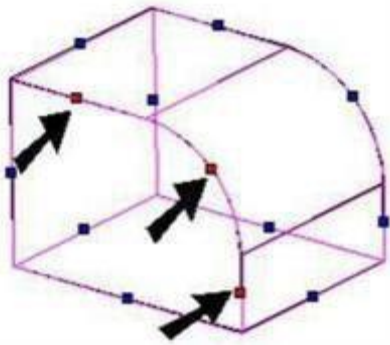
This is the result - only the selected edge was chamfered.



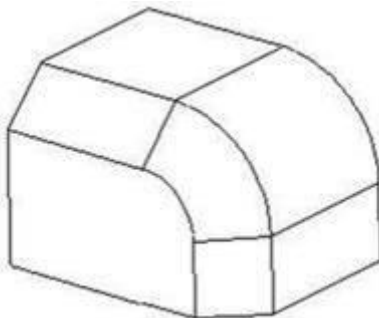
Undo, and this time select **Pick smooth sequence**.



This time when you select one of these edges for chamfering, all edges in the smooth chain are selected.

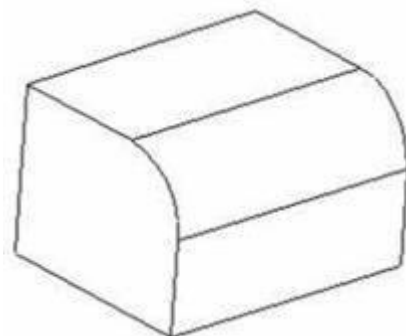


This is the result - all edges in the chain are chamfered.



Chamfer with Round Vertices

Rounding the vertices produces a smooth rounding along corners. This example will



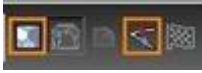
use the following model, with one filleted edge.

1. Select **RoundVertex** in the Inspector Bar or on the local menu.

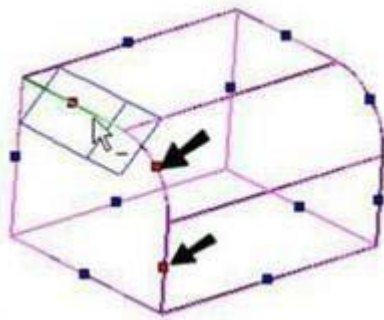


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Select the object whose edges are to be chamfered. Each edge is marked with a blue square. If you select **Hide Marks**, these edge markers will be hidden.

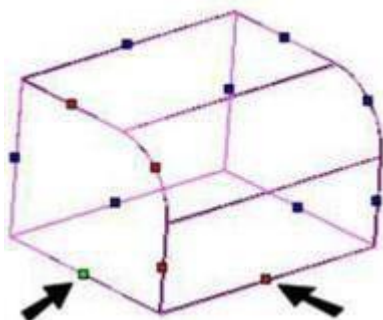


Select the first edge to chamfer. When an edge is selected, the blue square turns red. A preview appears indicating the chamfer size. If you select an edge that is smoothly connected to other edges, the connected edges will be selected automatically.

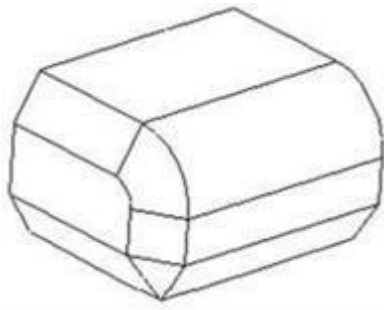


Note: Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex.

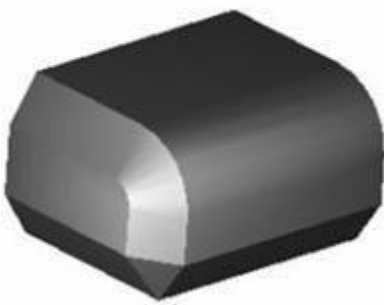
Continue selecting edges, if needed. (You can also drag a window to select multiple edges). In some cases, selecting an edge (such as the bottom front edge) will cause another edge to be chamfered as well (the bottom side edge), because they meet at a common vertex. These edges are indicated by a green square.



When all edges are selected, set the chamfering offsets in the Inspector Bar, and select **Finish** from the Inspector Bar or the local menu.



Viewing the results in render mode shows the affect of chamfering. The common vertex is smoothly rounded.



10.5. Editing 3D Objects using Selection Info

Note: For general information on this palette, see [Selection Info Palette](#)

You can use the Selection Info palette to edit properties and parameters of 3D objects, taking into account the order in which the objects were created. Objects must be created as solids; surface objects cannot be edited this way (see [3D Properties](#)).

In addition to seeing what objects were used in which operations, you can also edit certain properties of these operations:

Fillet Edges and Chamfer Edges: add or subtract affected edges, modify the chamfer length or fillet radius.

Boolean operations: Change the size or location of the original objects combined in the Boolean operation.

Shell: Change the open faces, set inward or outward shelling, modify the shell thickness.

Imprint: Change the type of imprint, add a draft angle, and change the fillets and chamfers at the edges. See [Editing Imprint Parameters](#)

After you make a change to any operation, all subsequent operations will update to reflect the change.

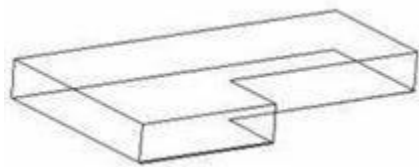
Warning: Facet Deform operations cause the edited objects to be deleted from the editing history.

In order for object history to be recorded, you must check **Create Editing History** in the **ACIS** page of **Drawing Setup (Options / ACIS)**.

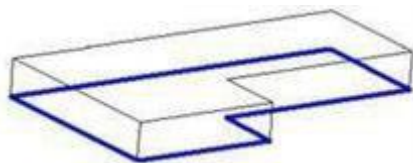


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This example started with a Simple Extrude, based on a linear polyline. Make sure that the **Use compound profile** option is turned on, otherwise the profile will not be linked to the resulting extrusion.



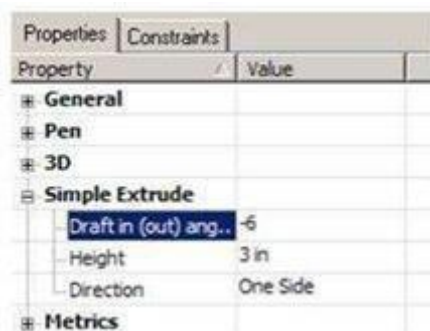
2. Highlight Polyline and change its color and line weight.



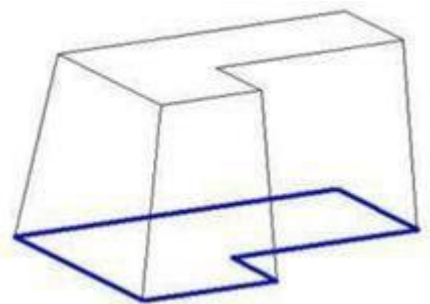
Now the polyline is easily differentiated from the solid.

For the Simple Extrude, you can edit all parameters that are available in its **Properties** window. For example, the **3D** category has an option to change the object into a surface object and to set the object's **Material**.

Under **Simple Extrude**, increase the **Height** and add a slight **Draft In (out) Angle**.

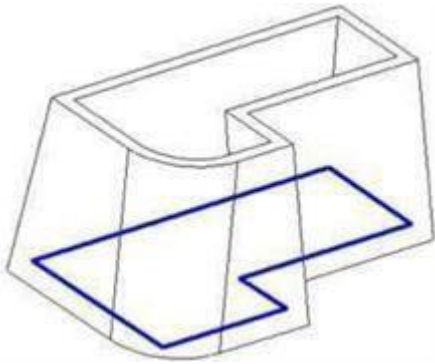


The solid is now higher with drafted walls.

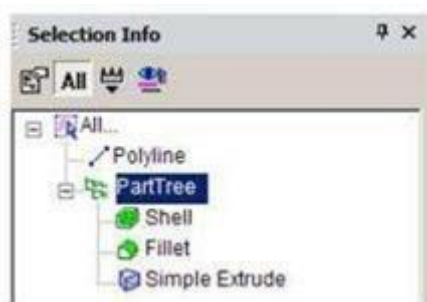


Use **Fillet Edges** to round one edge (see [Fillet Edges](#).) Then use **Shell Solid** to shell this shape, leaving the top face open (see [Shelling Solids](#).)

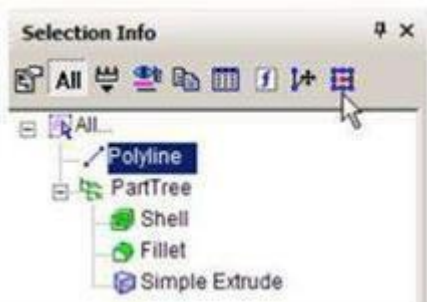
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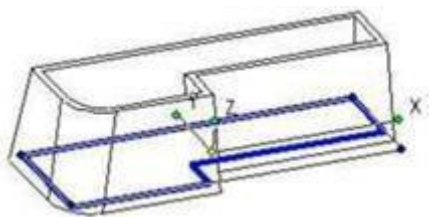
In Selection Info, the Extrude, Fillet and Shell operations are listed under "Part Tree."



The Simple Extrude is based on the polyline, and changing the polyline will update the solid. Highlight Polyline and click **Select**.



Scale the polyline in Select Edit mode to make it longer. The solid shape, as well as the fillet and shelling update automatically.



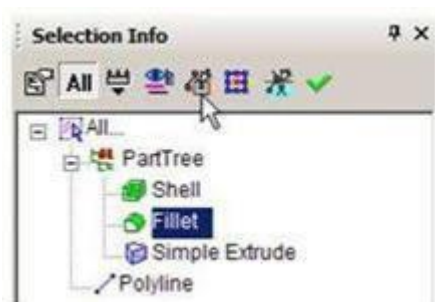
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Subsequent operations do not have to update automatically. To disable this, right-click on "Shell" and select.

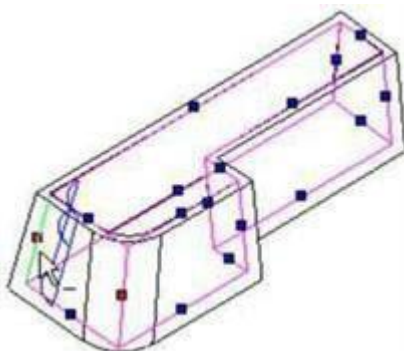


Note: If you click **Suppress Operation**, the operation will be removed from part history, though its item will still remain in the list. You can bring the operation back into the model by selecting **Unsuppress Operation**.

8. To edit the fillet, highlight it and select.

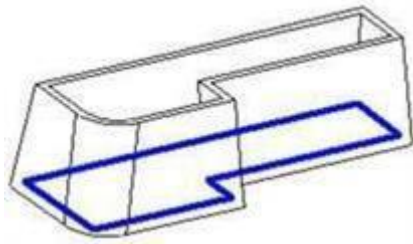


Edit in Place enables you to return to the original operation and make changes. Click another edge for filleting. You could also change parameters such as the fillet radius.

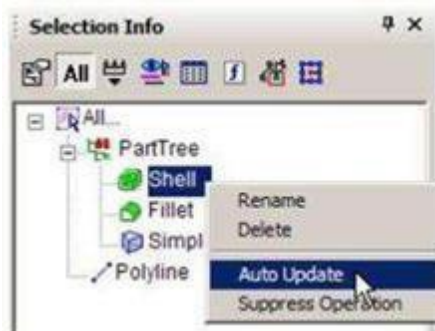


10. Select **Finish** to update the operation. The shell has not updated.

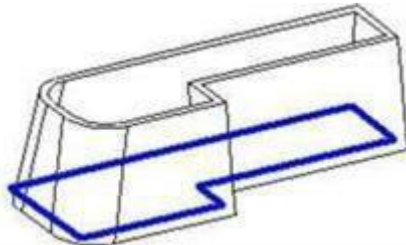
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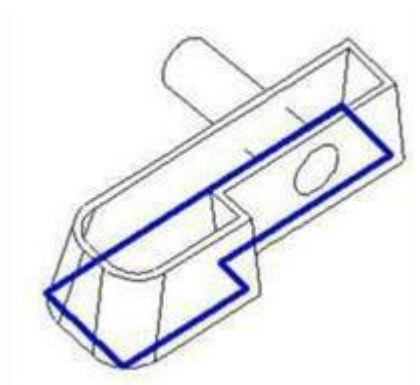
11. To update the shell, right-click it and select.



Now the shell reflects the new filleted edge.

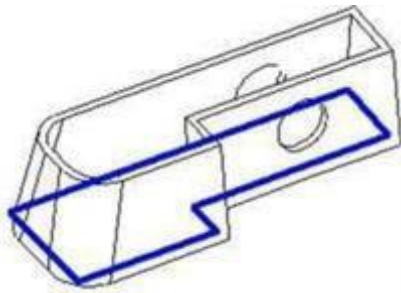


For the next operation, create a cylinder, with its workplane located on one of the vertical faces.

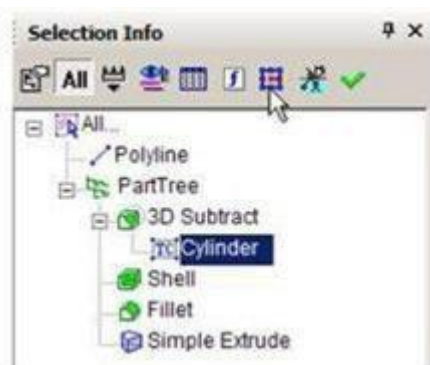


Then use the Boolean operation **3D Subtract** to remove the cylinder from the shelled solid. (See [3DSubtract](#).)

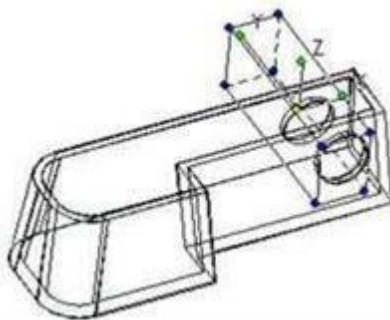
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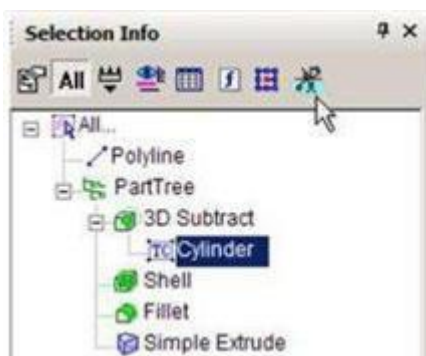
The **3D Subtract** operation is now listed. If you expand it, you can see the **Cylinder** on which it is based. Highlight the cylinder, and click **Select**.



15. Move the cylinder, which in turn updates the 3D Subtract operation.

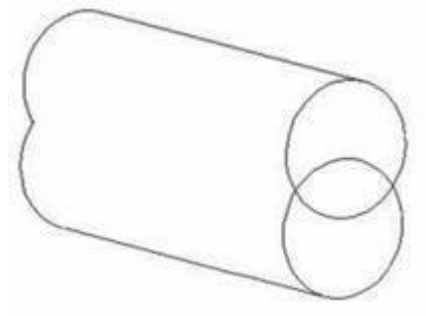


16. For another way to edit the cylinder, select **Edit Content**.

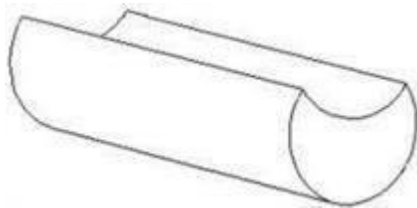


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The cylinder is now the only object in the drawing area, and any changes will modify the operations currently based on the cylinder. Make a copy of the cylinder so that they overlap.



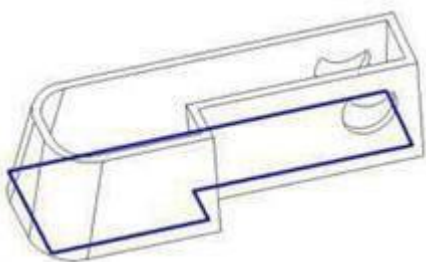
18. Then use **3D Subtract** to get this shape:



When finished, click **Finish Editing Content**.



The entire solid is updated, based on the modified cylinder.



For one of the solid operations, open **Metrics**. It only informs you that the object is a solid, as opposed to a surface.

21. Click Show Physical Metrics.



This displays properties such as volume, area, center of gravity, and moments of inertia.

TC Mesh Simplification

(Available only in Platinum)

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Default UI Menu: Modify/Modify 3D Objects/TC Surface Simplification

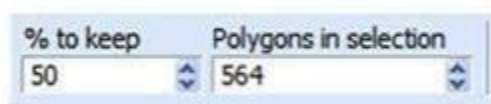
Ribbon UI Menu:

Simplifies meshes by reducing the total polygon count. For example you can use it for reducing the number of the polygons in the laser scanned model (e.g. from 400000 to 4000 triangles).

TC Surface Simplification enables when a 3D Boolean operation has been performed on two or more SMeshs.

1. Select a mesh or TC Surface.

Select the Percent to Keep.



Click the **Finish** button or select **Finish** from the local menu.

Local Menu Options

Ignore Boundaries, Contract Boundaries at end, Fix Boundaries: this switch tells the simplifier how should it process the models boundaries (boundary = set of the edges where each edge belongs to the one triangle only)

Do Full Update before simplification: you should use this setting if the simplifier fails or its result is incorrect. Usually it means that the simplifier's input model was incorrect. You may try to heal the model by using "Do Full Update" in this case.

XClip



(Available in all TurboCAD Variants)

Default UI Menu: Modify/Clip/XClip

Ribbon UI Menu:

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The XClip tool creates a cropped/clipped display of a selected external reference or block reference based upon a selected boundary. You can use any circle, or closed polyline consisting of only straight segment as a boundary.

1. Select an xref or a block or a group of xrefs or blocks.

2. In the local menu select the Rectangular/Polygonal option.

3. Draw the desired cutting area.

4. Press Finish from Local menu or Inspector bar.



In Local Menu, If you click the "Generate Polyline" option, it will create a polyline of area you have cutted.

A cropped version of the xref or block is created. the original xref or block insertion is destroyed. The xclip and the selected boundary are not associative, so updating the boundary does not update the xclip. If the xref/block contained 3D object these objects are shown as "hollowed" whether they are surfaces or ACIS solids. Xclipping does not create new geometry for the clipped entities so the missing faces are simply not displayed. In this regard xclips are not like booleans. Regardless of the current UCS, the clip depth is applied parallel to the clipping boundary.

XClip Properties

XClip properties provide added control for how xclips are displayed.

Display only result: if this option is unchecked the clipping boundary will be ignored, and all of the geometry of the clipped blocks or xrefs will be displayed.

Enable front clip: If this option is selected the xclip will clip everything in the clipped entities above a specified height. Front clipping always occurs parallel to the original clip boundary.

Front Clip: Sets the height for front clipping.

Enable back clip: If this option is selected the xclip will clip everything in the clipped entities below a specified height. Front clipping always occurs parallel to the original clip boundary.

Back Clip: Sets the depth for back clipping.

In the following picture the the xclip has a Front Clip of 12in. and a Back Clip of 1in.

10.6. Facet Deform

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Facet Editing

Ribbon UI Menu:

These functions enable you to change the shape of a surface or solid face by applying a uniform pressure, or by specifying deformation points and constraints.

Facet Deform works only with ACIS solids and surface. It does not work on TurboCAD surface objects, objects that have **TC Surface** selected in the **3D** page of the **Properties**.

See [3D Properties](#)

Warning: **Facet Deform** causes the edited object to be deleted from the Part Tree. See [Editing 3D Objects using Selection Info](#)

Deform to Point

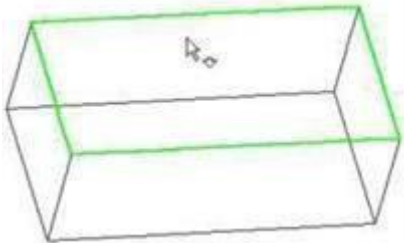


Default UI Menu: Modify/Modify 3D Objects/Facet Editing/Deform to Point

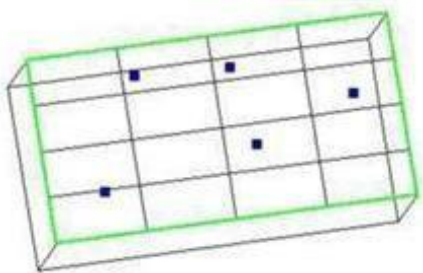
Ribbon UI Menu:

Enables you to deform a face or surface by moving one or more points on the face. The result is a NURBS surface or solid.

Activate the **Deform to Point** function and select the face you want to deform - in this case, the top face of the box.



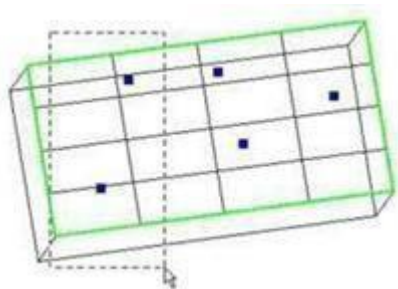
The next step is to define deformation points and, if necessary, constraint points. You can click anywhere on the face to create points. If you created points in advance, use the **Vertex** snap to select them.



Any of these points that are not moved will be used as constraints; they will hold the face in place.

Now select one or more points that will move. You can select points individually, or use a selection box.

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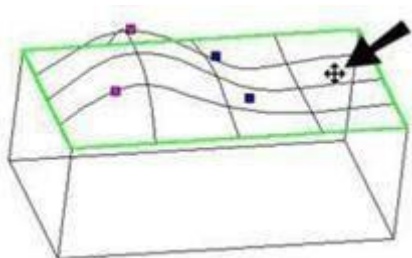


Once points are selected, you can move them manually by dragging their reference point, or enter delta values in the Inspector Bar. In this example, the two selected points will move up in the Z direction.

Stretch	Bend	Resolution	Isolines	DeltaX	DeltaY	DeltaZ
2	2	6	4	0 in	0 in	0.35

Note: For explanations of the other parameters (**Stretch**, **Bend**, etc.), see [Pressure Load](#)

Press Enter to move the points, and the face preview is updated. If necessary, you can now move other points. In this case, a single point is selected (rather than using a selection window).

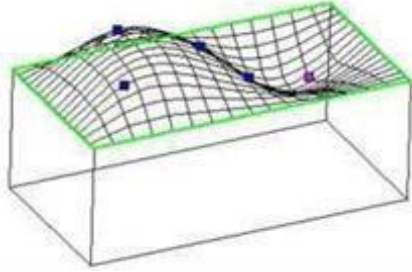


6. This point will move in both Y and Z.

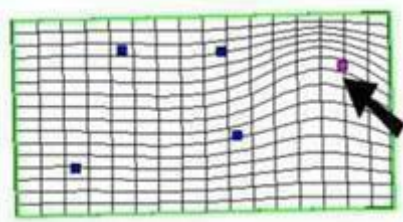
Stretch	Bend	Resolution	Isolines	DeltaX	DeltaY	DeltaZ
2	2	6	4	0 in	0.15	-0.5

The preview now shows all three moved points. The two points that were not moved are used to hold the face in place. You can increase the number of isolines to better visualize the deformed face.

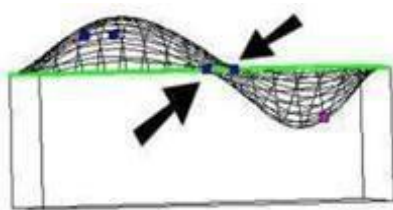
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In this view, you can see the movement in Y of the single point shown.



And in this view, you can see the effect of the constraint points, those not selected to move.



Local menu options:

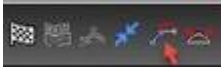
Note: For **Linked neighboring faces** and **Fix Tangent**, see [Pressure Load](#)

Delete constraint: Deletes the selected point or points. The deformation or constraint effect on the face will also be removed.

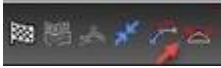


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One Step Back: Use this if you want to change delta values for the points just moved.



Move Along Normal: Moves the points in a direction normal to the surface. In this case, you only need to specify a single **Direction** value.



Pressure Load



Default UI Menu: Modify/Modify 3D Objects/Facet Editing/Pressure Load

Ribbon UI Menu:

Enables you to deform a face or surface by applying a uniform pressure over the entire face. The pressure is always normal to the face. The result is a NURBS surface or solid.

Activate the **Pressure Load** function, and set the pressure parameters (defined below).



Stretch: A factor controlling elasticity - the resistance of the face to stretching. The higher the value the higher the resistance, resulting in less deformation.

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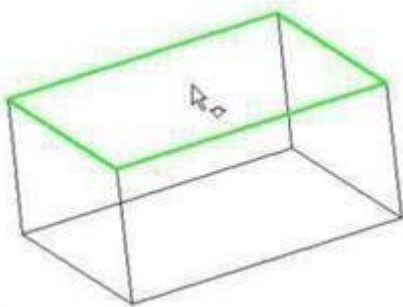
Bend: A factor controlling the resistance of the face to bending (flexure) and torsion. Higher values prevent sharp changes to the face.

Resolution: The number of internal grids (knots) used to control the deformation. The higher the value, the greater the influence of **Bend** and **Stretch** factors. Lower values, however, yield faster results.

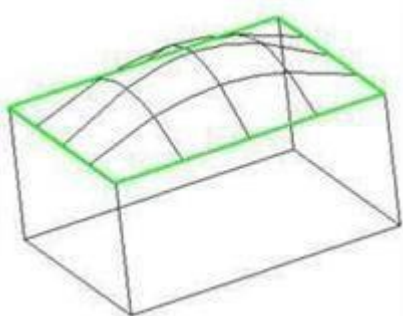
Isolines: Number of display lines. This does not affect the results, but is used to better visualize the deformed face.

Gain: The pressure value. A gain of 1,000 on a 10 x 10 face will have more of an effect than the same gain on a 100 x 100 face. Negative values can be used.

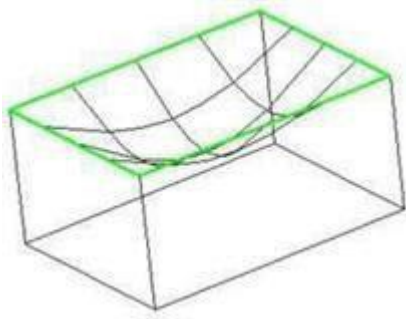
Select the face or surface you want to deform. In this example, the pressure will be applied to the top face of a box.



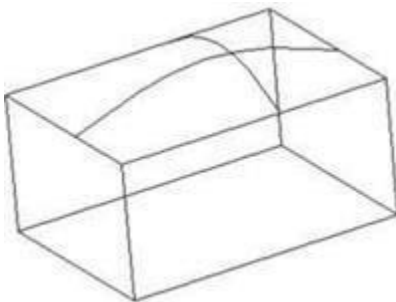
A preview of the deformed face will appear, according to the parameters you set. You can adjust the values and press Enter to update the preview. If you need a better representation of the deformation, increase the number of isolines. A positive **Gain** value pushes the face up.



If you switch the **Gain** to a negative value, the face will be pushed inward.

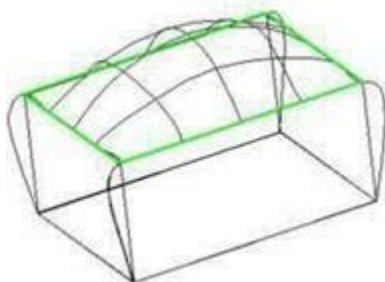
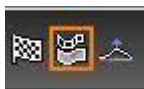


When you are satisfied with the deformation, select **Finish** from the local menu or Inspector Bar.

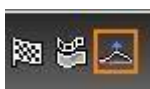


Local menu options:

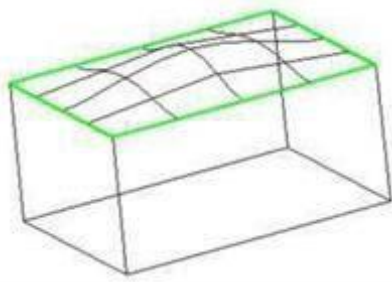
Linked neighboring faces: Adjacent faces will also be deformed.



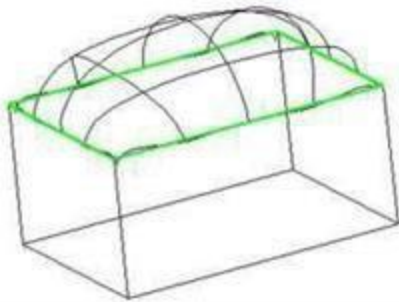
Fix tangent: Tangency at the face boundaries will be maintained. In this example, the face will remain horizontal along its boundaries.



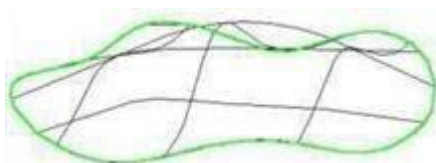
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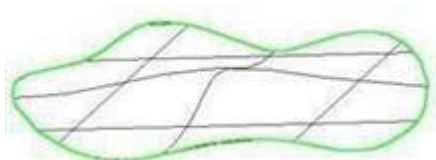
You can combine **both** local menu options to produce a deformation that is tangent to adjacent faces, with minimal deformation of the neighboring faces..



You can also deform a stand-alone surface such as this one, created by using **Create Surface from Profile** on a closed spline. (See [Create Surface from Profile](#) .)



In this case, the local menu option **Linked neighboring faces** is not available. If you select **Fix Tangent**, the face remains horizontal along its boundary.



10.7. Facet Edit

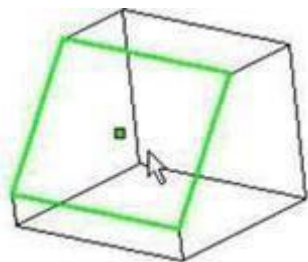
(Available in Platinum only)

Default UI Menu: Modify/Modify 3D Objects/Facet Editing/Facet Edit

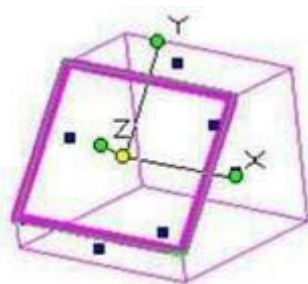
Ribbon UI Menu:

Enables you to move or rotate individual facets of a solid (not surface) object.

Select the facet you want to edit. Before it is selected, the facet is highlighted in green and indicated by a green square at its center.

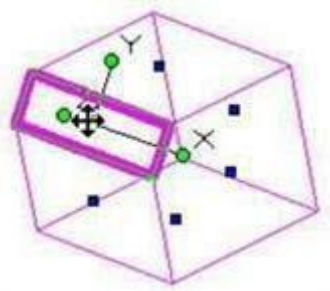


When the facet is selected, all its other facets are indicated by a blue square. You can select another facet for editing at any time while the tool is active. The selected facet appears similar to **Select Edit** mode, with rotation handles for the three axis (relative to the facet plane), and a yellow reference point for moving.

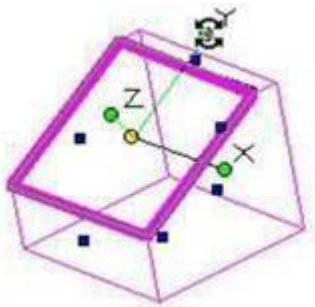


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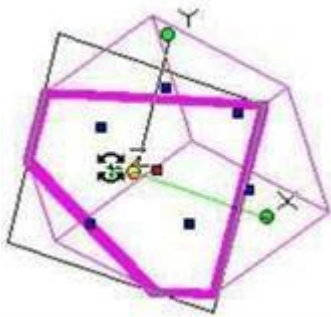
2. Drag the facet by its reference point to move it normal to its current plane.



3. You can also rotate about X or Y to change its angle relative to adjacent facets.

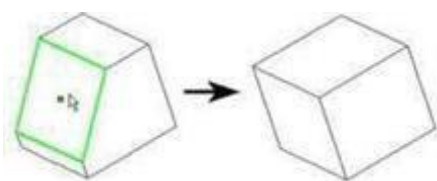


4. You can rotate around Z to "tilt" the facet sideways.



Lastly, you can delete a node by selecting it and pressing the Delete key. This is possible only when the ACIS object can be reconstructed without the facet.

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Note: If you want to delete a facet without reconstructing the object, you can convert the solid to a surface. See [Create Surface from Solid](#)

Inspector Bar options:

Properties: Opens the **Facet Editor Properties**. The options here are the same as for the 3D Selector. See [2D-3D Selector](#)



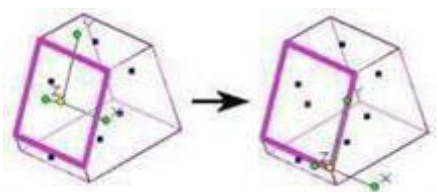
Make Copy: Copies the Selected Face



Edit Reference Point: Moves the reference point of the facet, located by default at the facet center.



Moving the reference point enables you to rotate the facet about different axes.



Note: You can also press the **D** key to select and move the reference point.

Default Reference Point: If the reference point was moved, this option moves it back to its default location at the center of the facet.

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Default Rotation Bars: If the Rotation bars were moved, this option moves it back to its default position.



UCS by Selector: This option selects the user coordinate system by selector.



Selector by UCS: This option sets selector by user coordinate system.



Lock Axis: This will lock the axis.



Toggle Facet / Feature:



Delete Face(s): Deletes the selected face



These Options can be accessed in **Local Menu:**

Select Feature or Defeature Part

Select a single face and the tool will select all similar classes of faces by blend networks, depressions, or protrusions of a part body. Combined with a remove face, this tool is useful to replace a feature from a imported part (no history) or to defeature a part.

Blend Network: Selects blends of edges/facets.

Blend Sequence: Selects single edge/facet

Depression: Selects depression of face(s)

Protrusion: Selects extension below or above a surface.

Note: *The Facet Editor has the ability to work in multiple, open drawing windows.*

10.8. Facet Offset

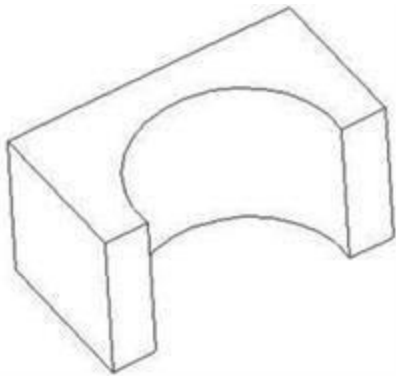
(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Facet Editing/Facet Offset

Ribbon UI Menu:

Extends a solid face inward or outward.

1. Start with a solid object. This example is a **Simple Extrude** based on a polyline.



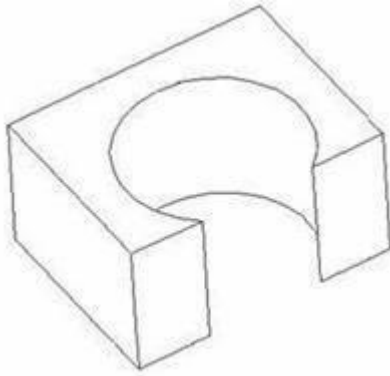
Activate **Facet Offset** and select the solid. Then select the face or faces you want to offset.

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Enter the **Offset** distance - a positive value will offset the face outward, and a negative value will offset inward. Select **Finish** to offset the faces.

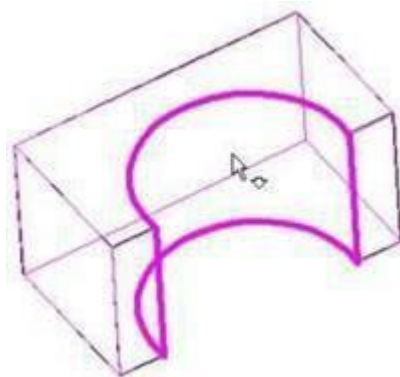


The faces are moved outward.



Note: If the **Offset** distance will produce a geometrically incorrect solid, or will make the solid disappear, the action will be ignored.

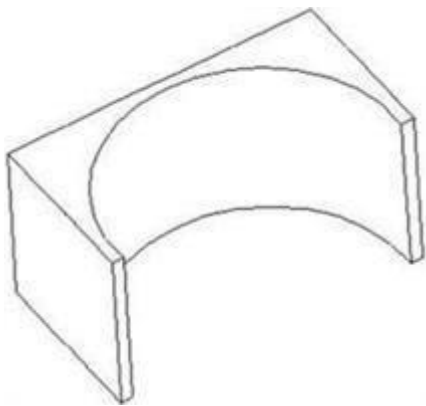
5. Offset faces do not have to be planar. In this example, select the curved face.



Use a negative **Offset** and select **Finish**.



The face moves inward.



If the Part History is activate, you can edit the offsets after they are created. See [Editing 3D Objects using Selection Info](#)

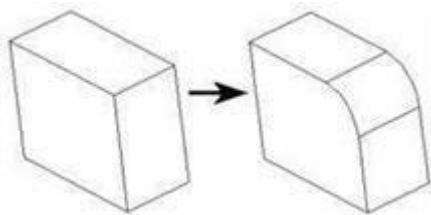
10.9. Fillet Edges

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Modify Edges/Fillet Edges

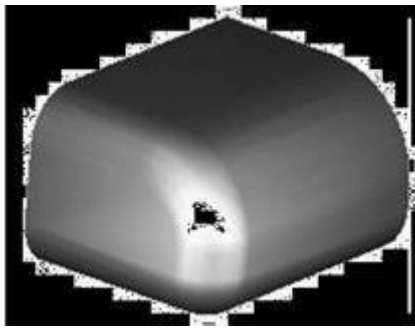
Ribbon UI Menu:

Rounds edges of solid (ACIS) 3D objects.



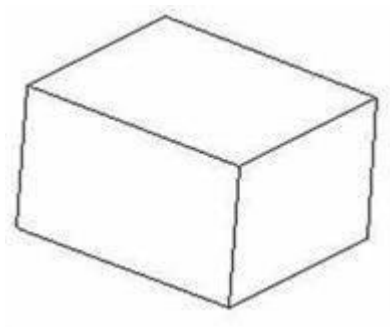
Only solid objects (not surfaces) can be selected for filleting. Filleting can either add or remove material; the material is added to a concave edge, and it is removed from a convex edge. You can choose whether to create rounded or mitered vertices.

Note: You can edit a **Fillet Edges** operation in the Selection Info palette. See [Editing 3D Objects using Selection Info](#)



Fillet with Miter Vertices

Mitering the vertices produces rounded edges that meet at an edge at the corners. This example will use the following model.

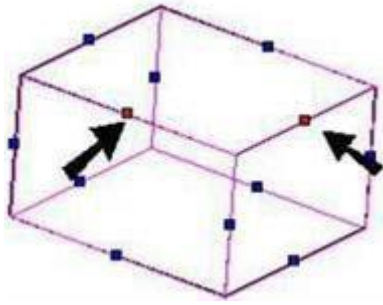


Make sure **Round vertex** is disabled.

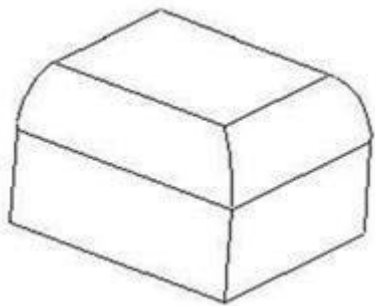


Select the object whose edges are to be rounded. Each edge is marked with a blue square.

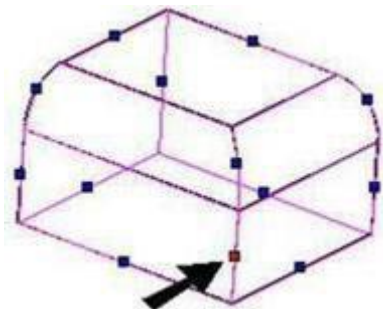
Set the **Start Radius** radius in the Inspector Bar, and select the edges.



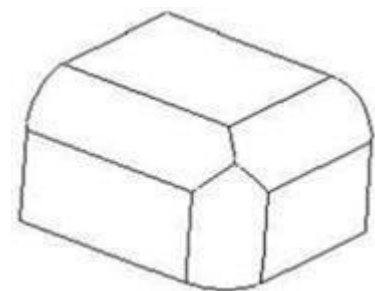
4. The edges are rounded, and meet at a sharp corner.



To round another edge, select the object again and select the next edge. Make sure **Pick smooth sequence** is not selected.



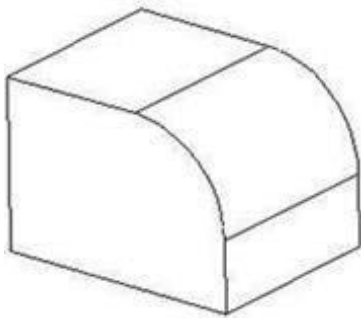
This is the result - the intersection of all three corners has sharp edges.



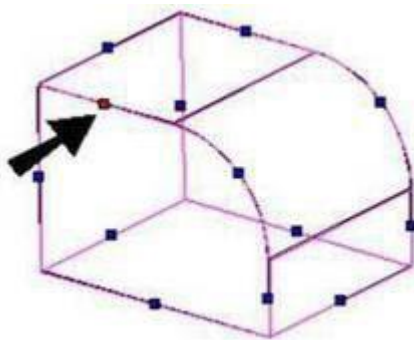
Pick Smooth Sequence

Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex. When working with **Round vertex**, smoothly connected edges are automatically selected. But this is not the case when **Round vertex** is turned off.

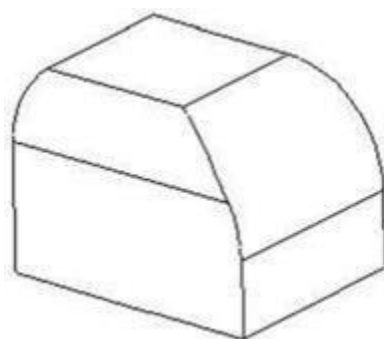
1. Start with a box with one filleted edge, using a large radius.



2. Activate **Fillet** without **Round vertex**, and fillet one edge, using a smaller radius.



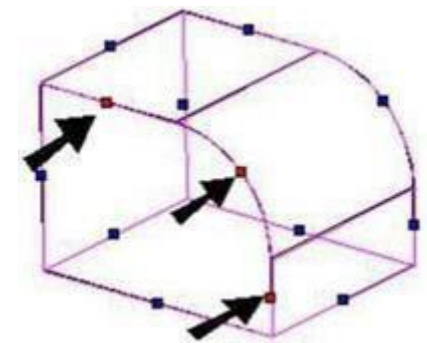
This is the result - only the selected edge was rounded.



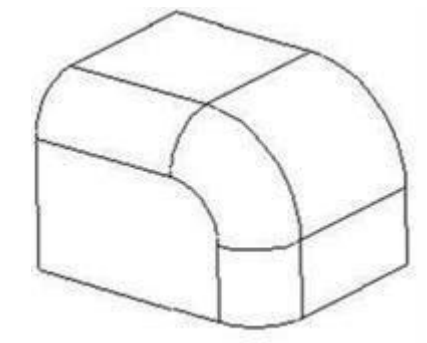
3. Undo, and this time select **Pick smooth sequence**.



This time when you select one of these edges for filleting, all edges in the smooth chain are selected.

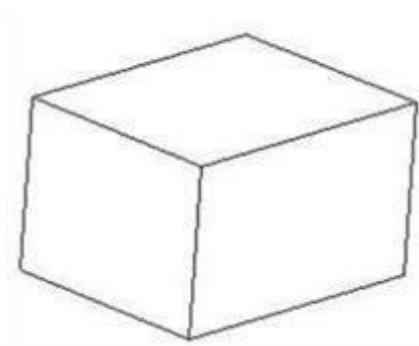


This is the result - all edges in the chain are rounded.

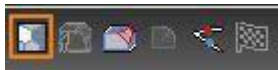


Fillet with Round Vertices

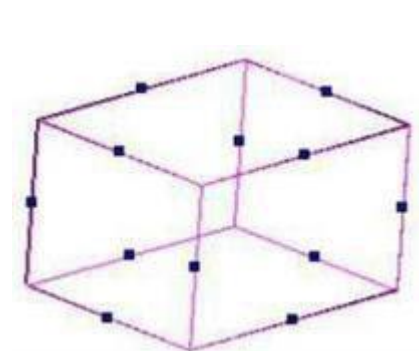
Rounding the vertices produces a smooth rounding along all edges and corners. This example will start with a box.



Set the **Start Radius** in the Inspector Bar. Select **RoundVertex** in the Inspector Bar or on the local menu.



Select the object whose edges are to be rounded. Each edge is marked with a blue square.

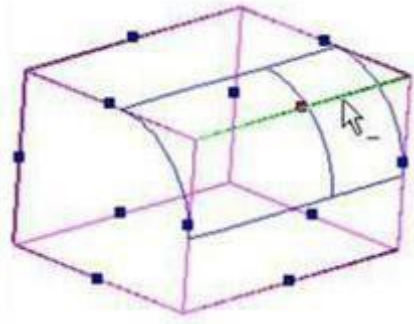


If you select **Hide Marks**, these edge markers will be hidden.



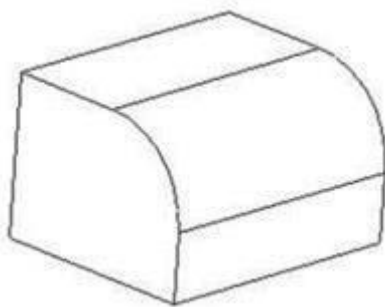
3. Select the first edge to round. A preview appears showing the fillet that will result.

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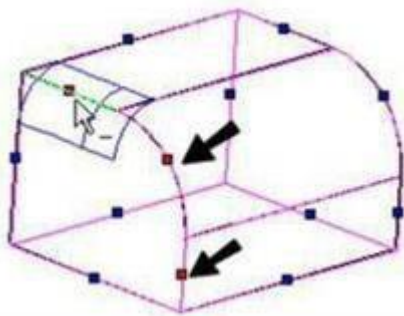


Continue selecting edges, if needed. (You can also drag a window to select multiple edges). Selected edges are indicated by a red square.

To complete the fillet, select **Finish** from the Inspector Bar or the local menu.

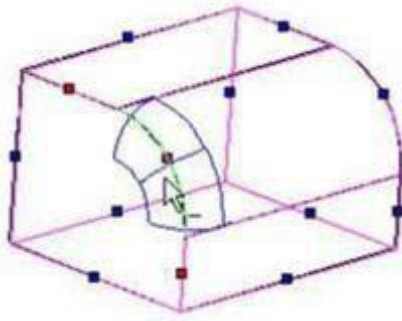


In this example, if you select an edge that is smoothly connected to other edges, the connected edges will be selected automatically.

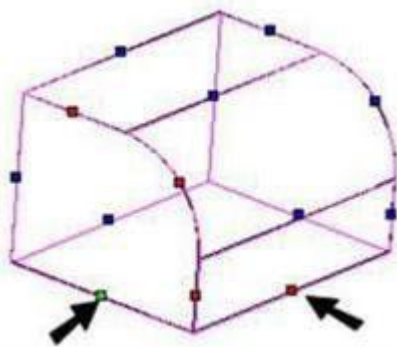


Note: Adjacent edges are smoothly connected if they are connected by an arc or rounded vertex.

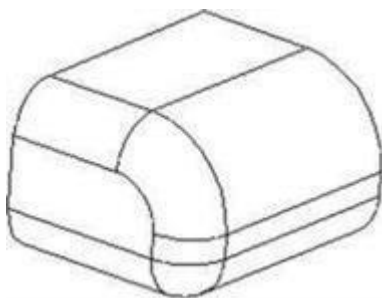
7. Clicking other edges will show the fillet preview along that edge.



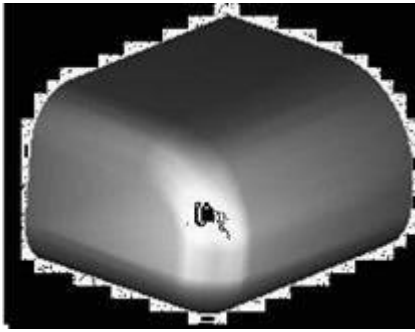
In some cases, selecting an edge (such as the bottom front edge) will cause another edge to be rounded as well (the bottom side edge), because they meet at a common vertex. These edges are indicated by a green square.



When all edges are selected, set the rounding radius in the Inspector Bar, and select **Finish** from the Inspector Bar or the local menu.



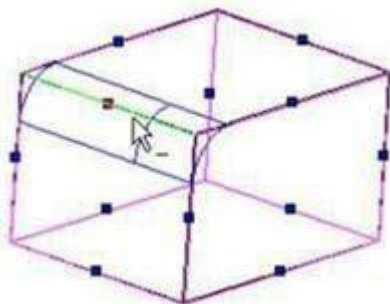
Viewing the results in render mode shows the effect of rounding. The common vertex is smoothly rounded.



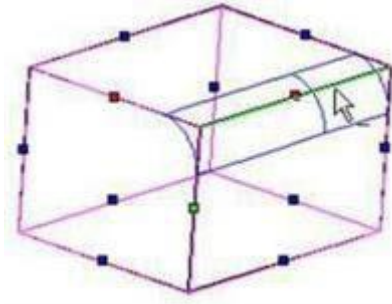
Fillet with Setbacks

When rounding edges with the **Round Vertex** option, you can enter setbacks in the Inspector Bar. The setback is the distance from the vertex to the point on an edge where the rounding begins. Setbacks are visible at vertices where all edges are rounded. The setback value must be greater than the edge rounding radius.

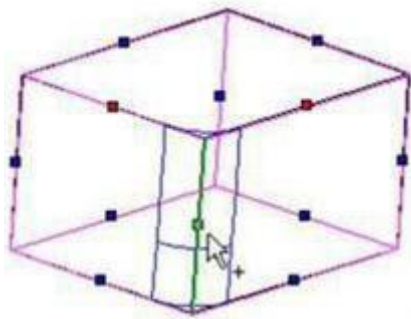
Select the object, set the radius, and select the first edge to round. When an edge is selected, the blue square turns red. If you select an edge that is smoothly connected to other edges, the connected edges will be selected automatically.



2. Continue selecting edges.

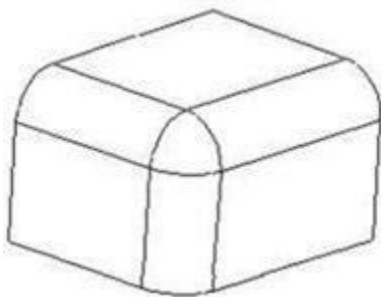


When you select two adjacent edges, any other edge that meets that vertex will be automatically selected as well, indicated by the green square.

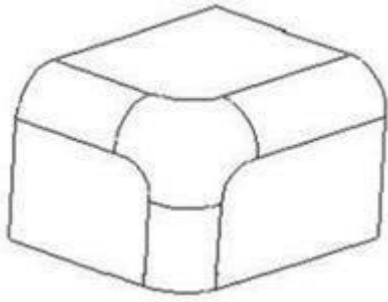


When all edges are selected, set the rounding radius in the Inspector Bar, and select **Finish** from the Inspector Bar or the local menu.

If no setbacks are used, the rounding extends along the entire edge.



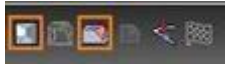
If setbacks are used (entered in the Inspector Bar), the rounding starts at the specified distance from the vertex.



Note: If the setback values are different, where they are applied depends on the structure of the object itself. You may need to experiment to get the desired result.

Fillet with Variable Radius

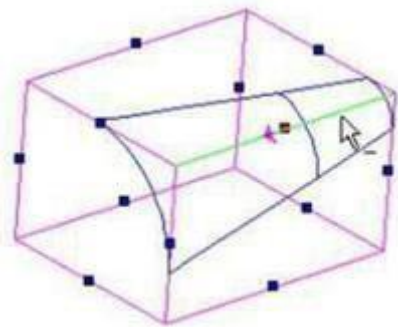
For all rounding types, you can use the **Unequal Radius** option to specify the radius at the start and end of the edge.



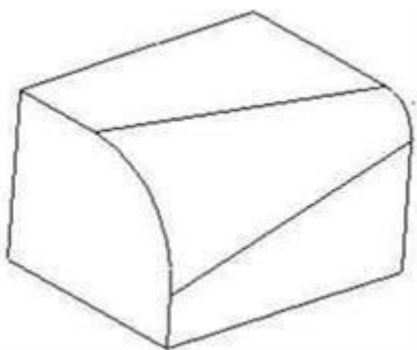
Specify the **Start Radius** and **End Radius** in the Inspector Bar.

Start Radius	End Radius	Setback 1	Setback 2
1.2 in	0.4 in	0 in	0 in

Select the object, and select the edge or edges to round. For each edge, a small magenta arrow indicates the direction of the edge, so that you will know where the **Start Radius** and **End Radius** will be applied. You can reverse the direction by pressing Ctrl and selecting again.



The edge is rounded with a varying radius.



Holdline Blend

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

This option allows you to impose limits and shape to the fillet you are creating. This is done by selecting line, circles, arcs, and curves to define the boundary of a fillet.

Select the Fillet tool.

Select the object to be filleted.

3. Select the Holdline Blend option.



4. Select the edge/s to be filleted.

Select **Finish** from the Local menu or Inspector bar.

Select Holdline.

Editing the Hold Line

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You can change the holdline by moving or node editing to change the variable blend.

By Moving:

By Node Editing:

Below are three new Blend types that were added in TurboCAD 2019.

Curvature Continuous Blend

This option allows you to create explicit blends.

The blend surface will be curvature-continuous with both faces.

To make it you should:

- Define Radius, using inspector bar

- Select Edge by click

It is a single click blend. You cannot select more than one edge.

Example 2:

Fixed Width Blend

Creates a blend with a fixed width between the two spring curves of the blend. The actual blend radius is calculated implicitly along the blend to produce a blend with a constant cross-sectional width.

To make it you should:

- Define width,

- Select edges,

- Finish

Example 2:

Three Face Blend

It is for a variable radius of three face blend. The blend surface will be tangent to all three support faces. The cross-section of the blend surface is circular and the variable radius of the blend is driven by the requirements of tangency to the surfaces.

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To make Three Face Blend you should:

- Select Left face

- Select Right Face

- Select Center Face

Example 2:

Demo Video:

<https://youtu.be/M4-SOMnfgH4>

Variable Blend

This option allows you to create a complex blend by using a series of point to specify radii. The Part Tree must be active for all of the features of Variable blending to work. But the points must be EXACTLY on the original edge (to 8 significant digits). Use the N SEKE (near) or Near snap to ensure correct placement.

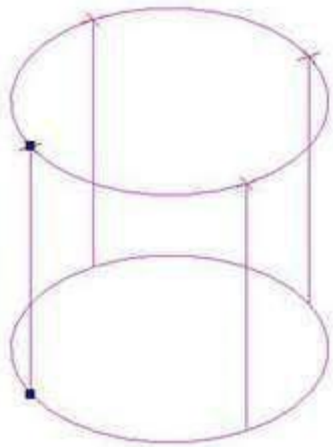
Do not use the Dot or AutoCAD point options as they are infinitely small and the Variable Blend option uses the size of the point to control the size of the fillet.

1. Use the point tool to set points along the edge that is to be filleted.

Select the 3D Fillet tool.

Select the Variable Blend option.

Select the object to be filleted.

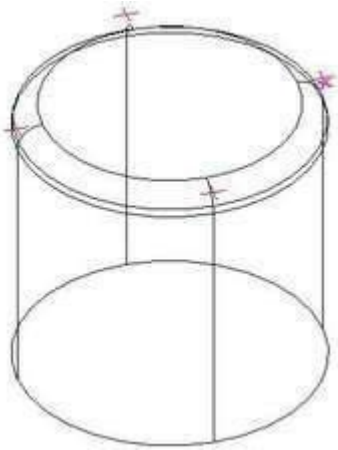


5. Select the Edge to be filleted.

Select Finish

Select the points that will be used to control the variable blend.

Select Finish.



Changing the Fillet Radii

You can adjust the radii of the various fillet points by using the Blend Radius Positions in the Part Tree.

Select the Variable Radius Blend in the Part Tree

Expand each position field to be edited.

Type in a new value for the Radius.

You can also adjust the value of each radius by changing the size of the controlling points.

Select the point

Go to the Selection info palette.

Expand the Point section.

In the Size field, enter a new value for the size of the point/radius.

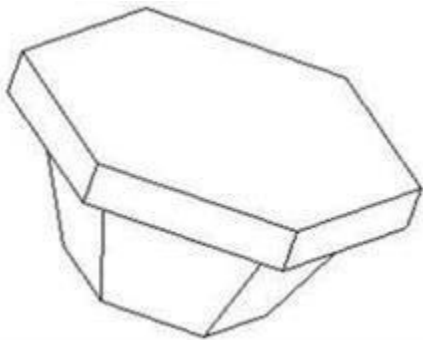
10.10. Shelling Solids

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Facet Editing/Shell Solid

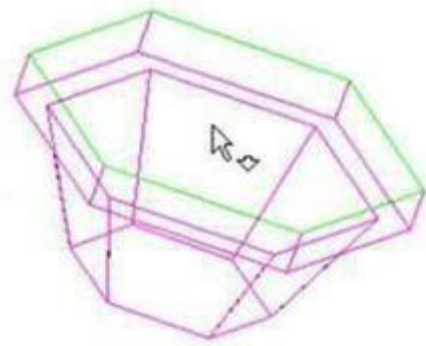
Ribbon UI Menu:

Creates a shell of a specified thickness from a single solid (not surface) object. The new faces are creating by offsetting existing faces inside or outside. This example will use the solid shown below. (It is actually two solids, one **Lofting** and one **Simple Extrude**, combined into one solid using **3D Add**.)

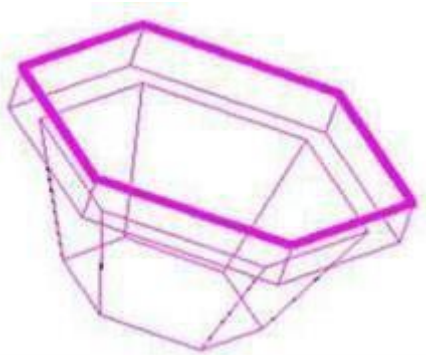


Activate **Shell Solid**. Select the object, which turns magenta.

Select any faces you want to remain open. Faces are highlighted in green when the cursor passes over them.

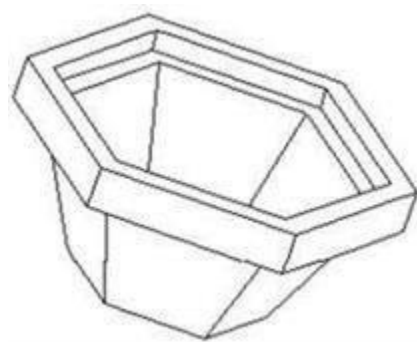


Open faces, when selected, are indicated by a thick magenta line. You can use the Page Down or Page Up key to select the next face.

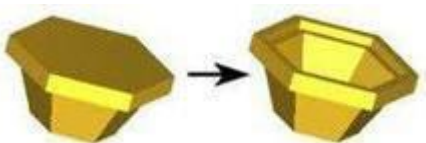


Enter the shell thickness in the Inspector Bar. A positive value created an outward shell, and a negative value creates an inward shell.

Select **Finish** from the local menu or Inspector Bar.



Viewing the results in render mode shows the shelling results.



After the shell operation is complete, you can edit it in the Selection Info palette.

See [Editing 3D Objects using Selection Info](#)

Sheet Thicken:

Apply the Sheet Thicken to Sheet ACIS® entities to obtain Sheet Thicken entities. Use the option 'Perform in both directions' to apply by Shell Solid. **Part t ree** option must be enabled in ACIS in drawing setup otherwise sheet thicken will not work.

Apply thickness to an unclosed line or a 2D polyline/ spline from selection info palette or properties window in 3D option.

- ii) Enable Shell Solid and select 'Perform in both directions' from the inspector bar.

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Note: The selected object's Solid mode should be solid. You can set Solid mode in the selection info palette or Properties box in the 3D tab.

Local Menu Option:

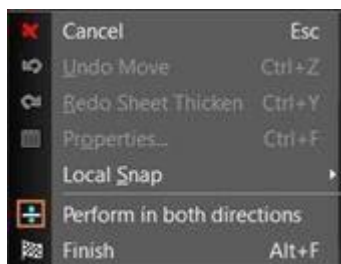
Perform in both directions: Enables Sheet thickens on both sides of the solid.

Sheet Thicken can also be performed on ACIS solid object.

Draw any 2D object.

Apply surface from profile from solid/ surface toolbar.

Enable Shell Solid and select 'Perform in both directions' from the inspector bar or local menu.



Sheet thicken operation is performed on both sides.

Demo Video:

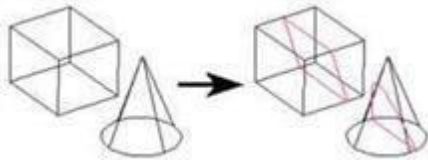
<https://youtu.be/-1DFP9HzPY8>

10.11. Section

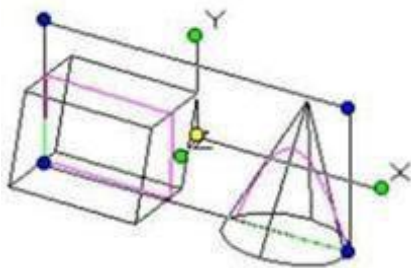
Default UI Menu: Modify/Section

Ribbon UI Menu:

Creates a 2D or 3D section of one or more 3D solid or surface objects.



The resulting sections are grouped into one object.



The sections are considered a "Group of graphics," which can be viewed and manipulated in the Selection Info Palette (see [Selection Info Palette](#)).

For a 2D section, the "Group of graphics" is composed of lines, circles, ellipses, circular arcs, elliptical arcs and/or splines as needed to create the section. For a 3D section

(**Section by Closed Polyline**), the "Group of graphics" is composed of solids and/or surfaces generated by the intersection of the selected objects and the normal extrusion of the closed polyline.

Note: The **3D Slice** tool is similar, but uses the specified plane to divide the object into new objects. See [3D Slice](#)

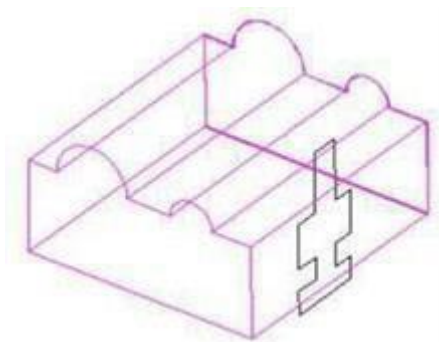
Section by Closed Polyline

Creates a 3D section by using a closed polyline, rectangle, or polygon to cut through the solid. The section is normal to the plane of the polyline.

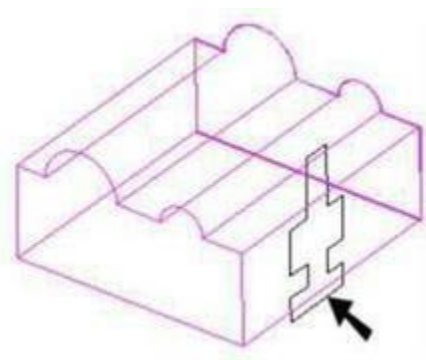
Select **Section by Closed Polyline** from the local menu or Inspector Bar.



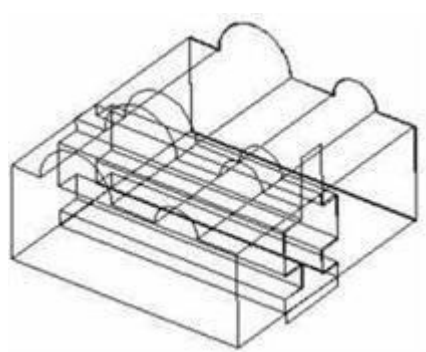
Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).



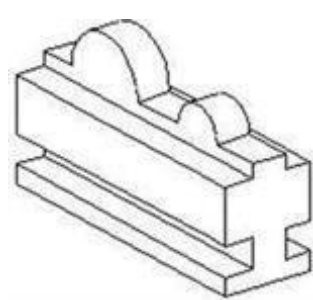
3. Select the closed polyline.



4. The 3D section is created.



You can view the 3D section more clearly by removing the original solid. (The render mode here is **Hidden Line**.)



Section by Line

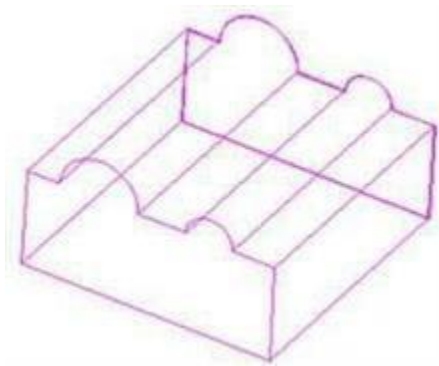
Creates a section perpendicular to the current view, by defining two points. This is the default option.

1. Select **Section by Line (2 Points)** from the local menu or Inspector Bar.

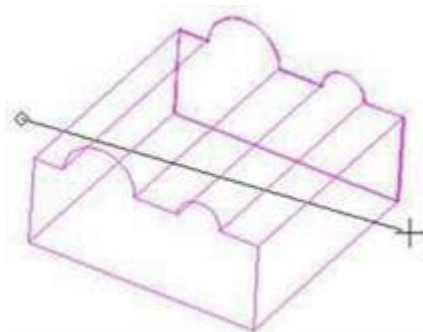
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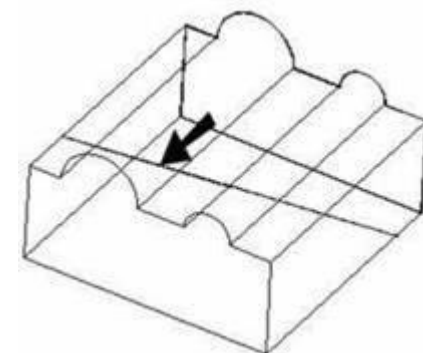
Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).



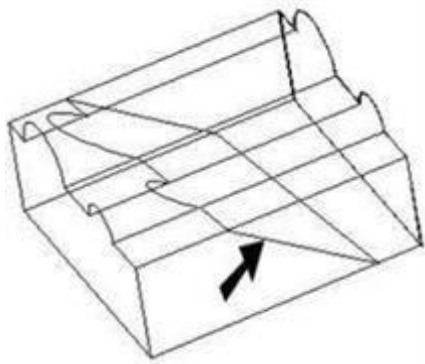
Select two points. The section plane will pass through the line defined by these points, in the direction normal to the current view (into the screen).



4. The section is created, but in the current view it appears as a line.



You can rotate the view to see the section.



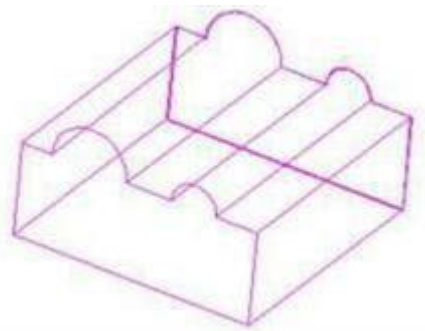
Section by Plane

Creates a section by defining the sectioning plane. The plane is defined by three points.

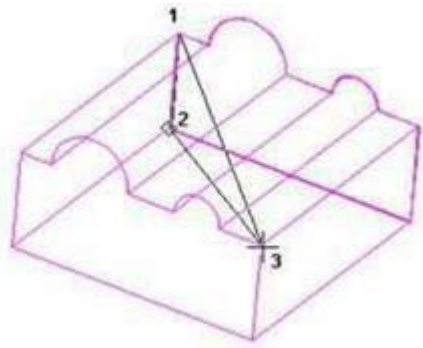
Select **Section by Plane (3 Points)** from the local menu or Inspector Bar.



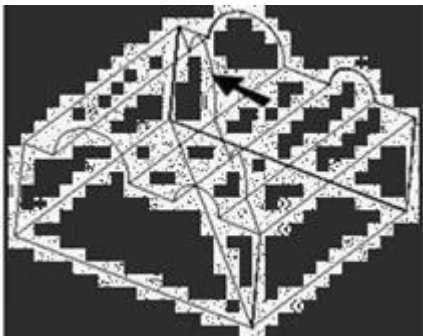
Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).



3. Select three points to define the sectioning plane.



4. The section is created.



Section by Workplane

Creates a section by using the current workplane as the sectioning plane. For details on workplanes, see [Workplanes](#)

Select **Section by Workplane** from the local menu or Inspector Bar.



Select the object to section. You can use Shift to select more than one object, or use a selection window (first click outside the objects, then drag the window).

Note: To section multiple objects, start out using a different option - **Section by Line** or **Section by Plane**. Use Shift to select the objects to slice, then click the **Section by Workplane** icon.

3. The section is created.

You can rotate the view to see that the section lies on the workplane.

10.12. Hole

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Hole

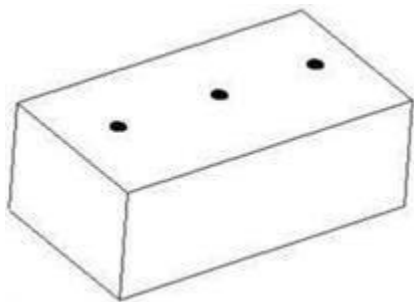
Ribbon UI Menu:

Note: Hole tool is only available in Platinum.

Works by using a Boolean subtract operation to remove hole volumes from a solid object (or add material).

Start with a solid object, like a box.

Create points for each hole you want to create (see [Point](#)). The points must lie on the face from which the hole will be removed.



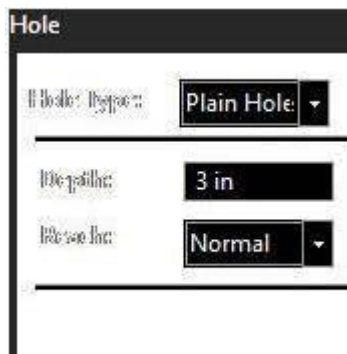
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Note: As long as **Workplane by Face mode** is checked in **Drawing Aids**, the points will be placed on the facet where your cursor is. (See [Snap Settings](#)) Otherwise, you can use **Workplane by Facet** to place points on the correct facet.

Activate **Hole**, and open the **Properties**.



On the **Hole** page, select the type of hole. The types of holes are Plain hole, Boss.

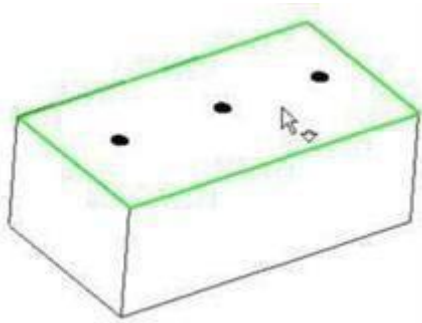


5. The other parameters on this page depend on the hole type.

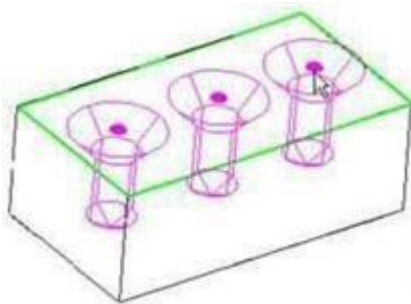


6. Close the **Properties**, and select the face on which the holes will be created.

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Select the points. Use the Shift key to select multiple points. The preview shows what the final result will look like.

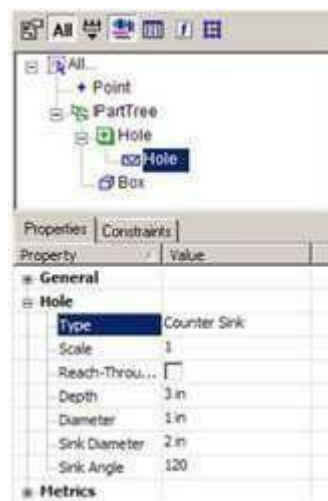


8. Select **Finish** from the local menu or Inspector Bar, and the holes are created.



You can edit hole parameters after they are created. See [Editing 3D Objects using Selection Info](#)

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10.13. Imprint

Imprint

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Imprint

Ribbon UI Menu:

Modifies a facet of a solid (not surface) object by imprinting a closed 2D profile object (polyline, polygon, circle, arc, or curve). Multiple profiles can be imprinted at once. You can also add a fillet to the bottom and/or chamfer to the top. **Imprint** uses Boolean operations of **Add** and **Subtract** on the original solid and the extrusion of the 2D closed profile. See [3D Boolean Operations](#). Imprint is available on the fly-out toolbar from the **Drawing Tools** toolbar or the **3D Object** toolbar.

NOTE: The closed 2D profile object must be located on a facet of the 3D object. You can use text as part of the compound profile for imprinting.

Imprint Auto Detect

Adds or subtracts the extrusion of the closed profile to/from the solid, depending on which direction the profile is extruded.

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Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.

Make sure **Auto Detect** is selected. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



Select the closed profile. If you want to select more than one profile, press the **Shift** key.

Pull the profiles away from the solid, and click to create the imprint, or enter a **Height** in the Inspector Bar. Because the extrusion direction was outward, the extrusions were added to the solid.

If you **Undo** and recreate the imprint, pushing into the solid this time, the extrusions will be subtracted from the solid.

Imprint Add

Adds the extrusion of the closed profile to the solid, regardless of the extrusion direction.

Select the desired face.

Now select **Add**.

Select and extrude the profile, and now the extrusion is added to the solid.

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Imprint Subtract

Adds the extrusion of the closed profile to the solid, regardless of the extrusion direction.

Select the desired face.

Now select **Add**.

Select and extrude the profile, and now the extrusion is added to the solid.

Imprint with Chamfer or Fillet

You can add a chamfer or fillet to the top or bottom of the imprint.

Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.

In this example, **Auto Detect** is selected. When the last two options are set to **Normal Top** and **Normal bottom**, the imprint edges will be sharp.

To fillet or chamfer, click the option icons to set them to **Fillet Top / Bottom** or **Chamfer Top / Bottom**. Clicking repeatedly will scroll through the options. In this example, the bottom has a chamfer, and the top has a fillet.

Set the **Top radius** and **Bottom radius**.



Select and extrude the profile. The top is filleted and the bottom is chamfered.

You can edit all parameters of the imprint (including changing or removing fillets and chamfers) in the **Selection Info** palette.

Imprint with Dimple Sheet Metal

You can use the imprint tool to create a dimple effect as in sheet metal.

Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.

Select the **Dimple sheet metal** option.



Select a closed profile.

The height must be a negative value greater than the sheet thickness. The top radius must be equal to or greater than the thickness of the sheet.

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Imprint with Dimple Sheet Metal

You can use the imprint tool with slots use as the profile/s.

1. Select the desired face.

Now select **Slot operation**.



Select and extrude the slot profile/s, and now the extrusion is combined to the solid.

The Selection Info palette can be used to edit parameters of the imprint itself, as well as the fillets and/or chamfers. Select the imprint, and its **Part Tree** appears at the top of the palette. Under **Imprint**, highlight **Simple Extrude**.

At the bottom of the palette, the Simple Extrude category contains parameters for the imprint body:

Draft Angle: Creates an extrusion of increasing or decreasing cross-section.

Enter the angle of deviation from the extrusion path.

Draft Start / End Distance: If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.

Height: The distance of the extrusion.

Direction: Switch between one-sided and two-sided.

The **Imprint** category is where you can change the imprint type (**Auto Detect**, **Add**, or **Subtract**) and change or remove fillets and chamfers and change their radii.

Editing Imprint Parameters

The Selection Info palette can be used to edit parameters of the imprint itself, as well as the fillets and/or chamfers.

Note: For general information on this palette, see [Selection Info Palette](#)

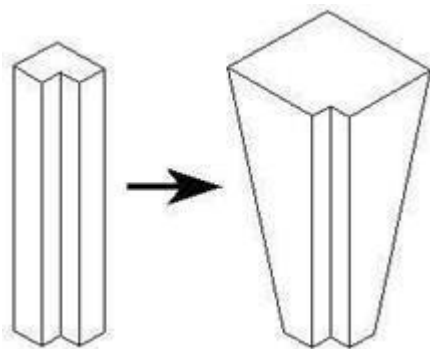
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Select the imprint, and its **Part Tree** appears at the top of the palette. Under **Imprint**, highlight **Simple Extrude**.



At the bottom of the palette, the **Simple Extrude** category contains parameters for the imprint body:

Draft Angle: Creates an extrusion of increasing or decreasing cross-section. Enter the angle of deviation from the extrusion path.



Draft Start / End Distance: If **Draft Angle** = 0, you can specify a draft angle by entering the offset distances.

Height: The distance of the extrusion.

Direction: Switch between one-sided and two-sided.

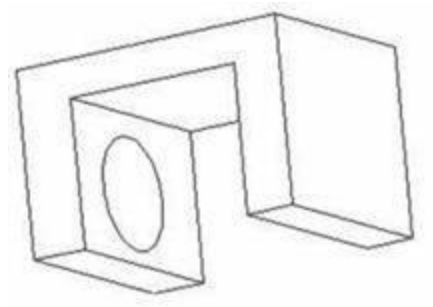
The **Imprint** category is where you can change the imprint type (**Auto Detect**, **Add**, or **Subtract**) and change or remove fillets and chamfers and change their radii.

Imprint Add

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Adds the extrusion of the closed profile to the solid, regardless of the extrusion direction.

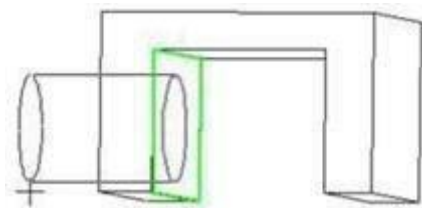
1. Start with a solid like this, with a closed profile within one of the inside facets.



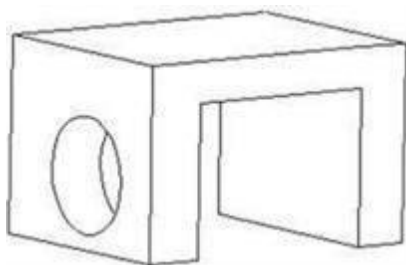
Activate **Imprint**. For the sake of comparison, first select **Auto Detect**. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



Select the inner face that contains the profile, then select the profile itself. Extrude the profile this way:



This is the result: because the profile was extruded inward with respect to its face, its extrusion was subtracted from the solid.

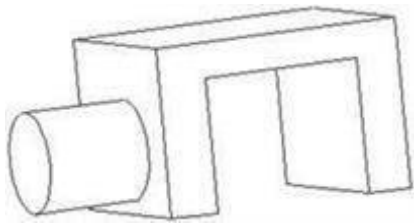


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Undo, and now select **Add**.



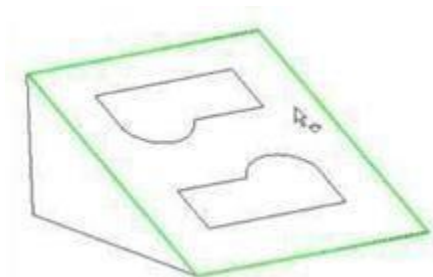
Select and extrude the profile the same way, and now its extrusion is added to the solid.



Imprint Auto Detect

Adds or subtracts the extrusion of the closed profile to/from the solid, depending on which direction the profile is extruded.

Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.

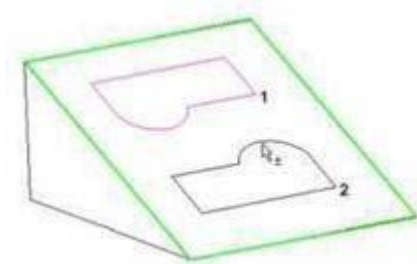


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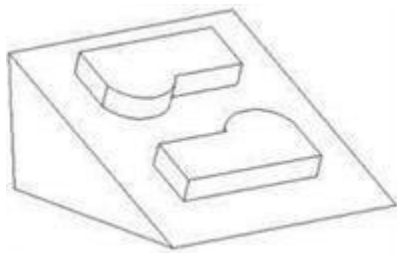
Make sure **Auto Detect** is selected. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



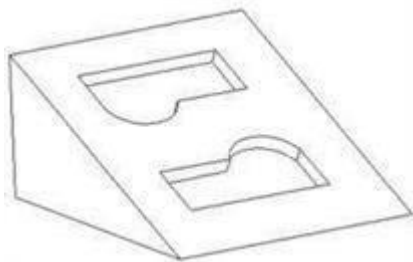
Select the closed profile. If you want to select more than one profile, press the Shift key.



Pull the profiles away from the solid, and click to create the imprint, or enter a **Height** in the Inspector Bar. Because the extrusion direction was outward, the extrusions were added to the solid.



If you **Undo** and recreate the imprint, pushing into the solid this time, the extrusions will be subtracted from the solid.

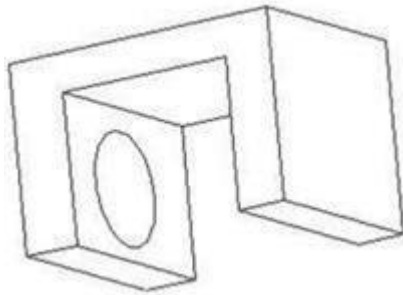


Imprint Subtract

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Subtracts the extrusion of the closed profile from the solid, regardless of the extrusion direction.

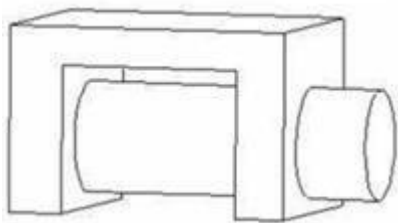
1. Start with a solid like this, with a closed profile along one of the inside facets.



Activate **Imprint**. For the sake of comparison, first select **Auto Detect**. In this example, the last two options in the Inspector Bar have no blend (fillet or chamfer).



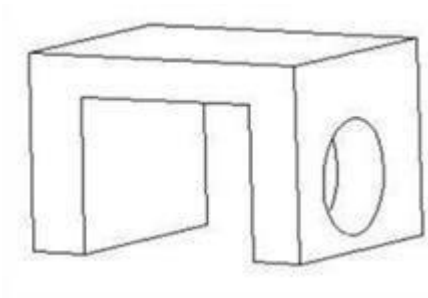
Select the inner face that contains the profile, then select the profile itself. Extrude the profile this way. Because the profile was extruded outward with respect to its face, its extrusion was added to the solid.



Undo, and now select **Subtract**.



Select and extrude the profile the same way, and now its extrusion is subtracted from the solid.

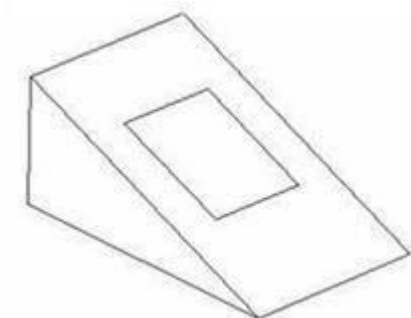


Imprint with Chamfer or Fillet

You can add a chamfer or fillet to the top or bottom of the imprint.

Note: For details on these edge tools, see [Fillet Edges](#) and [Chamfer Edges](#)

Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.



In this example, **Auto Detect** is selected. When the last two options are set to **Normal Top** and **Normal bottom**, the imprint edges will be sharp.

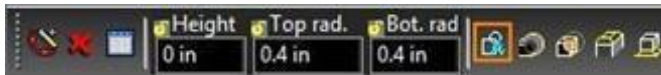


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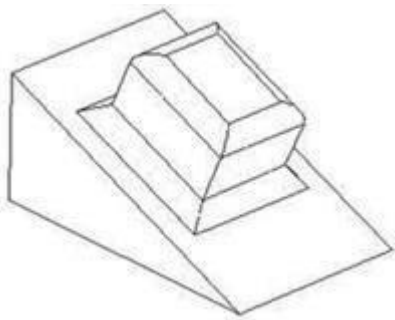
To fillet or chamfer, click the option icons to set them to **Fillet Top / Bottom** or **Chamfer Top / Bottom**. Clicking repeatedly will scroll through the options. In this example, the bottom has a chamfer, and the top has a fillet.



Set the Top radius and Bottom radius.



Select and extrude the profile. The top is filleted and the bottom is chamfered.



You can edit all parameters of the imprint, including changing or removing fillets and chamfers, in Selection Info palette.

Imprint with Dimple Sheet Metal

You can use the imprint tool to create a dimple effect as in sheet metal.

Start with a solid object with one or more profiles on one facet. Activate **Imprint** and select the facet that will be imprinted.

Select the **Dimple sheet metal** option.



Select a closed profile.

The height must be a negative value greater than the sheet thickness. The top radius must be equal to or greater than the thickness of the sheet.

10.14. Modifying Object Geometry

As with 2D geometry, there are several ways to modify the geometry of an object. Node editing (**Edit Tool**) can be used on an object itself, or on the profile used to create an object. (A profile can also be moved, rotated, or scaled in **Select Edit** mode.)

Objects in 3D can be moved, scaled, copied, and rotated in **Select Edit** mode (see [Select Edit](#)), in the same way as for 2D objects. The 3D Selector must be active for work in 3D (see [2D-3D Selector](#)).

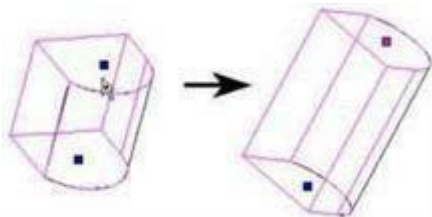
In 3D **Select Edit**, you can move in 3 dimensions and rotate about 3 axes.

Edit Tool in 3D

Most 3D objects do not have nodes to edit. The exceptions are **Box**, **Cone**, **Sphere**, **Hemisphere**, **Revolve**, and **Extrude**. These objects have limited nodes that can be used only for scaling and orientation

These nodes are lost once Boolean operations are performed.

Editing nodes of 3D objects works the same way as for 2D objects (See [Edit Tool](#)). The main difference is that nodes can be moved anywhere in 3D space. Also, 3D nodes cannot be added or deleted from objects themselves (though they can be added or deleted while node editing profiles).



While node editing is limited, if possible at all, for most 3D solids, you can node edit an exploded surface object.

You can change a solid object to a TurboCAD (non-ACIS) surface object via the **3D** page of its **Properties** window. Explode the surface so that it can be node edited.

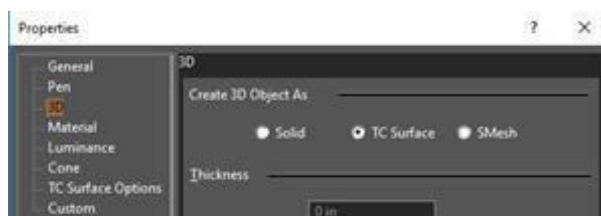
Tip: You can explode a solid object twice to turn it into an exploded (node editable) surface object.

Editing Nodes of Exploded Surfaces

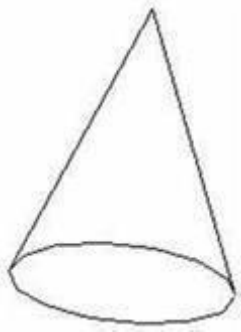
This example uses a basic cone. If it was created as a solid, and you try to node edit it, you will see that it cannot be selected.



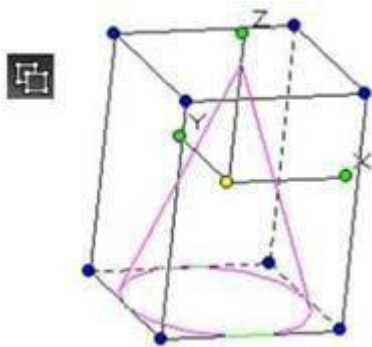
Select the cone and open its **Properties** (see [Object Properties](#)). On the **3D** page, select **TC Surface**.



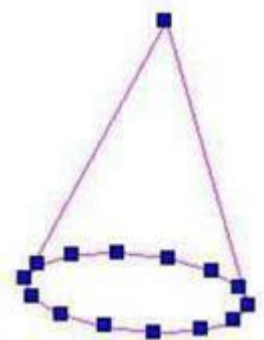
The cone is now comprised of surfaces, but it still cannot be node edited.



2. Select the cone again, and click **Explode** (or select **Modify / Explode**).

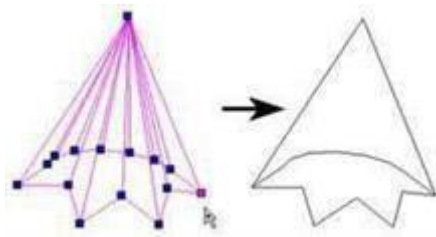


3. You can now node edit the cone nodes.

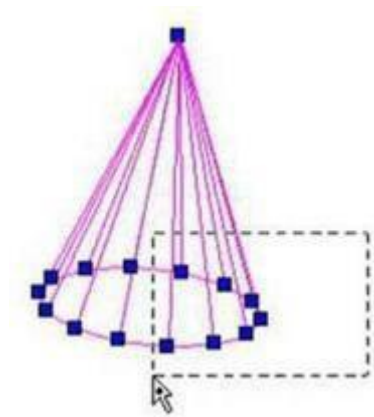


Note: You also could have exploded the original solid object twice to obtain an exploded (node editable) surface.

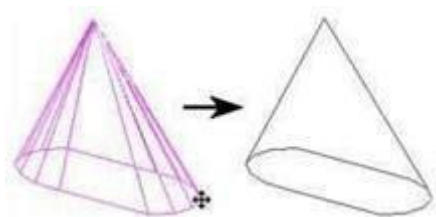
4. Drag individual nodes to change the shape of the cone.



You can also drag a group of nodes simultaneously. Use a selection window to select the desired group of nodes.



After they are selected, the nodes appear in magenta. Now when you drag any one of the selected nodes, the entire group moves.

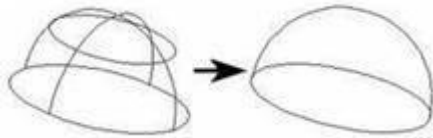


Editing Nodes of Edges of Exploded Surfaces

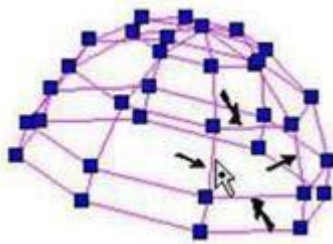
For exploded 3D surface objects, you can use the **Edit Tool** to control the visibility and usage of edges.

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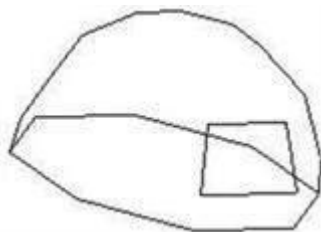
Create a solid object and transform it into an exploded surface object, using the methods described in [Edit Tool in 3D](#)



Node edit the object. To make an edge visible, right-click between its nodes and select **Make edge visible**.



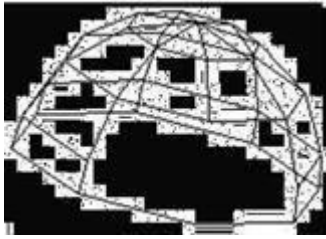
Do the same for all edges you want to show. You can hide visible edges by selecting **Make edge invisible** from the local menu. Press Esc twice to leave the **Edit Tool**, and the selected edges are shown.



To modify form-building edges, turn them on by checking **Draw form-building edges** on the **Display** page of the **Drawing Setup (Options / Display)**.



5. Right-click on nodes you want to hide and select **Make edge non form-building**.



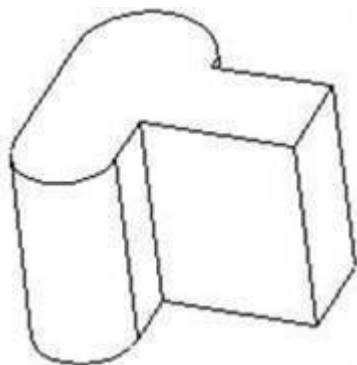
You can display edges by selecting **Make edge form-building**.

Profile Editing

Some types of 3D objects are created by performing a function on a profile (see [Profile Objects and Face to Face Modifications](#)). Such objects include **Simple Extrude**, **Revolve**, and **Lofting**. Some standard objects are also based on profiles, such as **Cone**, **Wedge**, **Cylinder**, **Polygonal Prism**, and **Torus**. For standard objects, the profiles are created within the 3D tool; profile objects reference a profile that has already been created.

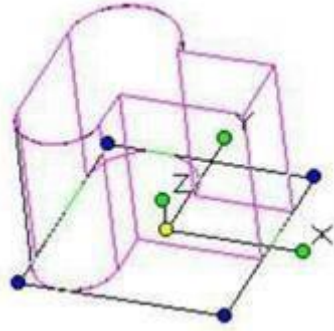
For objects that are based on a 2D profile, you can edit the profile in **Select Edit** mode (see [Select Edit](#)).

This example will use a **Simple Extrude** object, created from a closed polyline.



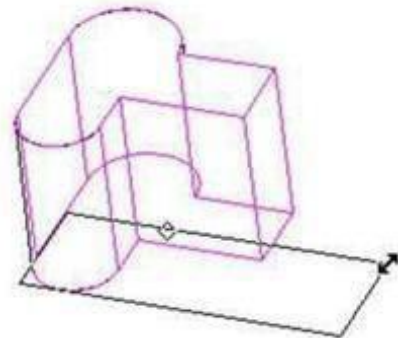
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In **Select** mode, select the object. Right-click and choose **Select Profile** from the local menu.

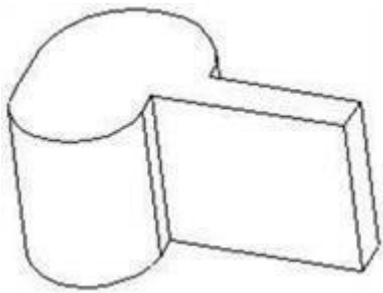


Warning: The workplane moves to coincide with the selected profile. This may change the workplane you want to be active. If needed, you can use **Previous** to return to the desired workplane.

You can move, rotate, or scale the profile as you would any 2D object in **Select Edit**. See [Select Edit](#)



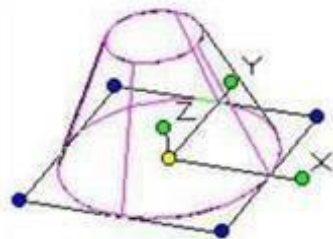
Once the profile is edited, click outside to return to the 3D object, and click outside again to clear the selection. (You can also press Esc twice to exit **Select** mode.) The object is now based on the edited profile.



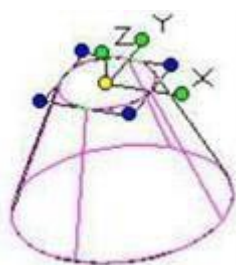
Editing Multiple Profiles

Some objects like truncated cones and lofts use multiple profiles. You can access and edit each profile separately. This example will use a truncated cone.

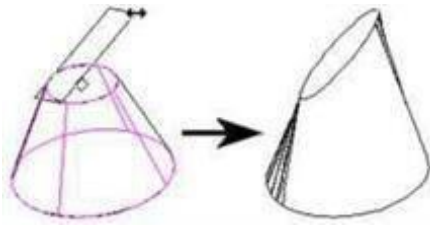
In **Select** mode, select the object. Right-click and choose **Select Profile** from the local menu.



If the selected profile is not the profile you want to edit, right-click again and choose **Select Next Profile**. You can use this method to scroll forward and backward (**Select Previous Profile**) until you reach the desired profile.



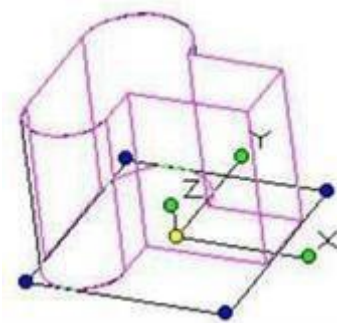
3. Edit the desired profile as needed to change the object.



Editing Nodes of a Profile

If you want to edit profiles beyond the moving, rotating and scaling possible in **Select Edit**, you can also edit profile nodes. For details, see [Edit Tool](#)

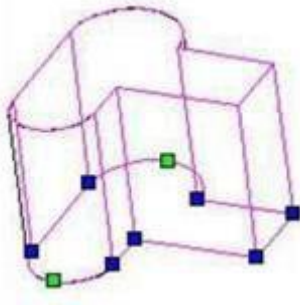
In Select mode, select the object. Right-click and choose **Select Profile** from the local menu.



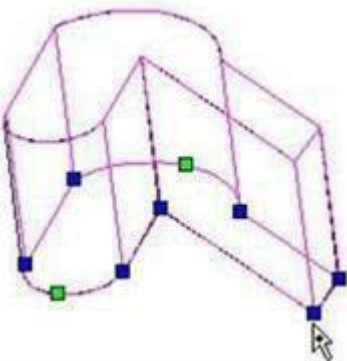
Right-click again and select **Edit Node**, or click **Edit Node** in the Inspector Bar.



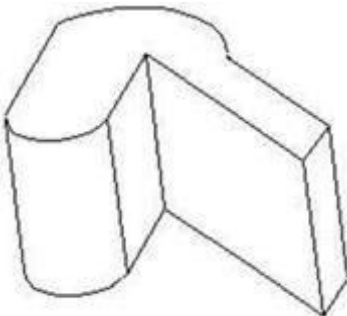
The profile nodes appear.



3. Drag the nodes as needed to change the profile shape.



4. Return to **Select** mode, and then clear the selection, by pressing Esc twice.



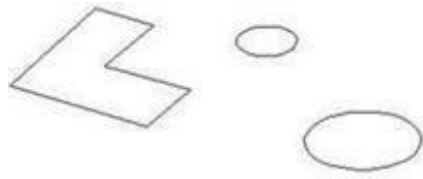
Updating Compound Profiles

(Available only in Platinum)

Profile objects can be based on simple (single-curve) profiles or compound profiles. Once the 3D object is created, the curves that comprise the compound profile can be changed, and the resulting solid will update.

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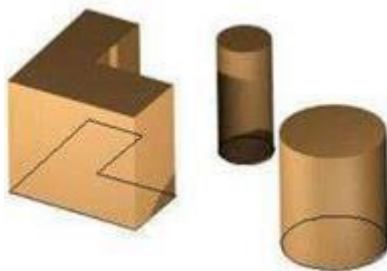
This example uses these three closed 2D compound profiles, but simple profiles will work as well.



In this example, a **Simple Extrude** will be created (see [Profile Objects and Face to Face Modifications](#)). Make sure **Use Compound Profile** is selected.



Select one profile, then press Shift and select the other two.

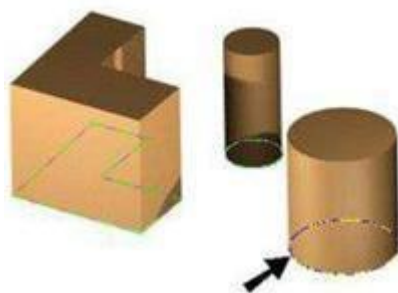


3. There are two ways to update the profile:

Select **Update Compound Profile** and select the profile used to create the extrusion. Or **Select** the 3D object. With it selected, right-click and select **Select Profile**. With the profile then selected, right-click again and select **Update Compound Profile**.

1. Click one profile to deselect it.

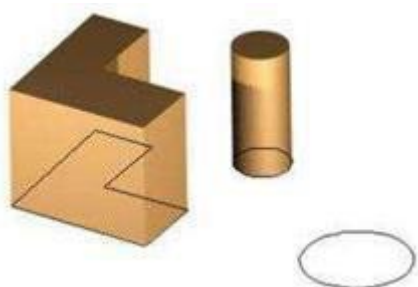
TurboCAD 2023 User Guide Publication



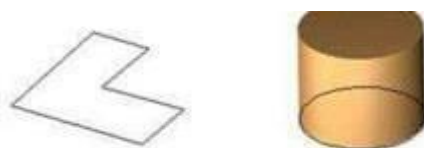
When the profile is modified, select **Finish Selection of Profile**.



The extrusion updates to reflect the selected profile curves.

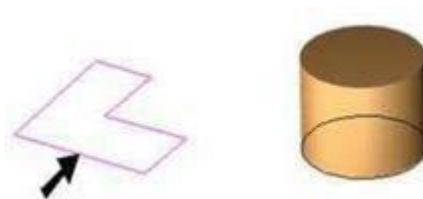


You can also use this tool to add or delete regions from multiple-compound region extrusions. You can also select profiles that were previously unused. Finally, you can update a 3D object that is based on a simple profile to be based on a compound profile (but not vice-versa). In this example, the extrusion is based on a circle, and was not



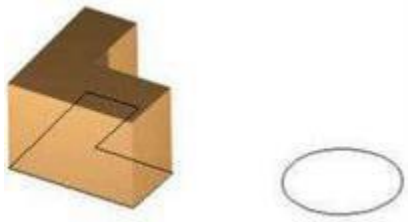
created using **Use Compound Profile**.

Select the extrusion, and select **Select Profile**. Right-click again and select **Update Compound Profile**. Select the compound profile.



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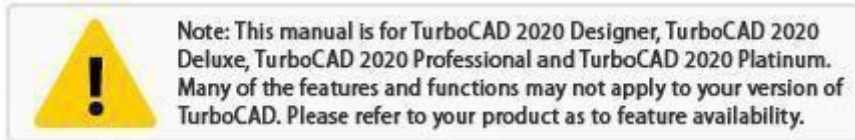
Click **Finish Selection of Profile**, and the extrusion is now based on the compound profile.



While creating solids based on compound profiles, it is recommended to work with Editing History active (see [Editing 3D Objects using Selection Info](#)). This ensures that you will be able to modify the profiles in **Select Edit** mode.

10.15. Flat Shot

Flat Shot



(Available in Platinum only)

Default UI Menu: Modify/Modify 3D Objects/Flat Shot

Ribbon UI Menu:

The FlatShot tool allows users to create 2D views of a 3D object, including Visible and Invisible Lines. These views are saved as Blocks in the Vlocks Palette,

To create a Flat Shot:

Select a 3D entity from the drawing

Enable Flat Shot tool from the Tools menu

The Flat Shot Options dialog opens:

From here, users can create Custom line types for Visible and Invisible lines, as well as choosing to display Hidden lines or not. Clicking the 'Edit Style' box, takes the use to the Pen Properties menu, where custom Pen (Line) Pattern, Color and Width can be defined, as well as Dash Scale in the Pattern.

5, After making any edits to the Pen properties, press OK to exit from this menu, and then press OK again from the Flat Shot Options menu. The view will be written to the Blocks palette.

10.16. Surface and Solid Conversion Operations

Surface and Solid Conversion Operations

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Create Surface, Modify/Modify 3D Objects/Create Solid

Ribbon UI Menu:

In TurboCAD, "surface" refers to a 3D ACIS object, whereas a TurboCAD surface (when **TC Surface** is checked in the **3D** page of an object's **Properties**) refers to an object created with TurboCAD's internal engine, which is not an ACIS object. (See [3D Properties](#))

These are conversion operations that create surfaces or solids from existing objects, including surfaces, faces, and profiles. You can display the **Solid/Surface** toolbar by right-clicking in any toolbar area and selecting **Solid/Surface**.



Create Solid from Face(s)

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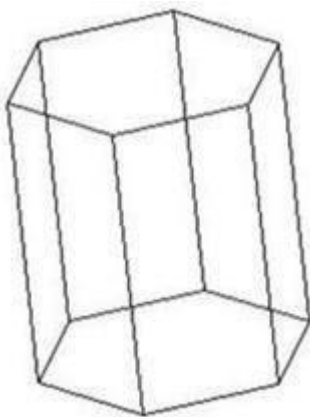
Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Modify/Modify 3D Objects/Create Solid/From Face(s)

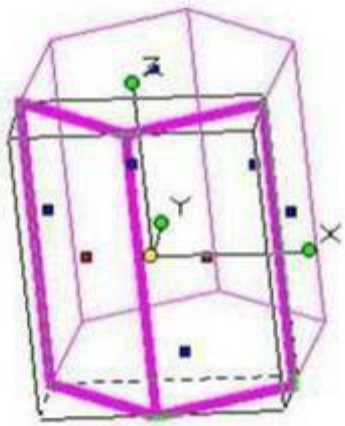
Ribbon UI Menu:

This tool works with ACIS objects only. It does not work with TC Surfaces. Enables you to convert one or more surfaces / faces into a solid. The faces must be contiguous, but they do not need to share boundaries over the entire boundary length.

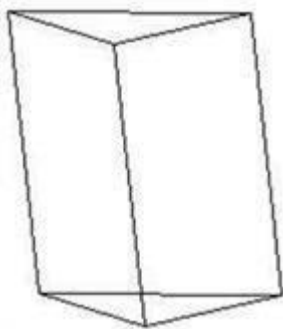
Start with one or more solids. This example uses a 6-sided polygon extruded (using **Simple Extrude**) into a solid.



Activate **Facet Edit** and select one or more faces. (See [Facet Edit](#)) Use Shift to select multiple faces.



Activate the **Solid from Face(s)** function, and the selected faces are now facets of a solid. Faces are added as needed to complete the solid.



You do not have to start with a solid; this function also works with objects that are already surfaces.

4. You will be prompted as to whether you wish to retain the original object.

Create Solid from Surface



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Modify/Modify 3D Objects/Create Solid/From Surface

Ribbon UI Menu:

This tool works with ACIS objects only. It does not work with TCSurfaces.

Enables you to convert a surface into a solid volume, within the surfaces' boundaries.

Start with a solid or surface. If you have a solid, select it and use **Create Surface From Solid** to convert it into a surface. This example uses a sphere.



Once you have a surface, activate **3D Slice** in order to cut the surface (see [3D Slice](#)). Make one or two slice lines.

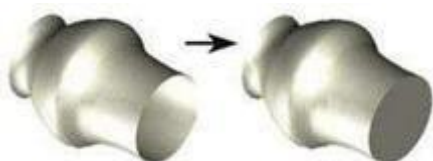
3. Select and delete the sliced portions.



Now select what's left of the surface and activate **Create Solid From Surface**. The surface is converted into a solid volume, within the boundaries of the original surface (holes are filled).



As a similar example, create a revolved surface (see [Profile Objects and Face to Face Modifications](#)). This is an ACIS object. Simply select it and make it a solid.



Create Surface from Face(s)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

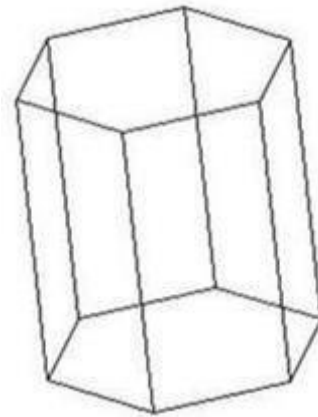
Default UI Menu: Modify/Modify 3D Objects/Create Surface/From Face(s)

Ribbon UI Menu:

Enables you to create a surface from selected planar facets of solid objects.

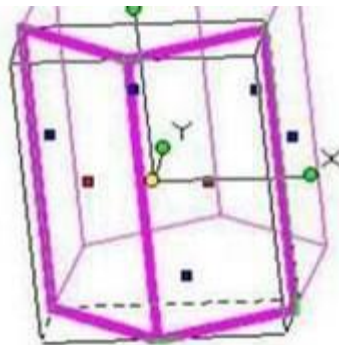
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Start with a solid that has planar facets. This example uses a 6-sided polygon



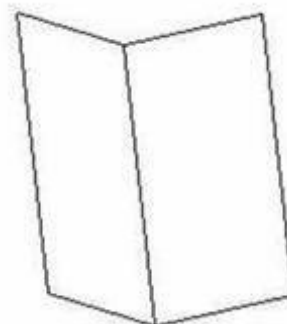
extruded (using **Simple Extrude**) into a solid.

Activate **Facet Edit** and select one or more faces. (See [Facet Edit](#)) Use Shift to



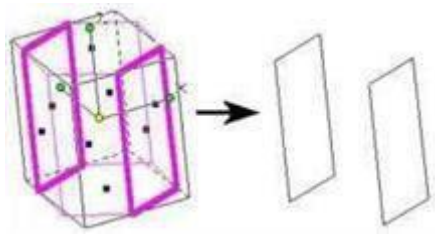
select multiple faces.

Activate the **Surface from Face(s)** function, and the selected faces are now



surfaces. The rest of the solid is deleted.

The selected faces do not have to be contiguous; surfaces can be created from any face(s).



4. You will be prompted as to whether you wish to retain the original object.

Create Surface from Profile



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

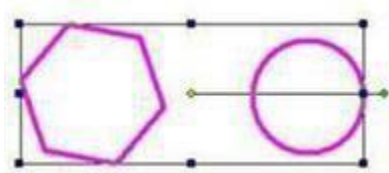


Default UI Menu: Modify/Modify 3D Objects/Create Surface/From Profile

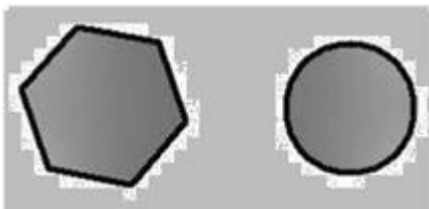
Ribbon UI Menu:

Enables you to create an ACIS surface within a closed profile. The profile can be 2D or 3D.

1. Select one or more closed profiles.



Activate the **Surface from Profile** function, and the boundaries are filled by one or more surfaces. You can see the surfaces in render mode.



Create Surface from Solid



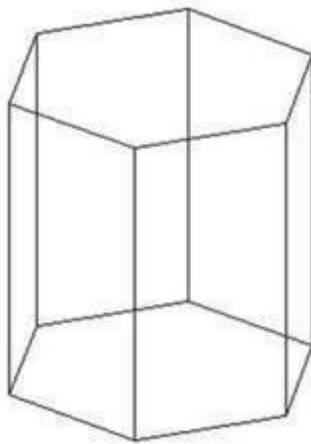
Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Modify/Modify 3D Objects/Create Surface/From Solid

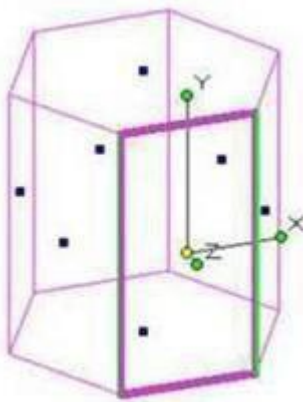
Ribbon UI Menu:

Enables you to convert a solid volume into a hollow form; each solid face is converted into a surface.

Start by selecting one or more solids. This example uses a 6-sided polygon extruded (using **Simple Extrude**) into a solid.



Activate the **Surface from Solid** function, and the solid is hollowed out. Each of its facets is now a surface. To more easily see the results, activate **Facet Edit** and select one of the vertical faces. (See [Facet Edit](#))

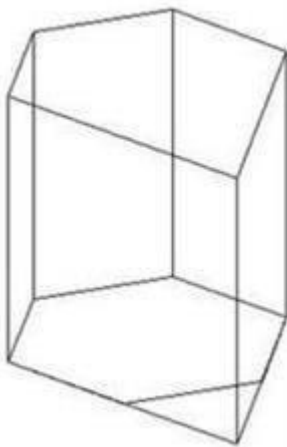


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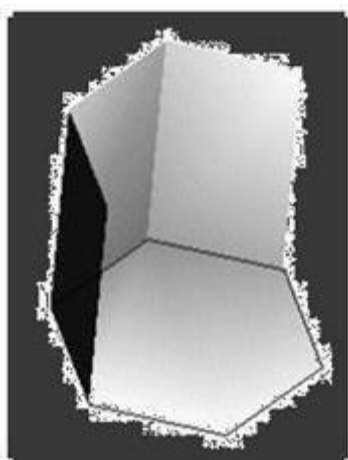
Delete the face (using the Del key), and view the model in render mode to see the missing facet.



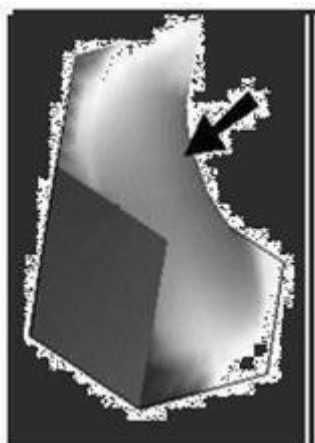
Note: If you used **Facet Edit** on a solid, without converting it into surfaces, this would be the result - the other faces would update to accommodate the deleted face.



One usage of surfaces is to recreate a solid from the changed volume. If you delete a few more faces ...



... you can then use **Create Solid from Surface** to add one missing face and produce a new solid. (See [Create Solid from Surface](#).)



10.17. Imprint Edge



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Imprint Edge

Ribbon UI Menu:

Tool for imprinting 2D objects (line, polyline, arc, circle, curve) into faces of ACIS objects. This creates separate areas on the imprinted face. These areas can now have different materials.

The imprinting object must meet the following criteria:

It must bisect the face either by being closed or by having each end terminate beyond the face.

It must be on the face to be bisected.

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Examples:

Primitive solids such as spheres, boxes, and cylinders must be exploded into an ACIS Solid in order for the Imprint Edge tool to work. If the object has not been exploded prior to selection you will be prompted with the following message when you select the first edge.

To use the Imprint Edge tool:

Select the tool.

Select the ACIS object which is to be imprinted.

If you wish to delete the source object for the edge select the **Delete source** option in the Local menu or the Inspector bar. This option is on by default

If multiple faces need to be selected, select the **Select faces to be imprinted** option in the Local menu or the Inspector bar.

If the **Select faces to be imprinted** option is on select the desired faces, then click Finish.

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If you accidentally select the wrong face/s use the **Deselect all selected faces** option in the Local menu or the Inspector bar, or click again on the faces you wish to deselect.

6. Click any edge you wish to imprint.

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Repeat step 7 as many times as you need.

Click Finish after you have select all the desired edges. You can now select another solid.

9. Press ESC or the Space bar to drop the tool.

You can now use the Drag on Facet feature of the Materials palette to put different materials sections of the face.

Options:

Delete source: When on the source objects for the edges will be deleted. It is on by default.

Select faces to be imprinted: When active this option allows you to select multiple faces to be imprinted.

Deselect all selected faces: This option allows you to deselect any currently selected faces. This option is only available when faces have been selected.

10.18. Extract Entity

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Extract Entity

Ribbon UI Menu:

The extract Extract Entity is used to derive 2D elements from 3D ACIS solids and to modify ACIS surfaces/sheet bodies.

ACIS surfaces/sheet bodies result when non-closed objects, such as a line segment, are extruded, revolved or swept. They can also be created from ACIS Solids by using the [Create Surface from Solid](#) tool, or by using the [Create Surface from Profile](#) tool on a closed 2D object, and by using the this tool.

Modes:

Extract Edge

In this mode the tool is used to extract an edge from a 3D entity.

To extract an edge:

Select the Extract Entity tool.

Select the Extract Edge mode from the Inspector bar or Local menu.

Hover over the edge you wish to extract, and click on it.

Result:

Options:

Extract Loop from Seed Edge and Face: When this option is on you can extract all connected edges on a selected face of the 3D object. Note: a Loop is the closed sequence of edges that bound a face.

To extract an edge loop:

1. Select the Extract Entity tool.

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Select the Extract Edge mode from the Inspector bar or Local menu.

Select the Extract Loop from Seed Edge and Face option from the Inspector bar or Local menu.

Hover over the face you wish to extract from, and click on it.

5. Hover over the edge on the loop you wish to extract, and click on it.

Result:

Remove Open Gap

In this mode the tool is used to remove holes in ACIS surfaces/sheet bodies.

To remove an open gap:

Select the Extract Entity tool.

Select the Remove Open Gap mode from the Inspector bar or Local menu.

Hover over the edge of the hole you wish to remove, and click on it.

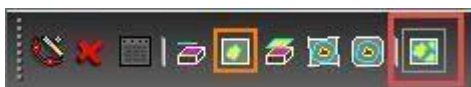
Result:

To remove an open gap with the Remove All Open Gaps option.:

Select the Extract Entity tool.

Select the Remove Open Gap mode from the Inspector bar or Local menu.

Select the **Remove All Open Gaps** option from the Inspector bar or Local menu.



Hover over the object with the holes you wish to remove, and click on it.

Result:

Extract Face

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In this mode the tool is used to extract a face from a 3D entity.

To extract a face:

Select the Extract Entity tool.

Select the Extract face mode from the Inspector bar or Local menu.

In the Inspector bar specify the Offset Distance at which the new sheet body will be created.



Hover over the face you wish to extract, and click on it.

Offset Face Loop

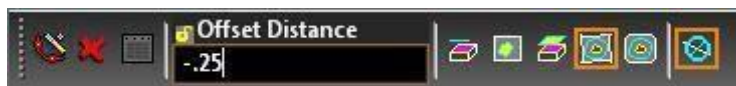
In this mode the tool is used to extract a face from a 3D entity by offsetting a closed loop around the hole in an object. This only works with ACIS surfaces/sheet bodies.

To Offset a face loop:

Select the Extract Entity tool.

Select the Offset Face Loop mode from the Inspector bar or Local menu.

In the Inspector bar specify the Offset Distance at which the new sheet body will be created. Negative values will make the hole larger, and positive values will make the hole smaller.



Hover over the face you wish to offset from, and click on it.

5. Hover over the edge of the hole you wish to offset, and click on it.

Result using a negative value (-0.25) as the offset.

Result using a positive value (0.25) as the offset.

Options:

Delete source object: When selected the original object will be deleted.

Offset Planar Face Loops

In this mode the tool is used to extract a face from a 3D entity by offsetting a the closed loops around the bounding the plane of the object. This only works with ACIS surfaces/sheet bodies.

To Offset a face loop:

Select the Extract Entity tool.

Select the Offset Planar Face Loop mode from the Inspector bar or Local menu.

In the Inspector bar specify the Offset Distance at which the new sheet body will be created. Negative values will make the hole larger, and positive values will make the hole smaller.

Hover over the face you wish to offset from, and click on it.

Options:

Delete source object: When selected the original object will be deleted.

Rounded corners: When selected the exterior corners (relative to the offset direction) will be rounded.

10.19. Twist Entity

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Twist Entity

Ribbon UI Menu:

Twists a solid object along and around the length of a designated axis. Twist actions can be stored in the Part tree, and adjusted later.

To twist an entity:

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Select the solid.

Select the first point for the axis, then the second point. or

Select the **Use line entity as axis** option in the local menu or Inspector bar, and click on a line.

Drag to set the angle and click to set the twist. or

Tab into the **Angle** field, type the value and press the **Enter** key.

You can also set the degree of continuity G0, G1, G2

G 0 – the curve or surface has continuous position; that is, it is a single piece with no gaps. **G1** – the curve or surface has continuous normals (tangent planes); that is, it has no sharp creases. **G2**– the curve or surface has continuous curvature.

G0	G1	G2

10.20. Stretch Entity

(Available in Platinum and Professional)

Default UI Menu: Modify/Modify 3D Objects/Stretch Entity

Ribbon UI Menu:

Stretches a solid object along the length of a designated axis. Stretch actions can be stored in the Part tree, and adjusted later.

To stretch an entity:

Select the solid.

Select the first point for the axis, then the second point. or

Select the **Use line entity as axis** option in the local menu or Inspector bar, and click on a line.

Drag to set the angle and click to set the twist. or

Tab into the **Angle** field, type the value and press the **Enter** key.

You can also set the degree of continuity G0, G1, G2

G 0 – the curve or surface has continuous position; that is, it is a single piece with no gaps. **G1** – the curve or surface has continuous normals (tangent planes); that is, it has no sharp creases. **G2**– the curve or surface has continuous curvature.

10.21. Gusset

(Available only in Platinum)

Default UI Menu: Modify/Modify 3D Objects/Gusset

Ribbon UI Menu:

This tool is used to create sheet metal gussets and stiffening ribs.

A key factor in using the Sheet Metal Gusset tool is that when the Gusset function is used the adjoining faces which are to be gusseted **must** be connect via a fillet or bend which has continuous thickness. The face which connects the two faces must have a cylindrical cross section.

Discontinuous thickness may result in many cases, such as fillet with non-corresponding centers, surfaces extracted from entities, and Boolean operations. Normal bends automatically will have continuous thickness.

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Yes	No

This is not an issue for the stiffening rib function

Gussets are created using a solid object and two points. The solid object must have flat faces, and the points must be placed adjacent faces which are connected via a single fillet or bend. In the case of stiffing ribs the point may be placed on directly adjacent faces.

To create a gusset:

Select the tool.

Select the Sheet Metal Gusset option.

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3. Specify the Width, Fillet Radius, and Angle.

Select the solid.

Select the first point.

Select the second point.

Click Finish.

To create a stiffening rib:

Select the tool.

Select the Stiffening Rib option.

Specify the Width, and Fillet Radius.

Select the solid.

Select the first point.

Select the second point.

Click Finish.

Options

Width: Sets the width of the gusset/rib.

Fillet Radius: Set the radius of the fillet radius for the edges of the gusset/rib.

Angle: Sets the splay Angle of the gusset. The value must be greater than zero.

Leave Source Copy: Leave the original solid object and make a copy of it with the gusset/rib.

In the part tree:

Editing the gusset points

If the Part Tree is active when the gusset/rib is created you can modify the gusset/rib by moving the defining points.

10.22. 3D Print Check Tool

3D Print Check Tool

(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Tools/3D Print Check Tool

Ribbon UI Menu:



Before sending your design to your 3D printer, gain valuable information about your 3D model regarding its printability. This new tool will effectively identify any issue in your design before sending it to the printer, and allow you to then fix it in TurboCAD. The Print Check tool provides General Information about select 3D Objects, as well as specific entity information including:

- Open Edges
- Non-Manifold Edges
- Collapsed Facets
- Duplicate Facets

When the tool is activated, icons will appear on the Inspector bar. These icons are used for the following functionalities:

- Select All Drawing
- Select Single Graphic
- Show General Information
- Open Edges
- Non-Manifold Edges
- Collapsed Facets

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Duplicate Facets

Using 3D Print Check Tool:

Enable 3D Print Check Tool from the Tools menu

Select object by "Select All Drawing" or "Select Graphic".

Select the desired option to display the information.

Inspector bar and Local Menu Options:

Select All Drawing: Selects every object in the drawing.



Select Graphic: Selects a specific graphic.

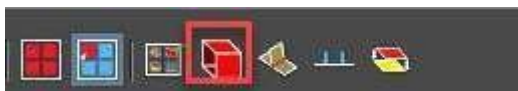


Show General Information: Shows general information in 3D Print Dialog.

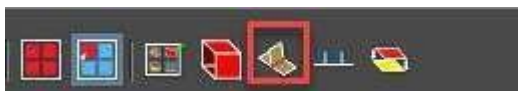


For example:

Select Open Edges: Displays Open edges.



Select Non-manifold Edges: Displays Non-Manifold edges.



Select Collapsed Facets: Displays Collapsed Facets.



Select Duplicate Facets: Displays Duplicate Facets and asks if you want to remove them.



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On clicking "Yes", duplicate facets are removed.

10.23. Generate Weight Report

Generate Weight Report



(Available in Platinum, Professional, and Deluxe)

Default UI Menu: Tools/Bill of Material/Generate Weight Report

Ribbon UI Menu:



A feature which can generate weight report of the solid objects with custom properties inserted.

To Generate a Weight Report and export is as a Bill of Materials (BOM):

For one or more 3D Solid Objects which have had Mass Properties assigned to them, from the Tools/Measurement menu click on the 'Generate Weight Report' icon

The following report appears:

To Save the BOM Report, click on the 'Export to CSV' button. The 'Save as' menu will appear and the report can be saved to the user's computer.

10.24. 3D Mapping

3D Mapping

Face UV Representation



Note: This manual is for TurboCAD 2021 Designer, TurboCAD 2021 Deluxe, TurboCAD 2021 Professional and TurboCAD 2021 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available only in Platinum)

Default UI Menu: Tools/Face UV Representation

Ribbon UI Menu:

Face UV representation is used to get a UV representation of the selected ACIS face as a separate object

Steps to create Face UV Representation:

1. Select Face UV Representation tool
2. Click on the face of the 3D object you want to create Face UV Representation of.
3. Place UV Representation on the desired location in the drawing.

UV representation has 4 properties: U scale, V scale, U period count, v period count

Local menu option:

Use Periodicity: It allows control UV representation, using periodicity of face equation.



Surface on Face



Note: This manual is for TurboCAD 2021 Designer, TurboCAD 2021 Deluxe, TurboCAD 2021 Professional and TurboCAD 2021 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available only in Platinum)

Default UI Menu: Tools/Surface on Face

Ribbon UI Menu:

Surface on Face is used to get 3D map surface from selected UV representation and 2d contour, which lies on the UV representation.

Creating Surface on Face:

1. Select "Surface on Face"
2. Select the Face UV representation object
3. Select the closed 2D entity lying on the Face UV Representation
4. Select/deselect closed 2D entity lying on the Face UV Representation or click Finish.

User can define thickness for 3d map surface. The resulting object is not a ACIS body. It is a TC surface

11. Architecture Tools

Default UI Menu: Architecture

Ribbon UI Menu:

TurboCAD provides specialized tools for architectural drawings, allowing you to quickly create and edit walls, window and door openings, and roofs.

You can display the **Architecture** toolbar by right-clicking in any toolbar area and selecting **Architecture**.



Note: The **Architecture** menu also contains **Markers**. See [Drawing Markers](#)

Note: To get started quickly with architectural tools try using the Architectural template. It has many of the features and settings for Architectural objects and styles preset.

LTE Workspace

In the LTE Workspace tools behave with slight differences to standard TurboCAD methods.

- They allow use of dynamic input.

- They automatically drop when the action is completed, whereas standard TC tools continue until you change the tool.

- They have a Multiple option (local menu) that turns off auto-dropping for the individual tool.

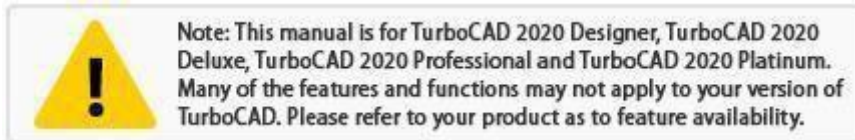
- You can complete drop the tool by pressing the either the Enter or Space keys.

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5. You can re-activate the tool by pressing the Space Key again.

11.1. House Wizard Toolset

(Available in Platinum, Professional and Deluxe)



Default UI Menu: Architecture/House Wizard

Ribbon UI Menu:

The House Wizard is a set of tools designed to allow for accelerated layout of a house design. It is structured to provide a set of room boxes that can be snapped together quickly. Once the initial layout of spaces is complete the click of a single button will generate a preliminary 3D model of the house, including walls, slabs, and doors. You can then quickly finalize the model to your needs. In order for the House Wizard to work correctly you must start with either the "House Template", or the "House Template Metric".

To get the maximum utility out of the House Wizard you should turn on the House Wizard toolbar. Go to **Customize**, navigate to the Toolbars page and select House Wizard.



Arranging and Snapping Room Boxes

To arrange the room boxes into your design simply pick up each box and drag it into position, and drop it.

If you position the boxes so that are aligned closely edge-to-edge the dropped box will "snap" to the adjacent box with edges aligned, and if possible with corners aligned.

It is a good idea to turn Off Dynamic snaps while you are arranging your boxes, since the two types of snapping may conflict in some instances.

You can resize the boxes while they are selected by using the node edit blue handles on the edge of the selector. If you hover over the edge of an adjacent box while resizing

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the edge you are adjusting will snap to align with the edge you are hovering over.

For best results, it is suggested that you put small ancillary rooms within larger rooms e.g. bedrooms.

Once you are done arranging your rooms, click the Build House button.
Here is an example of some rooms laid out together:

Notice the overlaps between the rooms.
Here is the result after pushing the Build House button.

Notice how the House Wizard has automatically add room dimensions, calculated the individual room areas, and automatically placed doors at logical positions within the house.

Multiple Floors

To create multiple floors via the house wizard place room boxes on different workplaces at different elevations.

Changing Room Size While Inserting

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When you are using the Insert Room tools you can dynamically change the size of the room you are inserting by changing the Width and Length parameters in the Inspector bar, or by using the local menu to access the tools properties.



Delete Room Boxes

Default UI Menu: Architecture/House Wizard/Delete Room Boxes

Ribbon UI Menu:

This will delete all of the room boxes in your drawing.

Warning: You will not be prompted to verify that you want to delete the boxes, so if you accidentally delete the boxes, use Undo immediately.

Hide-Show Room Boxes

Default UI Menu: Architecture/House Wizard/ Hide/Show Room Boxes

Ribbon UI Menu:

This option will hide room boxes if they are currently showing, or it will show room boxes if they are currently hidden. This function does not override layer visibility.

House Setup

Default UI Menu: Architecture/House Wizard/House Setup

Ribbon UI Menu:

This will open the House Wizard Setup dialog. With this dialog you can specify which style elements will be used to generate each of the components of your house. Changes in these settings will only be reflected in your model after the design is rebuilt

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with Build House.

House Wizard

Default UI Menu: Architecture/House Wizard/House Wizard

Ribbon UI Menu:

The first step in using the House Wizard toolset is to use the House Wizard itself. The Wizard will generate all of the room boxes that you select.

The Page One of the wizard is simply an introduction, Click Next to move to Page two

Page Two of the wizard is used to specify the number and size of the Garages and Main Rooms in your house design. The Garage drop-down controls allow you to specify the number of cars the garage is to hold. The Bedroom and Bathroom drop-down controls allow you to specify the number of those rooms you want. You can specify the additional rooms you want using the check boxes on the page.

While on this page you can use the Width and Length fields to override the default sizes for the rooms.

Click Next to go to Page Three of the Wizard.

On Page Three of the Wizard you can specify the size and number of decks you want in your design, as well as miscellaneous rooms and passageways. For all of these areas, you have the option of choosing None.

Click Next to go to Page Three of the Wizard.

Page Four of the Wizard gives you some final tips on how to use the boxes that the wizard generates. Read the info carefully then click the Finish button.

The result of the Wizard will be a 2D layout of all the boxes you generated.

Insert Room Boxes

Default UI Menu: Architecture/House Wizard/Bedroom

Ribbon UI Menu:

You do not have to use the House Wizard to generate your rooms, nor do you have to reuse the Wizard if you forgot a room. The Insert Room Box tools e.g. Insert Bedroom

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will allow you to add room boxes to your layout at any time.

Simply select an Insert Room Box tool then click at each location that you want to place that room type. The tool will remain active until you press ESC or choose a different tool.

Room Box Properties



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Most Room Box properties are the same as any standard 2D object, but there are a few special properties.

Type: Contains the room type designation for the box. This cannot be edited.

Text Height: Controls the text height of the text label of the box.

Width: Sets the width of the room box.

Length: Sets the Length of the room box.

Name: Sets the contents of the text label of the room box.

11.2. Openings

(Available in Platinum)

Default UI Menu: Architecture/Doors, Windows and Openings

Ribbon UI Menu:

Enables you to insert a 2D / 3D opening into a wall, roof, or roof slab representing a window, door, etc.

Note: To insert actual "smart" windows and doors, see [Windows and Doors](#)

In this tool, each opening is represented by two blocks - a 2D and 3D view, so that the block can be seen in both orthographic and isometric views. The blocks must be created in advance before they can be inserted into a wall. See [Creating a Block](#)
The blocks do not have to be the same thickness as the walls, roofs, or roof slabs they will be inserted into.

Edit Opening Modifier

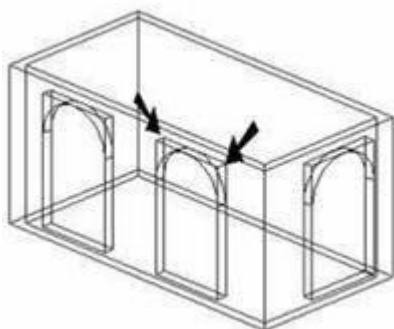


Default UI Menu: Architecture/Doors, Windows and Openings/Edit Opening Modifier

Ribbon UI Menu:

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By default, when using the **Insert Opening** tool, an opening is inserted into a wall by creating a rectangular cutout, even when the block is not rectangular, as in the case of an arched doorway or a circular window.



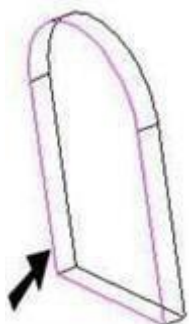
For non-rectangular openings, you can apply a modifier that will define the shape of the cutout.

In the Blocks Palette, select the 3D block whose cutout you want to modify, and



click **Edit Contents**.

Create the closed, 2D object you want to use as the cutout. Change the workplane if necessary. In this example, a polyline was added to the door face -



an exact trace of the doorway shape.

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Activate **Edit Opening Modifier**. If a modifier was previously defined, it will be highlighted.

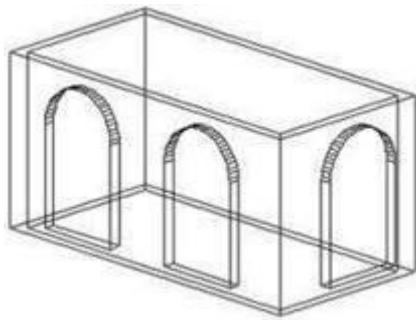
Select the object to be used as the modifier.

Select **Finish** from the local menu or Inspector Bar.



In the Blocks Palette, click **Finish Edit Content**

The openings now show the revised cutout shape.



Note: If you want to continue adding more openings, you will have to change the workplane back to **Workplane by World**.

Insert Opening

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Default UI Menu: Architecture/Doors, Windows and Openings/Insert Opening

Ribbon UI Menu:

Options:

Anchor to Wall / Anchor to Roof or Roof Slab - Specifies whether the open will snap to walls or roof objects.

Elevation - Distance off of the workplane at which the opening will be set.

Offset - Distance to the first adjacent wall. You can use this to exactly place the opening. however you can only specify one Offset: Offset or Offset 1

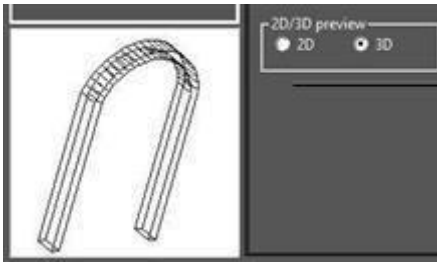
Offset 1 - Distance to the second adjacent wall. You can use this to exactly place the opening.

In this example, the opening has the following properties: For 2D representation: a block consisting of three lines



For 3D representation: a block consisting of an arched doorway, created by making a polyline (see [Polyline](#)) and extruding it (see [Profile Objects and Face to Face Modifications](#)).

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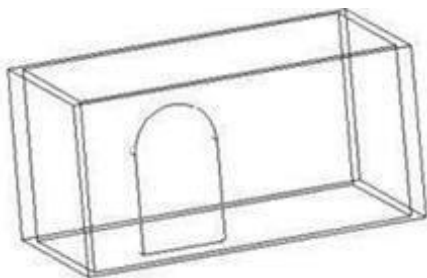
A value was assigned for **Height**, so that the block will be inserted above the floor. Remember, the blocks have to be created in advance, and are assigned to the opening in the **Opening Properties**.

If necessary, change the workplane back to **By World**. You may have changed the workplane while creating one of the blocks, but the workplane must be along the bottom of the walls.

Activate **Insert Opening**, and make sure **Anchor to Wall** is selected. (**Anchor to Roof or Roof Slab** is similar, except that openings will tilt themselves to fit along a sloped roof.)

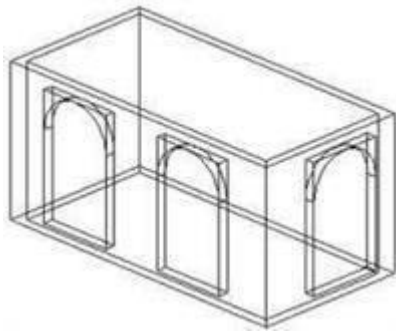
Drag the opening to one of the walls. The block is dragged by its reference point, which can be changed if needed. See [Editing a Block](#)

The block aligns itself to the wall, and cuts the opening.



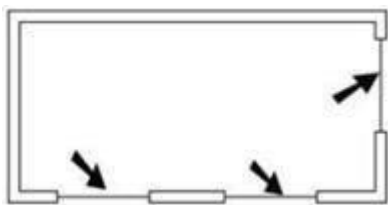
5. Drag more blocks onto the walls. The block will change its alignment to fit any

wall.



Note: This tool only creates **rectangular** cutouts, even if the block is non-rectangular. You can change this by modifying the block. See [Edit Opening Modifier](#)

In **Plan** view, the openings are represented by the 2D block.



If you assign the 3D block a material such as glass (in the **3D** page of its **Properties**), and assign a brick material for the walls, you can see the results in render mode.



Opening Properties

Before an opening can be inserted, you must select the 2D and 3D blocks (previously created) that will be used to represent it. This is done in the **Properties** window - opened by right-clicking on the **Insert Opening** icon.

2D/3D representation: The blocks that will be used to represent the opening. The 2D block will be used in plan views; the 3D block will be used in all other views.

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2D/3D preview: The preview is displayed on the left side of the window, so that you can verify the selected block is correct.

Position / Height: The distance from the workplane where the opening will be inserted. This is useful for windows that need to be located a certain height from the floor.

Auto Modifier: Inserting an opening with a non-rectangular outline, if checked.

The Auto Modifier option to the Openings Properties allows the Opening to be inserted with a non-rectangular outline.



Tip: If you want to create and save more than one type of opening, create separate styles (accessed from the **General** page of the **Properties** window). This way you will not have to recreate properties of openings you have already inserted. See [Property Value Presets](#)

11.3. Railings

(Available in Platinum)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Architecture/Stairs and Railings

Ribbon UI Menu:

Inserts a linear railing or a railing along a staircase. This tool works in conjunction with the Style Manager. See [Rail Styles](#)

11.4. Roofs



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Architecture/Roofs

Ribbon UI Menu:

You can easily add and modify roofs to a building. The roof slope can be constant or you can change the slopes of individual roof faces.

Add Roof

(Available in Platinum, Professional and Deluxe)



Default UI Menu: Architecture/Roofs/Add Roof

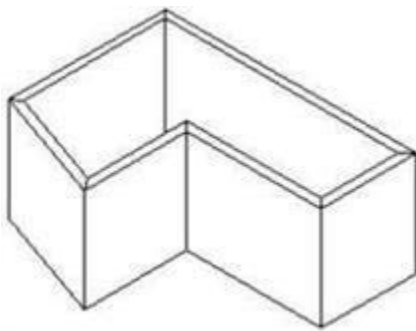
Ribbon UI Menu:

Creates a roof with a constant slope angle.

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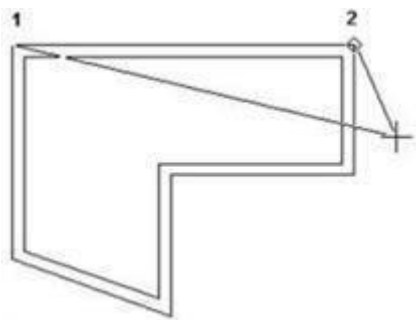
Create the walls of the building. If you do not use the **Wall** tools, you can use any 3D shape, or even a 2D outline.

Note: If you use the **Wall** tool, you can give the walls height by entering a **Thickness** in the **3D** page of the walls' **Properties**.

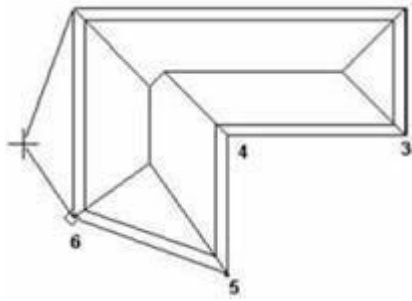


If necessary, bring the workplane to the correct level. You can use **By Facet** to set the workplane to the top of the walls.

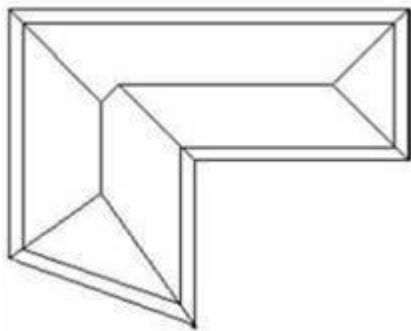
Activate **Add Roof**, and trace the outline of the roof.



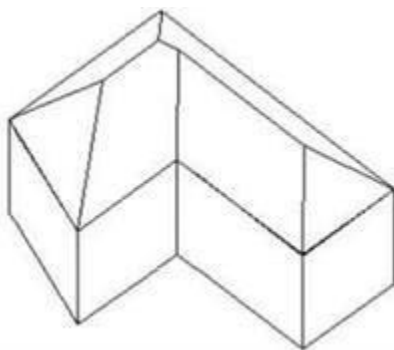
With each corner point you select, the roof adjusts to create a closed form. .



After you select the last point, select **Finish** from the local menu or Inspector Bar. The ridge and valley lines are calculated to maintain a constant slope angle.



Rotate the view to see the roof.



Add Roof by Walls

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(Available in Platinum, Professional and Deluxe)



Default UI Menu: Architecture/Roofs/Add Roof by Walls

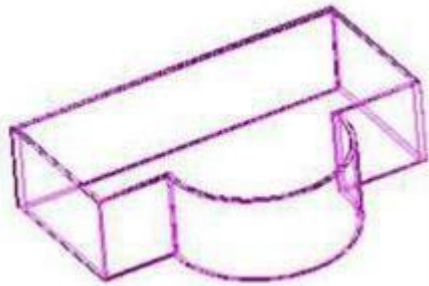
Ribbon UI Menu:

Creates a wall defined by a series of walls.

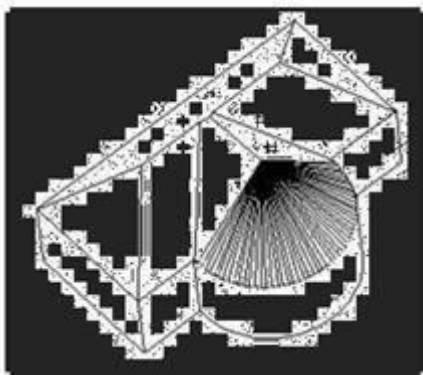
Start with a closed chain of walls, and activate **Add Roof by Walls**.

In the Properties, set the **Slope Angle** and **Overhang**.

Press Shift and select all walls in the chain.



4. Select Finish to create the roof.



5. More than one roof can also be created by selecting all walls.

Previously, you could create only one roof object. Now more than one roof can be created by selecting more than one wall.

Demo Video:

<https://youtu.be/-1DFP9HzPY8>

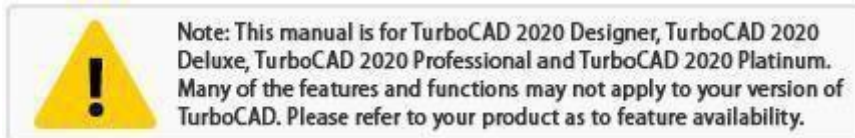
Add Roof Modifier

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(Available in Platinum)

Default UI Menu: Architecture/Roofs/Add Roof Modifier

Ribbon UI Menu:



The Add Roof Modifier tool allows you to use 2D objects to trim and make holes in roofs.

The 2D objects must be closed, and must be on the workplane, not on the roof planes.

To add a roof modifier.

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Click on the roof to select it.

Click on the object you wish to act as a modifier.

Repeat steps 1-2 for each modifier you wish to add.

The modifier remains associative to the roof. Moving it or modifying it will be reflected onto the roof.

Edit Slope Angle

(Available only in Platinum)

Changes the slope of individual roof faces.

1. Select the roof face whose slope you want to change.



Enter the new slope angle in the Inspector Bar. Or

Enter a new Rise in the Inspector bar. The Run value is fixed.



Node Editing Roofs

(Available only in Platinum)

Just as with other objects you can node edit roofs.

Blue nodes represent the layout of the roof they must remain at the same level.

Green nodes represent the roof peaks, they can be set to any position with two limitations:

Peak nodes cannot be moved outside of the boundaries defined by the blue nodes. In other words you cannot place the peak nodes in a way that will create an upside down roof.

Paired nodes (two adjacent nodes at the same level) must remain at the same level. In short, only logical roof solutions will be allowed.

You can add nodes to a roof layout by hovering over the along the side of the roof and selecting Add Node or Divide Segment. Divide Segment will put a new node exactly in the middle.

For the option to be available the cursor must be over the layout of the roof, not over the overhang.

You can drag a window around a node (or nodes) to select them.

When nodes are selected The Delete Node(s) option will appear in the local menu.

You cannot delete peak (Green) nodes on a roof.

Roof Properties

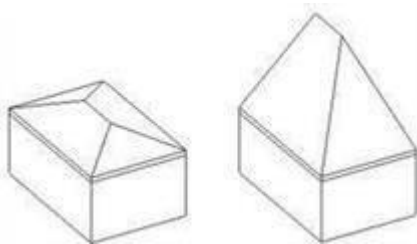
(Available in Platinum and Professional)

Thickness, slope angle, and overhang length of the roof.

Thickness: The thickness of the roof.

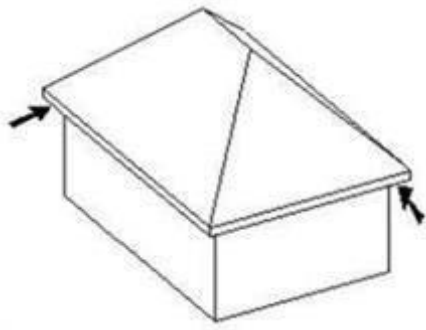
Rise/Run: Specifies the angle of the in standard ratio format. For English units the ratio is X over 12 (X/12). For metric the ratio is X over 100(X/100).

Slope: Specifies the angle of the roof. The higher the angle, the steeper the roof slope. Adjusting this value will alter the Rise/Run.



Note: This parameter changes the slope of the entire roof. To change the slope of one facet, use the **Edit Slope Angle** tool.

Overhang: The distance the roof extends past the defined roof outline.



From the Selection Info palette you will find the additional property Edge Cut.

Edge Cut: Provides two options: Plumb, and Square.

When set to plumb the edge will be perpendicular to the ground (vertical)

When set to Square the edge will be perpendicular to the plane of the associated roof section.

Roof Components

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The Roof Components properties allow for the setting of different Materials to each component of the roof. The components available are as Top, Bottom, Gable and Fascia

Roof Slabs and Converting to Roof Slabs

(Available in Platinum and Professional)

Roof slabs are used to create complex compound roof types that are not possible for the Roof tool. E.g. Skillion (Lean-to, Shed), Saltbox, M-Shaped, Gambrel, Mansard, Butterfly, Combination (Bonnet), Dormer, Dutch Gable, Jerkinhead, Flat, Cross Gabled. Roof Slabs are created from closed linear polylines (no arcs) or standard roofs.

To Convert to a Roof Slab:

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Select the Convert to Roof Slab tool.

Click on a closed polyline.

Or

Select the Convert to Roof Slab tool.

Click on a standard roof.

Roof Slab Properties

Roof Slabs have the following properties. As of now the can only be accessed through the Selection Info Palette.

Thickness: The thickness of the roof.

Slope: Specifies the angle of the roof. The higher the angle, the steeper the roof slope. Adjusting this value will alter the Rise/Run.

Run: Specifies the Run value for the angle of the in standard ratio format, Rise/Run

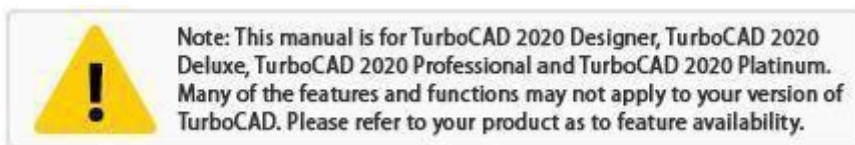
Rise: Specifies the Rise value for the angle of the in standard ratio format, Rise/Run

Top: Used for setting of the materials for this component of the roof slab.

Bottom: Used for setting of the materials for this component of the roof slab.

Fascia: Used for setting of the materials for this component of the roof slab.

Extend Roof Slab



Default UI Menu: Architecture/Roofs/Extend a Roof Slab

Ribbon UI Menu:

The Extend Roof Slab tool to extend a slab or roof slab to a plane defined by another slab, a roof slab, or a wall. The target itself does not have to intersect with the extended slab/roof slab, as long as it projects an intersection in the current user coordinate system (UCS).

To Ext end a Roof Slab:

1. Select the target flat plane. It will be highlighted in green.

2. Select the first edge to lengthen

3. Select the second edge to lengthen

4. The result will look like this:

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Another Example:

Notice that the roof slab has extended to where the wall would be if it were that tall.

Miter Two Roof Slabs

Default UI Menu: Architecture/Roofs/Miter Two Roof Slabs

Ribbon UI Menu:

The Miter Two Roof Slabs tool allow you to bring two Roof Slabs together with a smooth joint.

The tool has a single option in the local menu:

By Intersection

Use this option when you are mitering two intersecting roof slabs. Both roof slabs are trimmed along the intersection of their perimeter lines. Vertices are added or removed as required. The new edges of the roof slabs are set to angles required for a true miter cut. If the slab thicknesses of the two edges are different, the angles will be different. You cannot use this option if the slabs do not intersect.

To Miter Two Intersecting Roof Slabs:

Select the first roof slab on the part of the slab you wish keep. The opposite side will be trimmed.

Select the second roof slab on the part of the slab you wish keep. The opposite side will be trimmed.

3. The result will look like this.

If the two Roof Slabs do not intersect you can miter two slabs by selecting edges to extend and meet (similar to the 2D Meet 2 Lines tool). Only one edge of each roof slab is mitered. You can miter edges of roof slabs without altering the entire slab. To accomplish this, however, you may need to insert additional vertices.

Make sure that the By Intersection option is off in the local menu.

Select one edge on the first roof slab.

3. Select one edge on the second roof slab.

4. The result will look like this:

Another example:

Node Editing Roof Slabs

You can Edit Roof Slabs just as you can edit walls and most objects. You can:

- Select a Node and delete it using the local menu.

- Right click on an edge and add a node from the local menu.

- Select and move node/s.

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Select the arrow on an edge to drag it to expand or reduce the slab.

Editing Roof Slabs also has several special features.

Set the Pivot Edge: This specifies the edge which will serve as the pivot when any future adjustments are made to the Slope or Rise-Run properties. The green line indicates the pivot edge.

Set the Edge Cut: This allows you to set each edge of the slab to be Plumb or Square. By default the value is set to Square.

Set the Edge Cut Angle: This allows you to the angle between the top plane of the slab and the edge plane if the Edge Cut is set to Square, or between the ground plane the edge plane of the slab if the Edge Cut is set to Plumb. By default the value is set to 90.

To Set the Pivot Edge

1. Select the Roof Slab with the Edit tool
2. Select Pivot Edge Edit from the local menu.

3. Hover over the desired edge. It will be highlighted in red.

4. Click to set the edge.

5. The green indicator moves.

To Set the Edge Cut

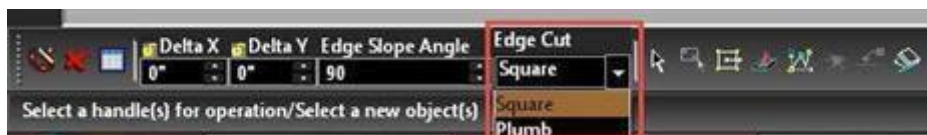
1. Select the Roof Slab with the Edit tool

Hold down the Shift key

Hover over the desired edge. It will be highlighted in white.

Click on the edge.

From the Inspector bar select Plumb from the Edge Cut drop-down.



The result will be a Plumb edge.

To Set the Edge Cut Angle

1. Select the Roof Slab with the Edit tool

Hold down the Shift key

Hover over the desired edge. It will be highlighted in white.

Click on the edge.

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From the Inspector bar type 60 into the Edge Cut Angle field.



Press Enter.

Trim Roof Slab by Plane



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Architecture/Roofs/Trim Roof Slab by Plane

Ribbon UI Menu:

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This tool allows you to trim a Roof Slab by the plane of another object. The objects that can be used are: Other Roof Slabs, Standard Roofs, or Walls. To Trim a Roof Slab:

1. Select the Trim Roof Slab tool.

2. Select the slab.

3. Select the plane to be used as the trimming element.

4. Select the side of the Roof Slab to be trimmed away.

11.5. Section-Elevation

(Available in Platinum)

Default UI Menu: Architecture/ Section/Elevation

Ribbon UI Menu:

There are three tools used to create Section-Elevations.

Horizontal



Default UI Menu: Architecture/ Section/Elevation / Horizontal

Ribbon UI Menu:

Create a horizontal section.

Create a 3D element. Now use section line tool (See [Section Line](#)) that cross the pertinent 3D elements.

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Select horizontal section tool. Right click to open the local menu. Select either 2D Section /Elevation or 3D Section /Elevation.

Alternatively, you can also select 2D Section /Elevation or 3D Section /Elevation from the inspector bar.

Select the Section line.

Select the pertinent 3D elements.

Select finish from the local menu or in the inspector bar. You will then have the section and will be prompted to select section position. Click in the drawing to place the section.

You can edit the size, orientation and position of the section line using the edit tool. (see [Edit Tool](#))

Section Line



Default UI Menu: Architecture/ Section/Elevation / Section Line

Ribbon UI Menu:

Create a section line.

Define the segment start point.

Define the end point, or specify the length and angle in the inspector bar.

(See [Vertical](#) and [Horizontal](#) for details on how to use the section line.)

Vertical



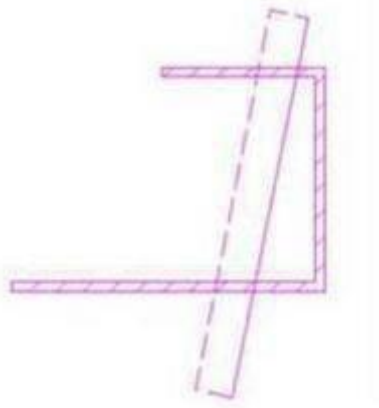
Default UI Menu: Architecture/ Section/Elevation / Vertical

Ribbon UI Menu:

Create a vertical section.

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Create a 3D element. Now use the section line tool (See [Section Line](#)) to cross the pertinent 3D elements.



2.

Select vertical section tool. Right click to open the local menu. Select either 2D Section /Elevation or 3D Section /Elevation.

Alternatively, you can also select 2D Section /Elevation or 3D Section /Elevation in the inspector bar.



Select the Section line.

Select the pertinent 3D elements.

Select finish from local menu or in the inspector bar. You will then have the section and will be prompted to select section position. Click in the drawing to place the section.

You can edit the size, orientation and position of the section line using edit tool. (see [Edit Tool](#))

11.6. Schedule

(Available in Platinum, Professional and Deluxe)



Default UI Menu: Architecture/Schedule

Ribbon UI Menu:

Inserts a schedule - a table detailing all selected windows and doors. This tool works in conjunction with the Style Manager. See [Schedule Styles](#)

Note: The **Fill Schedule Wizard** is another way to add a schedule, that scans for objects on specific layers. See [Schedule Styles](#)

11.7. Slabs

(Available in Platinum and Professional)

Default UI Menu: Architecture/Slabs

Ribbon UI Menu:

You can add a slab by clicking inside a room of closed walls, or by selecting a closed polyline. You can also create holes in slabs.

Before creating slabs, you should define slab styles. See [Slab Styles](#)

Add Hole

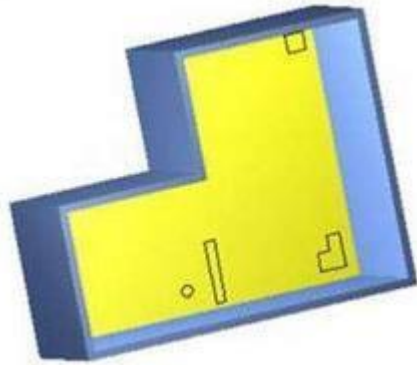
Default UI Menu: Architecture/Slabs/Add Hole

Ribbon UI Menu:

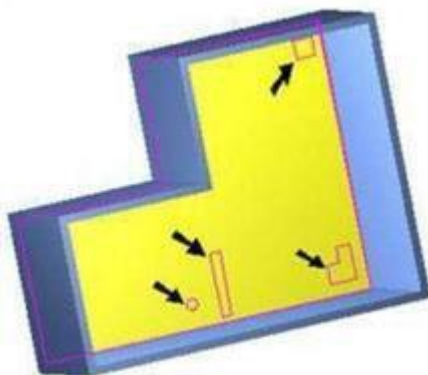
Creates holes in slabs or roof slabs, defined by closed curves or polylines.

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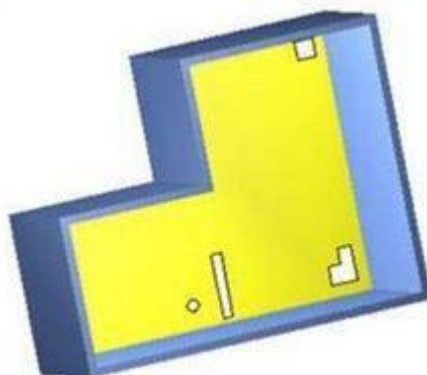
Start with a slab, and use one or more polylines or curves (rectangles, circles, etc.) to define the holes.



Activate **Add Hole** and click the slab on which you want to create holes. Then click the object representing the hole. If you want to create multiple holes, press Shift and click all of the hole objects.

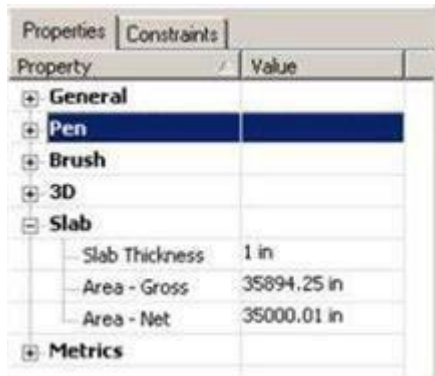


Select **Finish** from the Inspector Bar or local menu. The material inside the hole objects is removed.



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In the Selection Info palette, **Area - Gross** represents the area of the slab not including hole removal. **Area - Net** is the total area, with holes removed.



Properties		Constraints
Property	Value	
+	General	
+	Pen	
+	Brush	
+	3D	
-	Slab	
	Slab Thickness	1 in
	Area - Gross	35894.25 in
	Area - Net	35000.01 in
+	Metrics	

Add Slab by Click



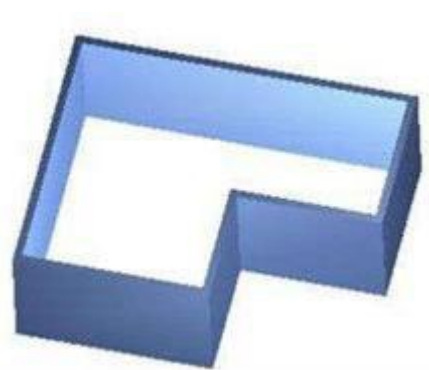
Default UI Menu: Architecture/Slabs/Add Slab by Click

Ribbon UI Menu:

Creates a slab in a room, defined by a closed set of walls.

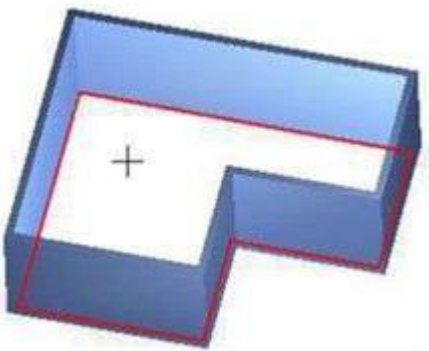
1. Start with a set of closed walls.

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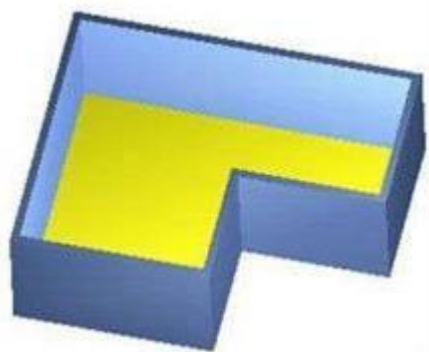


Open the tool's **Properties** and define the slab style on the **General** page. See [Slab Styles](#). You can also change a slab's style using the Selection Info palette.

Activate **Add Slab by Click** and click anywhere inside the room.



The slab is created inside the room, between the walls.



You can also specify to offset the slab from the walls by using the **Offset** field in the Inspector bar.



The thickness and area of the slab are listed in the Selection Info palette.

Local Menu Option

Use Wall Thickness: This option can be useful using walls with different widths. If this option is selected, the offset value for each side of the slab will be calculated separately as $\text{offset} = \text{offset} * \text{wall\_width}$. In this case, offset value is a dimensionless quantity and acts as a multiplier of the wall width.

Convert to Slab



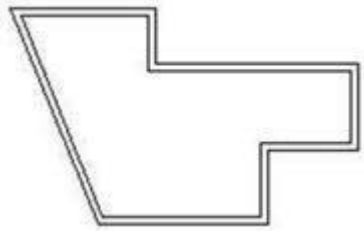
Default UI Menu: Architecture/Slabs/Convert to Slab

Ribbon UI Menu:

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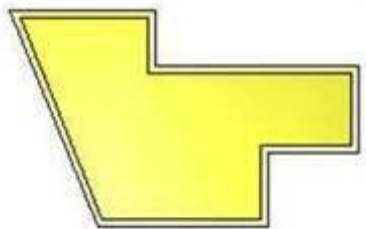
Creates a slab inside a closed curve (rectangle, circle, etc.) or polyline. The curves or polylines can be created using double line tools.

Start with a closed curve or polyline. This example uses a polyline created with **Double Line Polyline**.



Open the tool's **Properties** and define the slab style on the **General** page. See [Slab Styles](#). You can also change a slab's style using the Selection Info palette.

Activate **Convert to Slab** and click the polyline. The slab is created inside the polyline.



To create walls from the double lines, you can use **Convert to Wall** (see [Convert to Wall](#)).



Delete Hole

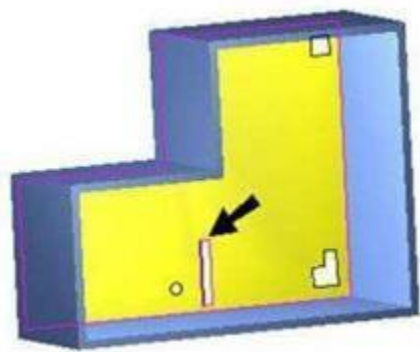


Default UI Menu: Architecture/Slabs/Delete Hole

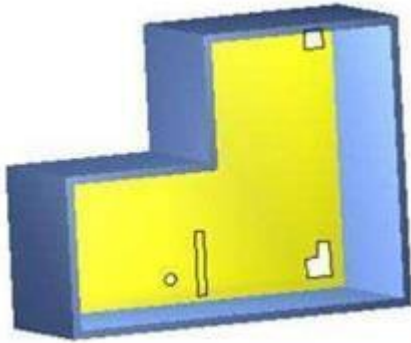
Ribbon UI Menu:

Removes holes already defined in a slab.

Start with a slab that has one or more holes defined. Activate **Delete Hole** and click the slab. Then click the object or objects whose hole you want to remove. If you want to remove more than one hole, press Shift and click multiple objects.



2. Select **Finish** from the Inspector Bar or local menu. The hole is removed.



11.8. Stairs

(Available only in Platinum)

Default UI Menu: Architecture/Stairs and Railings

Ribbon UI Menu:

These tools enable you to create four types of staircases: straight, spiral, multi-landing, and U-shaped.

Editing Stairs

The **Edit Tool** can be used to move or resize walls.



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This example shows a stair. Activate the **Edit Tool** and click the stair you want to

change.

If you click and drag a node, you can move it anywhere.

The Inspector bar gives you the option to apply a specific delta x and y for any selected node. It also gives the ability to edit the Height of the stairs.



Multi Landing Stair

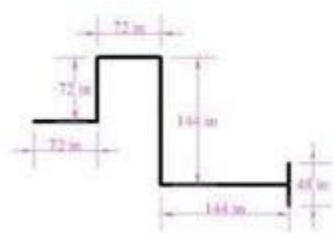
Default UI Menu: Architecture/Multi Landing Stair

Ribbon UI Menu:

Creates two or more lines of stairs.

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For reference, it may be helpful to lay out the stairs using lines in **World Plan**. In this example, the right-most vertical line represents the width of the stairs, and the other lines represent the path of the stair case.

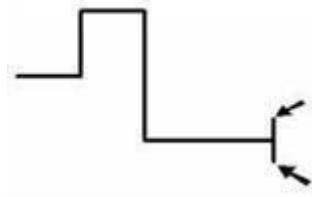


Activate **Multi Landing Stair** and open the **Properties** to the **Stair** page. This example shows **1/2 Landing**. The remaining parameters control the vertical and horizontal dimensions of each stair.

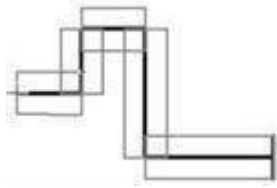
Note: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See [Stairs Styles](#). Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

The first two clicks define the width of the stairs, or you can use the Inspector Bar. (If this value is different than the **Base Width** set in the **Properties**, then **Base Width** will be overridden. However, you can always change this value later in the **Properties**.)

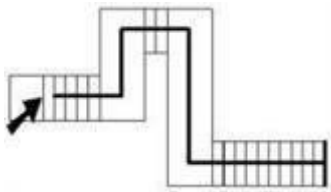
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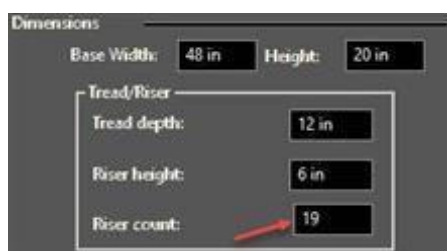
The next clicks, or values in the Inspector Bar, define the path of the stairs. This example uses five segments.



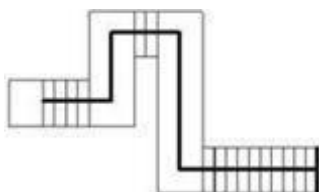
Select **Finish** to create the stairs. In **World Plan**, you can see whether the number of stairs is correct. In this example, there is one stair too many.



6. To correct this, open the stair's **Properties** and set the correct **Riser count**.

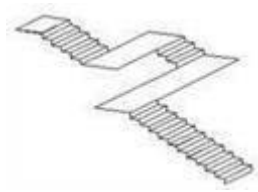


Now the number is correct.

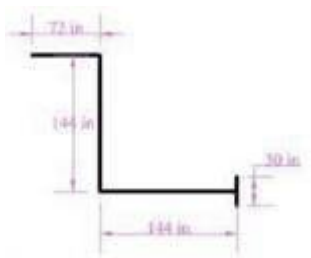


In Isometric view, this is the **1/2 Landing** staircase. The second, fourth, sixth, etc. segments of the stair path are created as landings.

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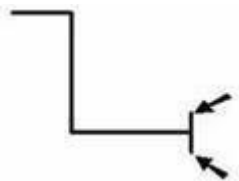


For the other turn types, each segment is created as a staircase. So start with a stair line like this, with fewer segments.

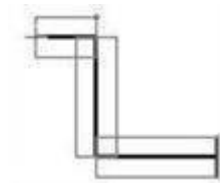


Select **1/4 Turn**, and set the **Riser Count** to an approximately accurate value (it can always be corrected later).

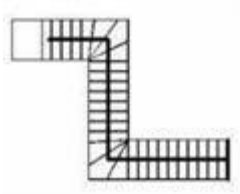
10. As before, set the width of the stairs.



11. Then select the stair path.



12. Select **Finish**, and adjust the **Riser Count** if necessary.



This is the **1/4 Turn** staircase in Isometric view. There are no landings; the stairs proceed along the turns.



This would be the result if you selected **1/4 Landing**. Each segment is a staircase, and square landings are created between each segment.



MultiLanding Stair by Path

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Architecture/Multi Landing Stair by Path

Ribbon UI Menu:

Creates stair based on a pre-existing 2D Polyline compose of line segments. Don't use arc segments. 3D polylines may be used but only their x, y coordinate geometry will be used in the creation of the stair.

1. In **World Plan** draw a polyline that will be used to create the stair.

Activate **MultiLanding Stair by Path** and open the **Properties** to the **Stair** page. This example shows **1/4 Landing**. The remaining parameters control the vertical and horizontal dimensions of each stair. Close Properties. For most multilanding stairs 1/4 landing is the best option.

Note: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See [Stairs Styles](#). Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

3. Click on the polyline.

A preview of the stair will appear.

Right click and select Finish to complete the stair.

Stair with reference polyline.
Modified to 1/2 Landing.

Spiral Stair

Default UI Menu: Architecture/Spiral Stair

Ribbon UI Menu:

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Creates a spiral staircase.

Activate **Spiral Stair** and open the **Properties** to the **Stair** page. This example will be **Clockwise**. The remaining parameters control the vertical and horizontal dimensions of each stair.

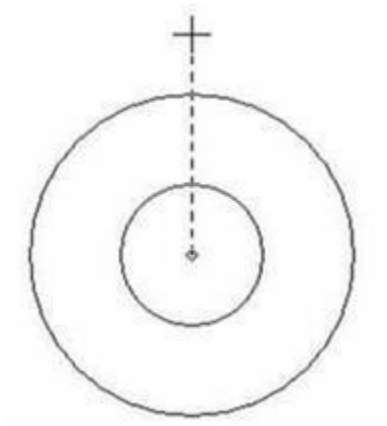
Note: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See [Stairs Styles](#). Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

The next three clicks define the plan dimensions of the stairs. The first sets the center, the second sets the interior radius, and the third sets the exterior radius. These values can also be set in the Inspector Bar.

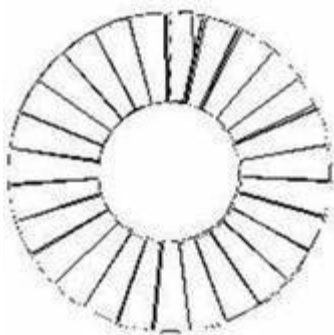
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The space between the interior and exterior radius is the width of the stairs. If this value is different than the **Base Width** set in the **Properties**, then **Base Width** will be overridden. However, you can always change this value later in the **Properties**.

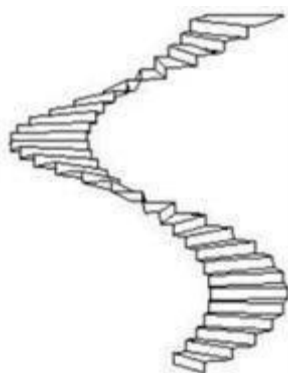
3. Next, set the angle from the circle center, where the first stair will be.



The spiral staircase is created.



This is the clockwise spiral staircase in Isometric view. The height of the staircase is the **Riser Count** times **Riser Height**, both of which can be adjusted in the **Properties**.



Stair Properties

The **Stair** page of the **Properties** contains parameters for defining the size of treads and risers.

Stair shape and **Turn type**: These vary depending on the type of stair you are creating. These options are explained for each stair type.

Base Width: The width of the stairs. You can define this here, or when defining the stair geometry.

Height: The height of the stairs. You can define this here or when selecting stair by edit tool.

Tread depth: The horizontal depth of each step.

Riser height: The vertical height of each step.

Riser count: The number of stairs. You can define this here, or when defining the stair geometry.

Terminate with: Defines what will appear at the top of the stairs.

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Note: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See [Stairs Styles](#). Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

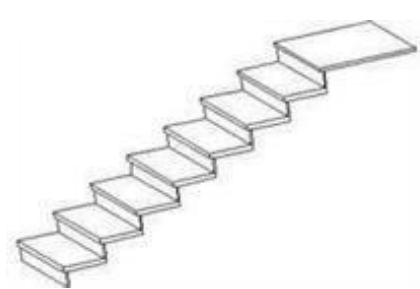
Straight Stair



Default UI Menu: Architecture/Sraight Stair

Ribbon UI Menu:

Inserts a straight line of stairs.



The first two clicks define the stair width, and the third defines the length. After the stairs are created, you can use the Properties to change the width, number of risers, etc.

U-Shaped Stair

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Default UI Menu: Architecture/U-Shaped Stair

Ribbon UI Menu:

Creates a line of stairs that doubles back on itself.

Activate **U-Shaped Stair** and open the **Properties** to the **Stair** page. This example will be **Clockwise** with a **1/2 Landing**. The remaining parameters control the vertical and horizontal dimensions of each stair.

Note: Other stair properties, such as tread and riser thickness, nosing length, and materials can be set using the Style Manager. See [Stairs Styles](#). Once you have styles defined, you can set a stair's style in the **General** page of the **Properties**.

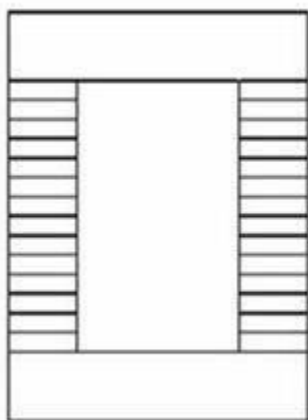
The next three clicks define the plan dimensions of the stairs. The first two clicks sets the stair width and angle. The third click sets the offset - this is the distance between the

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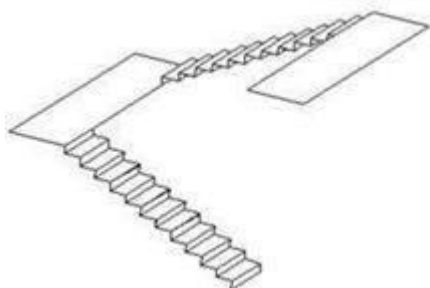
exterior edge of one stair run and the interior edge of the other stair run. These values can also be set in the Inspector Bar.



The space between the first two clicks is the width of the stairs. If this value is different than the **Base Width** set in the **Properties**, then **Base Width** will be overridden. However, you can always change this value later in the **Properties**. The U-shaped staircase is created.



This is the clockwise staircase in Isometric view. The height of the staircase is the **Riser Count** times **Riser Height**, both of which can be adjusted in the **Properties**. Each stair run has half the **Riser Count**, and a landing runs between them.



This would be the result if you selected **1/2 Turn**. Each segment is a staircase, and there are no landings. The total number of risers is divided among the three sets of stairs.



Stair by Linework

Default UI Menu: Architecture/Stair by Linework

Ribbon UI Menu:

This tool create stairs from a set of lines / polylines / arcs / curves.

To create the stairs:

Select the tool

Open the Selection Info palette, and set the Riser height.

Using this pattern as an example:

3. Select 2D polyline/circular arc for the Stair's Linework Left side path (1)

Note: the selection of the right/left side define the direction of the stairs.

4. Select 2D polyline/circular arc for the Stair's Linework right side path (2)

5. Select 2D polyline/circular arc for the Stair's Linework path (3)

6. Select 2D polyline/circular arc for the Stair's Linework Tread's Riser path (4-10)

Note: It is important to select the riser in sequence from first to last in order to get predictable results.

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After you have chosen two risers you will be prompted to **Select 2D polyline/circular arc for the Stair's Linework Tread's Riser path or use "Finish" from local menu to create stair**. Select additional risers or click Finish in the local menu or Inspector bar to complete the stair.

Note: at anytime in steps 3-6 you can opt to use **One Step Back** from the local menus or Inspector bar.

11.9. The Style Manager Palette

Default UI Menu: Tools/Palettes/Style Manager

Ribbon UI Menu:

The Style Manager enables you to define styles for the following commonly-used objects:

Text

Profiles (for modifying windows and doors)

Windows

Doors

Walls

Schedules

Slabs

AEC Dimensions

Dimensions

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Stairs

Rails

Tables

You select assign a style to an object through the object properties on the General page. Simply select the style in the Style drop-down box at the bottom of the page.

You can open the Style Manager by selecting **Tools / Palettes / Style Manager Palette**.



Each category in the Style Manager contains one pre-defined style, called "Standard." For each of these styles, you can define relevant properties.

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To create a new style, highlight a style you want to base the new style on, and click **Create New Style**.



Once you have new style, there are icons at the top you can use to **Rename** or **Delete** it.



The Delay Style Modification* icon can be enabled if you don't want to update your style with each property change.



If **Delay Style Modification** is enabled, you can view both old and new properties in the lower pane. At the top is an icon for updating the entire style at once, or for clearing changes not yet implemented.

Door Styles

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Door Styles

Ribbon UI Menu:

The **Door** tool can be used to insert doors into walls, but you need to first define door styles.

Note: If you want to save styles to a template, see [Saving as Templates](#)

In the Style Manager, there is one style, "Standard," listed under "Door styles." A preview showing a previous of the door style is on the lowest pane.

Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.



2. Assign a name.



This creates a new style which is a copy of "Standard."

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There are three categories of properties. Open **Dimension**, where you can specify measurements of the frame, stop, door panel, and glass.

Auto Adjust to Width of Wall: Sets the depth of the door so that it will cut all the way through the wall.

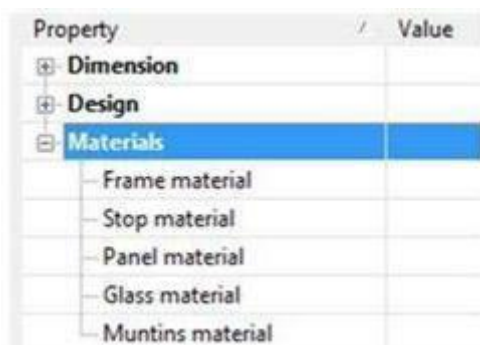
Open **Design**, where you can specify the door type (single, double, pocket, etc.) and shape (rectangular, arch, half round, etc.).

5. You can also select, edit and create muntins.

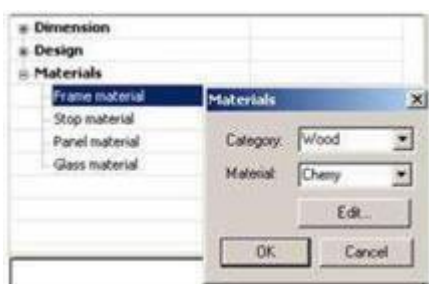
Note: Profile is used when you've created a profile for changing the door shape, or for adding holes. See [Profile Styles](#)

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The last category is **Materials**, in which you can set the materials of the frame, stop, panel, and glass. If you leave any material blank (such as the **Stop Material** below), the object will be colored according to the **Pen** color set in the window's **Properties**.



To set or change a material, just click in the field. In the **Materials** window that opens, select the **Category** and **Material**.



Note: For details on materials, see [Materials](#). Glass material will only be used if the door uses a profile that includes holes.

Before creating any doors, you must have at least one wall defined. See [Inserting Walls](#) To assign height to a wall, define a **Thickness** in the **3D** page of the wall's **Properties**.

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This is the icon for **Door**. Right-click on the icon to set the tool's **Properties**, and click the icon when you're ready to insert a door.



Open the **Door** tool's **Properties**. On the **General** page, select the **Door Style**.

On the **Door Main** page, set the overall dimension of the door. The preview at the lower left updates when you change the values.

Width and Height: Overall outer dimensions of the door.

Inside/Outside of Frame: Specifies that the dimensions of the door are applied by measuring to the inside or outside of the door frame.

Vertical alignment: Specifies whether the door vertical position is specified by the Threshold Height, or the Head of the door.

Head Height: Specifies the vertical location of the top of the door. This field is only accessible if the vertical alignment option is set to **Head of object**.

Threshold Height: Specifies the vertical location of the sill of the door. This field is only accessible if the vertical alignment option is set to **Threshold**.

Rise: The distance from the top corner to the top center, in the case of arched or peaked doors.

Frame Inset: The distance from the front of the wall to the door frame.

Elevation: The distance from the bottom of the wall to the bottom of the door. This value is also available on the InspectorBar.

Open Percent: Defines how wide the door is open.

Leaf: in the case of uneven doors, the size of one of the panes.

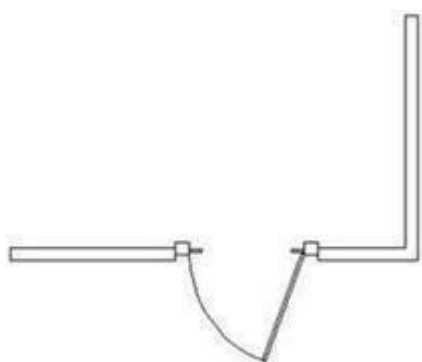
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On the **Pen** page, specify the color of the door. This color will be applied to any part of the door that does not have a material assignment.



When all parameters are set, click the wall to place the door. The wall material within the door area is removed. No matter where you click on the wall, the door is placed according to the **Elevation** value. In this example, the door stop is colored black, which is the default **Pen** color. The frame and panel have the assigned materials.

In **World Plan** view, you can see how the door cuts the wall.



You can orient of the window to the wall by using the **Flip Left-Right**, **Flip Inside-Out** side controls in the Local menu, or Inspector bar. All parameters that available on

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the **Properties** window, including **Door style**, are also available on the Selection Info palette. If you move a door, it will remain within the plane of the wall.

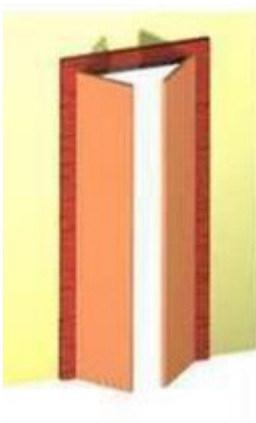
Door Types

These are the standard door types available in the **Design** category of the Style Manager.

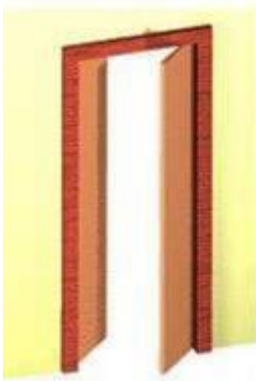
Single



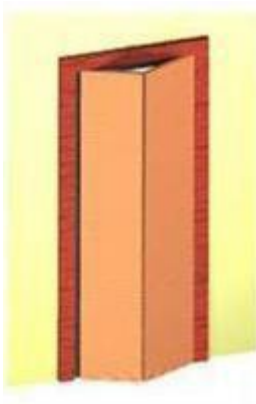
Double



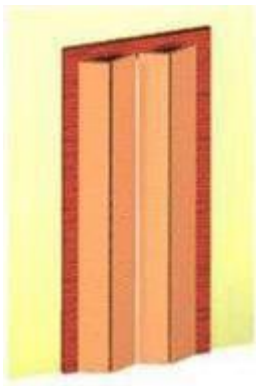
Double Opposing



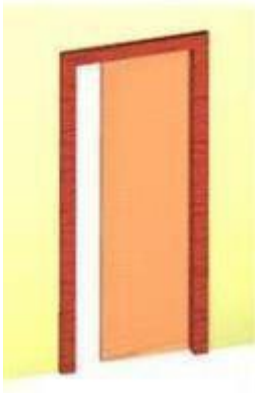
Bifold



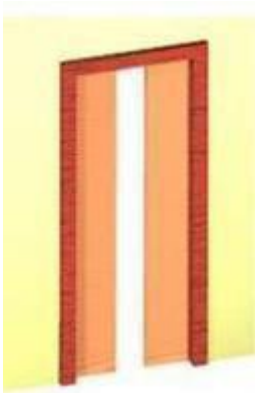
Bifold Double, Panel



Pocket



Double Pocket



Sliding Double



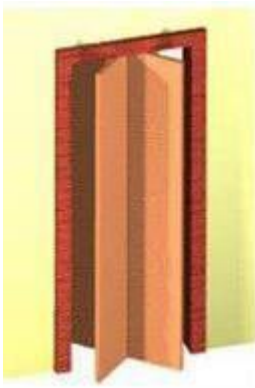
Sliding Triple



Overhead



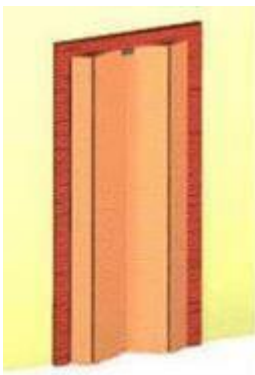
Revolving



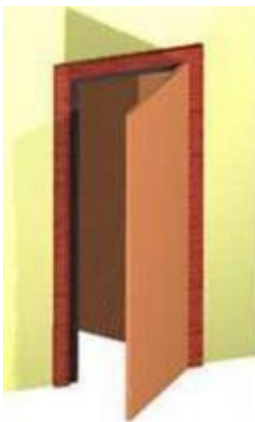
Pass Through (no panel)



Accordion



Communicating



Door Muntins

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Door muntins can be applied only to doors with glass components. To make a door with glass components you must create profile first (Style Manager | Profile Styles). As soon as you have profile style created, navigate to the door styles, select the door style you wish add muntins to and select the profile as door shape (Door Shape = Custom, Profile "Profile Style 1") Door muntins are available in several patterns that you can customize into many different styles:

Rectangular

Diamond

Prairie 9 Lights

Prairie 12 Lights

Starburst

Sunburst

The Starburst and Sunburst patterns are combined rounded/rectangular patterns with a half-round or a quarter-round spoked top and can be used for half- and quarter-round as well as rectangular muntins. The other patterns are primarily rectangular but will fit into a half-round. Multiple muntin blocks can be assigned to each style. This allows you to have different patterns of muntins for each panel of glass, or to combine muntin patterns. To setup muntins component please do the following:

Open the Style Manager Palette.

Navigate to the Door styles.

Select the door style you wish to add muntins to.

Expand the Design properties category and click Muntins.

The Muntins dialog will appear.

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This dialog contains list of all muntins assigned to the door style. For example you can add muntins with Rectangular pattern to the one door glass component and muntins with diamond to other component.

Add: displays the Muntins Block dialog where you can define a name for the muntins block and it's properties.

Edit: displays the Muntins Block dialog with properties for the selected muntins.

Delete: Deletes the selected muntins block.

Disable Muntins: when ON - muntins are disabled.

Open the Style Manager and select a door style you wish to add muntins to.

Expand the Design properties category and click Muntins.

The Muntins dialog will open.

Click the Add button to open Muntins block dialog.

Muntins can be applied to All door glass components, or to individual glass component pane. If you specify a single panel you must specify the number of the panel to be used. The number of the panel corresponds to the order in which you selected the hole profiles.

Note: If you cannot see the door because the preview is rotated, click on the preview with the center mouse button (wheel) hold and drag. This will rotate the preview.

Define the muntins width and depth.

Select the Rectangular in Pattern combo-box.

Enter number of horizontal muntins lights in "Lights High" edit box.

Enter number of vertical muntins lights in "Lights High" edit box.

Click OK to close the Muntins Block dialog.

Click OK to close Muntins dialog.

The Diamond Muntins pattern uses the same options as the Rectangular pattern. Prairie-9 Lights pattern, and the Prairie-12 Lights pattern use two alternate options: **Edge X Offset and Edge Y Offset:** these specify the distance of the outer muntins from the panel edge.

The Starburst Muntins pattern has two additional options:

Spokes: which sets the number of spokes that radiate out from the center to form the burst pattern.

Center: which sets whether the hub will be at the Top or Bottom of the radial pattern.

The Sunburst Muntins pattern has two additional options.

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Style: this sets the whether the center of the sunburst will be an open circle or a closed disk.

Radius: this sets the radius for the center of the sunburst.

Window Styles

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Window Styles

Ribbon UI Menu:

The **Window** tool can be used to insert windows into walls, but you need to first define window styles.

Note: If you want to save styles to a template, see [Saving as Templates](#)

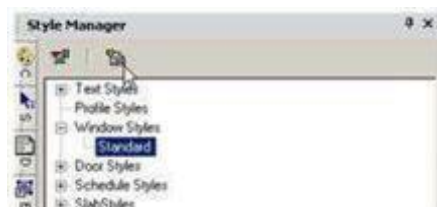
In the Style Manager, there is one style, "Standard," listed under "Windows." A preview showing a previous of the window style is on the lowest pane.

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Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.



2. Assign a name.



This creates a new style which is a copy of "Standard."

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There are three categories of properties. Open **Dimension**, where you can specify measurements of the frame, sash, and glass.

Auto Adjust to Width of Wall: Sets the depth of the window so that it will cut all the way through the wall.

Open **Design**, where you can specify the window type (double hung, transom, etc.) and shape (rectangular, arch, octagon, etc.).

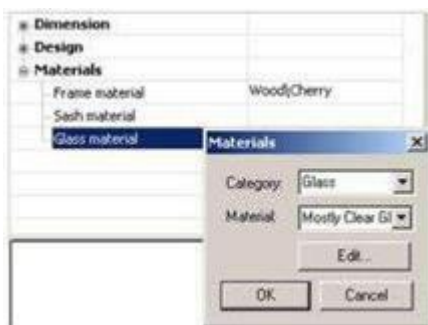
5. You can also select, edit and create muntins.

Note: Profile is used when you've created a profile for changing the window shape, or for adding holes. See [Profile Styles](#)

The last category is **Materials**, in which you can set the materials of the frame, sash, and glass. If you leave any material blank (such as the **Sash Material** below), the object will be colored according to the **Pen** color set in the window's **Properties**.

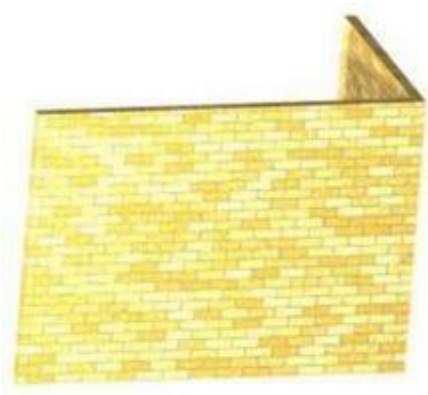
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To set or change a material, just click in the field. In the **Materials** window that opens, select the **Category** and **Material**.



Note: For details on materials, see [Materials](#)

Before creating any windows, you must have at least one wall defined. See [Inserting Walls](#). To assign height to a wall, define a **Thickness** in the **3D** page of the wall's **Properties**.



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This is the icon for **Window**. Right-click on the icon to set the tool's **Properties**, and click the icon when you're ready to insert a window.

Open the **Window** tool's **Properties**. On the **General** page, select the **Window Style**.

On the **Window Main** page, set the overall dimension of the window. The preview at the lower left updates when you change the values.

Width and Height: Overall outer dimensions of the window.

Inside/Outside of Frame: Specifies that the dimensions of the window are applied by measuring to the inside or outside of the window frame.

Vertical alignment: Specifies whether the window's vertical position is specified by the Sill Height, or the Head of the window.

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Head Height: Specifies the vertical location of the top of the window. This field is only accessible if the vertical alignment option is set to **Head of object**.

Sill Height: Specifies the vertical location of the sill of the window. This field is only accessible if the vertical alignment option is set to **Sill of object**.

Rise: The distance from the top corner to the top center, in the case of arched or peaked windows.

Frame Inset: The distance from the front of the wall to the window frame

Open Percent: Defines how wide the window is open.

Leaf: in the case of uneven windows, the size of one of the panes.

On the **Pen** page, specify the color of the window. This color will be applied to any part of the window that does not have a material assignment.



When all parameters are set, activate **Window** and click the wall to place the windows. The wall material within the window area is removed. No matter where you click on the wall, the windows are placed according to the **Elevation** value. In this example, the sash is colored black, which is the default **Pen** color. The frame and glass have the assigned materials.

13. If you click another wall, the window properly aligns to it.



You can orient of the window to the wall by using the **Flip Left-Right**, **Flip Inside-Outside** controls in the Local menu, or Inspector bar.

In **World Plan** view, you can see how the windows cut the walls.



All parameters available on the **Properties** window, including **Window style**, are also available on the Selection Info palette. If you move a window, it will remain within the plane of the wall.

Window Types

These are the standard window types available in the **Design** category of the Style Manager.

Single Hung



Double Hung



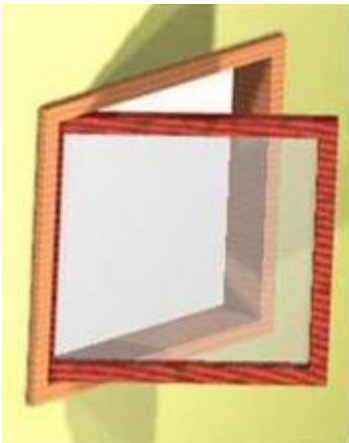
Pass Through (no glass)

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Picture (does not open)



Casement



Double Casement



Glider



Awning Transom



Hopper Transom



Single Hopper

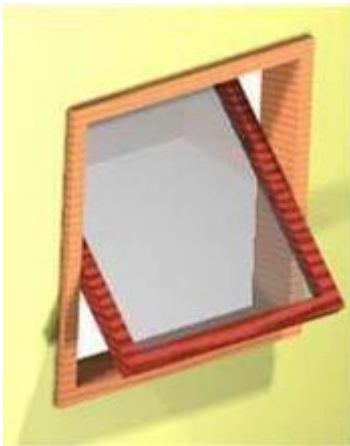


Single Awning

Vertical Pivot



Horizontal Pivot



Uneven Double Hung: the two panes have different heights.

Window Muntins

Window muntins are available in several patterns that you can customize into many different styles:

Rectangular

Diamond

Prairie 9 Lights

Prairie 12 Lights

Starburst

Sunburst

The Starburst and Sunburst patterns are combined rounded/rectangular patterns with a half-round or a quarter-round spoked top and can be used for half- and quarter-round as well as rectangular muntins. The other patterns are primarily rectangular but will fit into a half-round. Multiple muntin blocks can be assigned to each style. This allows you to

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have different patterns of muntins for each panel of glass, or to combine muntin patterns. To setup muntins component please do the following:

Open the Style Manager Palette.

Navigate to the Window styles.

Select the window style you wish to add muntins to.

Expand the Design properties category and click Muntins.

The Muntins dialog will appear.

This dialog contains list of all muntins assigned to the window style. For example you can add muntins with Rectangular pattern to the one window glass component and muntins with diamond to other component.

Add: displays the Muntins Block dialog where you can define a name for the muntins block and it's properties.

Edit: displays the Muntins Block dialog with properties for the selected muntins.

Delete: Deletes the selected muntins block.

Disable Muntins: when ON - muntins are disabled.

Open the Style Manager and select a window style you wish to add muntins to.

Expand the Design properties category and click Muntins.

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The Muntins dialog will open.

Click the Add button to open Muntins block dialog.

Muntins can be applied to All window glass components, or to individual glass component pane. If you specify a single panel you must specify the number of the panel to be used. The number of the panel corresponds to the order in which you selected the hole profiles.

Note: If you cannot see the window because the preview is rotated, click on the preview with the center mouse button (wheel) hold and drag. This will rotate the preview.

Define the muntins width and depth.

Select the Rectangular in Pattern combo-box.

Enter number of horizontal muntins lights in "Lights High" edit box.

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Enter number of vertical muntins lights in "Lights High" edit box.

Click OK to close the Muntins Block dialog.

Click OK to close Muntins dialog.

The Diamond Muntins pattern uses the same options as the Rectangular pattern. Prairie-9 Lights pattern, and the Prairie-12 Lights pattern use two alternate options: **Edge X Offset and Edge Y Offset:** these specify the distance of the outer muntins from the panel edge.

The Starburst Muntins pattern has two additional options:

Spokes: which sets the number of spokes that radiate out from the center to form the burst pattern.

Center: which sets whether the hub will be at the Top or Bottom of the radial pattern.

The Sunburst Muntins pattern has two additional options.

Style: this sets the whether the center of the sunburst will be an open circle or a closed disk.

Radius: this sets the radius for the center of the sunburst.

Wall Styles

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Wall Styles

Ribbon UI Menu:

The Wall tool is used to insert Walls. Every wall is based upon a defined Wall style created and managed in the Style Manager.

In the Style Manager, there is one style, "Standard," listed under "Wall Styles." A preview showing a previous of the Wall style is shown on the lowest pane of the palette.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.

Assign a name. This creates a new style which is a copy of "Standard."

Component Walls

Default UI Menu: Tools/Palettes/Style Manager/Wall Styles

Ribbon UI Menu:

Walls are composed of components with each component representing a part of the walls geometry. By default every Wall Style has at least one component of the "Standard" type. Components are essentially long boxes defined by the components properties.

To add components to a wall style

Select that wall style in the Style Manager.

Increase value in the **Number of Components** field to the desired value.

To delete components from a wall style

Select that wall style in the Style Manager.

Decrease the value in the Number of Components field to the desired value.

Warning: Components are added sequentially to the Wall style. If you increase the number of components the new components will be added to the bottom of the list of components. If you decrease the number of components the components at the end of the list will be deleted. The best practice is to add, build and complete each component in turn before adding more.

Component Wall Properties

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There are nine categories of wall properties. Several of them have sub-properties:

Name: This property is used to specify the name of the component

Pen Color: This property is used to specify the Pen Color of the component.

Width: These properties are used to specify the width of the component.

Edge Offset: These properties are used to specify the horizontal placement of the component.

Bottom Elevation: These properties are used to specify the bottom of the component.

Top Elevation: These properties are used to specify the top of the component.

Dimension: These properties are used to specify the width of the component.

Brush: These properties are used to specify the brush style of the component.

Component material: This property is used to specify the material of the component.

Component draw priority: This property is used to specify how the component will interact and intersect with other components and walls.

Direction

Walls have direction with a right and left side. Whether a side is Left or Right depends on the wall's direction. To determine side is by assuming you are standing at the start point wall and facing the end point of the wall. Another way to visualize In other words, if a wall is drawn from left to right, the side toward the top of the screen is the left side of the wall, and the side toward the bottom of the screen is the right side of the wall. You can show a walls direction by checking the **Show Direction** on the **Wall** page of the wall's properties. Horizontal values use for defining components may have a negative or positive value. The left side of the wall is the positive direction, and the right side of the wall is the negative direction.

Width

The width of a wall is measured from the Edge Offset of the wall. There are four Width properties that combine to set the actual width of a component:

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Width: This value sets base width value for the component.

Plus Wall Width: If this value is checked, the wall's modified Width value will be added to the Component's Width value to specify the total width of the component.

Operator: Specifies how the Value will modify the wall's Width (added, subtracted, multiplied, or divided) before it is added to the components Width.

Value: This number is used to modify the wall's Width before it is added to the component Width.

Note: A wall's width is specified in the wall's properties

These for properties combine in the following pattern:

Actual Component Width = Width + (Wall Width <operator> Value) - depending upon the operator selected. In other words:

Actual Component Width = Width + (Wall Width + Value) - if adding

Actual Component Width = Width + (Wall Width - Value) - if subtracting

Actual Component Width = Width + (Wall Width * Value) - if multiplying

Actual Component Width = Width + (Wall Width / Value) - if dividing

So, if the Width of the component is 2 feet, and the Width of the wall is 1 foot, if the operator is Divide, and the Value is set to 4, the components actual width will be 2.25 feet.) Or $2 + (1/4)$.

Edge Offset

There are four Width properties that combine to set the Edge Offset of a component: Offsets are measured from the baseline of the wall. The baseline of a wall is indicated in the Style Manager by the two blue nodes. The direction arrow of always resides on the baseline.

Edge Offset: this value sets base offset value for the component.

Plus Wall Width: if this value is checked, the wall's modified Width value will be added to the Component's Edge Offset value to specify the total offset of the component.

Operator: specifies how the Value will modify the wall's Width (added, subtracted, multiplied, or divided) before it is added to the components Edge Offset.

Value: this number is used to modify the wall's Width before it is added to the component Edge Offset.

Note: A wall's width is specified in the wall's properties

These properties combine in the following pattern:

Actual Offset = Edge Offset + (Wall Width <operator> Value) - depending upon the operator selected. In other words:

Actual Offset = Edge Offset + (Wall Width + Value) - if adding

Actual Offset = Edge Offset + (Wall Width - Value) - if subtracting

Actual Offset = Edge Offset + (Wall Width * Value) - if multiplying

Actual Offset = Edge Offset + (Wall Width / Value) - if dividing

So, if the Edge Offset of the component is 4 inches, and the Width of the wall is 6 inches, if the operator is Add, and the Value is set to 4, the components actual offset will be 14 inches.) Or $4+(6+4)$.

Bottom Elevation

The Bottom Elevation is specified by two properties:

Offset: this value specifies the distance between the From Elevation and the bottom of the component.

From Elevation: this setting provides four positions along the wall height, from which the Offset can be measured.

Wall Bottom – this is the absolute lowest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Baseline, but it may vary if a Wall Modifier has been applied to the wall.

Wall Top – this is the absolute highest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Base Height, but it may vary if a Wall Modifier has been applied to the wall.

Base Height – this is the Height value specified in the Wall's Height property.

Baseline – this is the walls Z location, its position on the Z axis

Top Elevation

The Bottom Elevation is specified by two properties:

Offset: this value specifies the distance between the From Elevation and the top of the component.

From Elevation: this setting provides four positions along the wall height, from which the Offset can be measured.

Wall Bottom – this is the absolute lowest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Baseline, but it may vary if a Wall Modifier has been applied to the wall.

Wall Top – this is the absolute highest point of the wall as measured from the position of the wall on the Z axis (Z location). It is normally the same as the Base Height, but it may vary if a Wall Modifier has been applied to the wall.

Base Height – this is the Height value specified in the Wall's Height property.

Baseline – this is the walls Z location, its position on the Z axis

Dimension

The options are:

From Left Side

From Center

From Right Side

Component Draw Priority

The Component Draw Priority is used to specify how components will draw, heal and clean up with other components and other walls. The following conditions apply with the Component Draw Priority:

Components with the same Draw Priority will heal to each other.

Components with a lower value will pass through components with a higher value.

Draw Priority takes precedence over Draw order

Table Styles

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Table Styles

Ribbon UI Menu:

Tables can be used to define table properties. Table can be created using **Insert / Table** (see [Tables](#)) or they are created automatically when you create a report (see [Custom Properties, Database, and Reports](#)).

Note: If you want to save styles to a template, see [Saving as Templates](#)

In the Style Manager, there is one style, "Standard," listed under "Tables."

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You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.

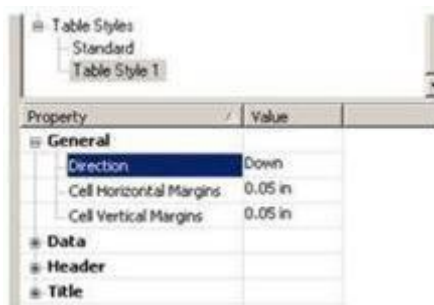


2. Assign a name or accept the default.



This creates a new style which is a copy of "Standard."

Highlight the new style, and open the **General** category. This is where you can define margins and direction. If **Direction** is down, the header and title appear at the top of the table.

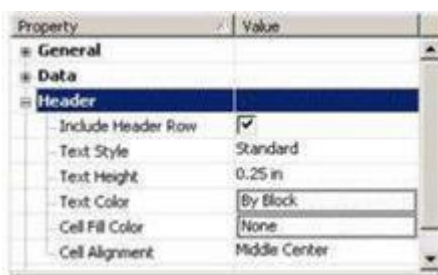


The **Data** category contains properties of the text and colors in the main cells of the table (not the headers or title). For details on defining text, see [Text Styles](#).

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The **Header** and **Title** categories are similar to **Data**. They have the additional option whether to display header or title rows.



Text Styles

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Text Styles

Ribbon UI Menu:

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The can be used to set various styles for text. For details on creating text, see [Text](#). In the Style Manager, there is one style, "Standard," listed under "Text styles." This style defines properties such as font and height. A preview showing a layout the style is on the lowest pane.

Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.



Assign a name or accept the default.



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This creates a new style which is a copy of "Standard."

3. Highlight the new style you created, and change some of the parameters.



4. To apply this style, open **Properties** of a text string to the **General** page.

Note: If a text string already has other parameters defined, such as color or a text box, these properties will remain. Only the changed style parameters, such as font or height, will update.

Standard Dimension Styles

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(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Dimension Styles

Ribbon UI Menu:

For styles and Style Manager usage for standard dimensions see [Dimension Styles](#)

AEC Dimension Styles

(Available in Platinum)

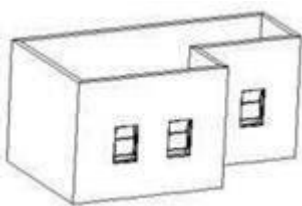
Default UI Menu: Tools/Palettes/Style Manager/AEC Dimension Styles

Ribbon UI Menu:

The Style Manager can be used to set various styles for wall dimensions. These dimensions are created with the **Wall Dimension** tool.

Note: If you want to save styles to a template, see [Saving as Templates](#)

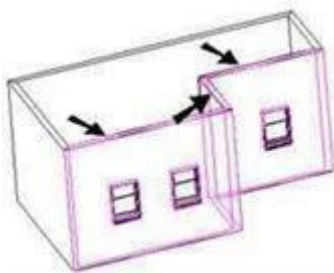
1. Start with some walls, and add openings like windows or doors.



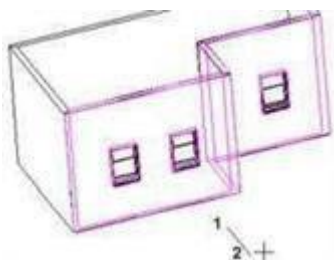
In the Style Manager, there is one style, "Standard," listed under "AEC Dimension styles." This style has one chain, and a preview showing a layout of AEC Dimension styles is below.

Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

Activate **Wall Dimension**, and select one chain of walls, using the Shift key for multiple selection. Select **Finish Selection**.



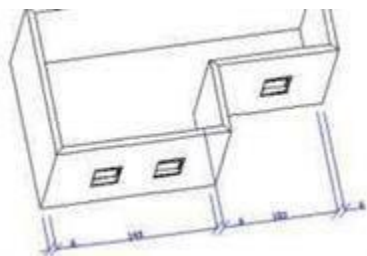
The next click defines the location of the dimension baseline. The subsequent click defines the dimension angle - if you want the dimensions to proceed along the wall chain, the angle should be perpendicular to the walls.



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Tip: To ensure a vertical or horizontal angle line, you can use the X or Y lock in the Coordinate Field.

This is the resulting dimension chain in the current style.



Note: If you want to change the properties of the dimensions themselves, such as font or leader lines, see [Dimension Properties](#)

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.



5. Assign a name or accept the default.



This creates a new style which is a copy of "Standard."

To change the current wall dimension to the new style, open its **Properties** to the **General** page.

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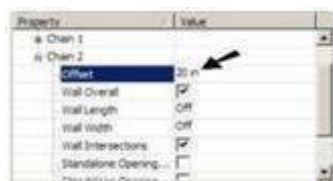


7. With the new style highlighted, change the **Number of chains** to 2.



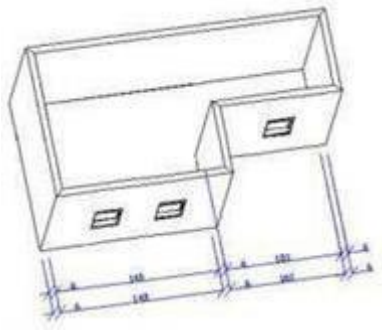
Note: If **Delay Style Modification** is enabled, you can see the old and new values for each field. Then you can update the style by clicking **Apply Style Changes**. If there is no delay, then all changes are implemented immediately.

8. To separate Chain 2 from Chain 1, add an **Offset**.

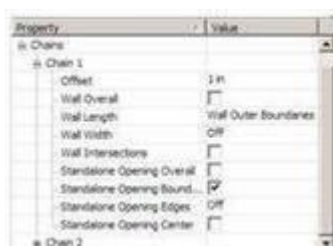


Chain 1 is offset from the baseline, which is where you first clicked to define the dimension line's location. Chain 2's offset is its distance from Chain 1.

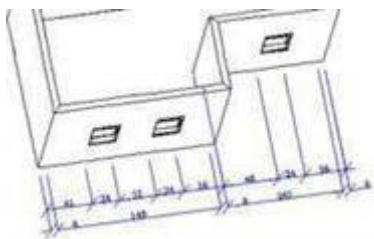
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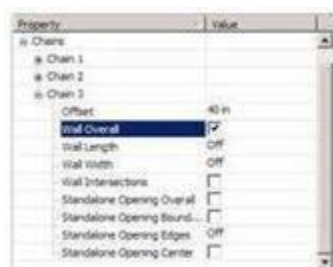
9. Open the properties for Chain 1 and make these changes.



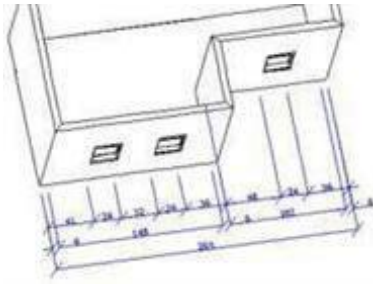
Now Chain 1 dimensions the openings. Chain 2 is still the same as the "Standard" style.



To add an overall dimension, add Chain 3 and offset it from Chain 2. Only **Wall Overall** should be checked.



This is the result.



Stairs Styles

(Available in Platinum)

Default UI Menu: Tools/Palettes/Style Manager/Stair Styles

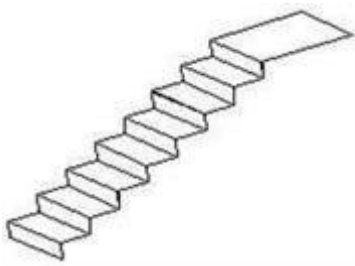
Ribbon UI Menu:

The Style Manager can be used to set various styles for stairs. For details on the types of stairs you can create, see [Stairs](#)

Note: If you want to save styles to a template, see [Saving as Templates](#)

1. Start with a staircase. This example uses a **Simple Stair**.

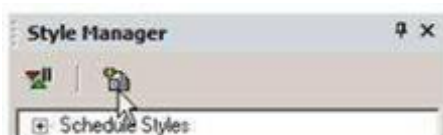
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In the Style Manager, there is one style, "Standard," listed under "Stair styles." This style has two categories: components and materials. The graphic at the bottom shows a preview of the style.

Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.

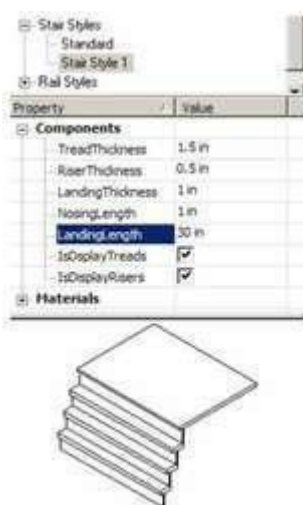


Assign a name or accept the default.

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The new style is a copy of the "Standard" style. To make some changes, open **Components** and change the parameters, such as tread and riser thickness, etc. Nosing Length* is how far the tread extends past the riser below. To view your changes, you can click inside the preview window to zoom in. (If you right-click in the preview window, a menu appears that enables you to change the view as well as render style.)



Note: Other stair properties, such as stair width, tread depth, and riser height, are set on the **Stair** page of the stair's **Properties**.

Open the **Materials** category, in which you can assign different materials to the treads, risers, and landing. Set the preview to **Quality Rendering** to see the materials.

Note: For details on materials, see [Materials](#)

To change the staircase in the model so that it has the new style, open its **Properties** to the **General** page. All styles defined for stairs will be available under **Stair styles**.



The staircase now has the properties of the new style.



Rail Styles

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(Available in Platinum)

Default UI Menu: Tools/Palettes/Style Manager/Rail Styles

Ribbon UI Menu:

The Railing tool can be used to create a standalone, linear railing, or a railing along a staircase.

Note: If you want to save styles to a template, see [Saving as Templates](#)

Menu: Architecture / Stairs and Railings / Railing



In the Style Manager, there is one style, "Standard," listed under "Rail styles." A preview showing a layout of this rail style is below.

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Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.



Assign a name or accept the default.



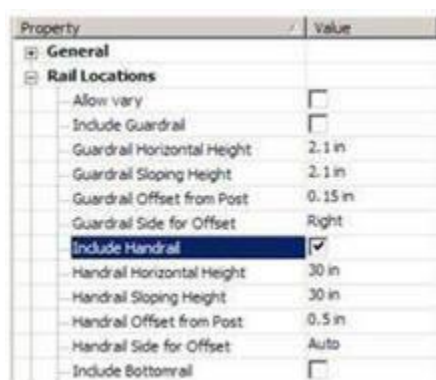
This creates a new style which is a copy of "Standard."

To create the railing with the new style, open the tool's **Properties** to the **General** page.

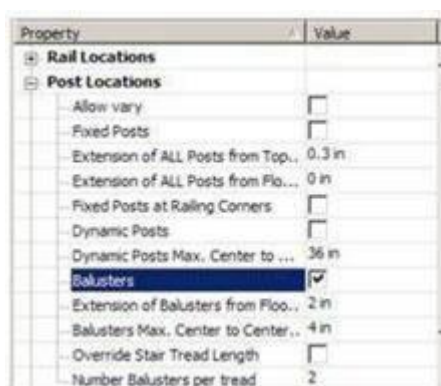
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Highlight the new rail style, and open the **Rail Locations** category. This category sets which rails are displayed, their heights along horizontal and sloped segments, and their offsets from the vertical posts. In this example, only a Handrail* is included.



Open Balusters* are placed between dynamic posts. In this example, only balusters are included.



Note: You can either specify the number of balusters per tread, or baluster spacing. If the number per tread is entered, this will by default override the spacing. If you check **Override**, the spacing value will be used.

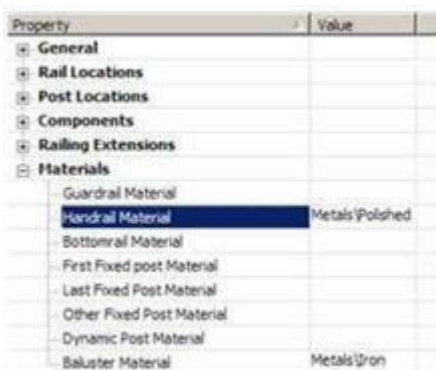
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Open **Components**, in which you specify the shapes and dimensions of rails and posts.



Open **Railing Extensions**, in which you can specify horizontal extensions of rails, relative to posts and landings.

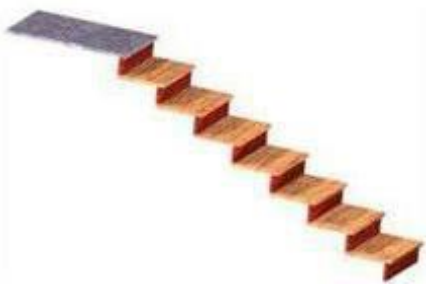
Finally, open **Materials**, in which you can specify the material for each post and rail type.



Note: For details on materials, see [Materials](#)

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This example is for a stair railing, so start by creating a staircase. (For details, see [Stairs](#).)



Activate **Railing**. By default, **Stair Railing** is active



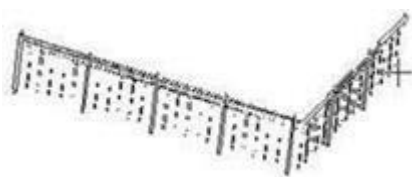
To place rails on both sides of the stair use the Attach to Both Sides option.



Select the staircase, and the railing is created. The other type of railing is standalone, defined by linear segments. To create this, make sure **Stair Railing** is not selected.



This railing is defined by one or more segments, similar to a polyline or wall. A preview of the railing appears while defining the segments. When finished, select **Finish** from the local menu or Inspector Bar.



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This example includes a handrail, guardrail, and bottom rail. The posts include a fixed post at the end, dynamic posts along the length, and balusters between bottom rail and guard rail.



Schedule Styles

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Schedule Styles

Ribbon UI Menu:

The **Schedule** tool can be used to create a table in your file detailing all selected walls, windows, doors, and/or slabs.

Note: The **Fill Schedule Wizard** is another way to add a schedule, that scans for objects on specific layers.

Note: If you want to save styles to a template, see [Saving as Templates](#)

This example starts with a model that has walls and slabs. There are two types of windows and three types of doors.

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Note: For details on these components, see: [Window Door Slabs](#)

In the Style Manager, there is one style, "Standard," listed under "Schedule styles."

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted, then click **Create New Style**.



3. Assign a name or accept the default.

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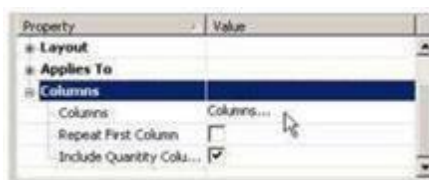
This creates a new style which is a copy of "Standard."

Highlight the new style, and open the **General** category. This is where you can define the schedule title.

Default Format and **Layout** contain various options for the text and cells in the table. Next to **Applies To**, click the **Value** field to specify which components you want to include in the schedule.

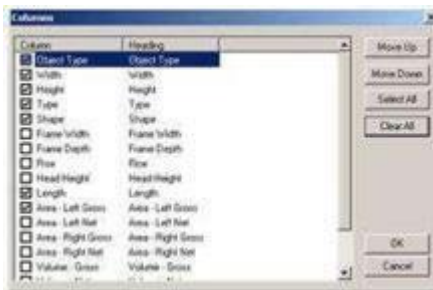


6. To specify the columns, click the **Value** field next to **Columns**.

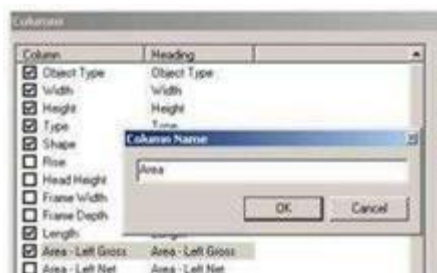


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This opens a menu from which you can select the schedule's columns. To move a column up or down, highlight it and click **Move Up** or **Move Down**. In this example, **Object Type** was moved to the top of the list.



If you want to change the name of a column, click its name in the **Heading** field. In this example, "Area - Left Gross" was changed to "Area."

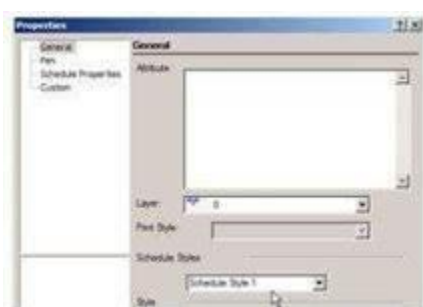


Click OK to end the column definitions.

This is the icon for **Schedule**. Right-click on the tool to set the tool's **Properties**

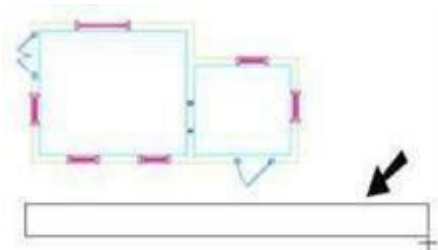


Open the **Schedule** tool's **Properties** to the **General** page.



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Switch to **World Plan** view. Activate **Schedule**, and click two corners to define the schedule's width.



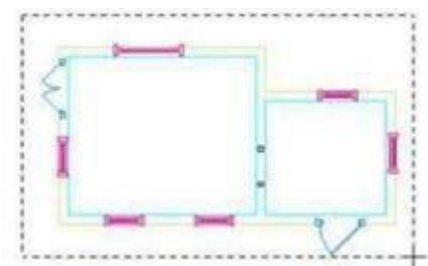
When first created, the schedule contains only the title and column headers.

Remish House Schedule						
Qty	Object Type	Width	Height	Type	Measure	Area

To build the schedule, select **Add Object to Schedule**.



First click the schedule you want to add to, then select the windows and doors to include. Use Shift to select multiple objects, or use a selection window.



When finished, select **Finish** from the local menu or Inspector Bar. The windows and doors appear in the schedule.

Remish House Schedule						
Qty	Object Type	Width	Height	Type	Measure	Area
1	Wall	8.00	1100.00		0	8200.00 796.17
1	Wall	8.00	1100.00		0	8200.00 866.67
1	Wall	8.00	1100.00		0	8200.00 796.25
1	Wall	8.00	1100.00		0	8200.00 51.48
1	Wall	8.00	1100.00		0	777.00 477.92
1	Door	16.00	72.00	Double Door	Unit	2
1	Door	16.00	72.00	Double Door	Rectangular	2
1	Window	24.00	40.00	Acoustic Window	Rectangular	1
1	Door	24.00	72.00	Double Door	Rectangular	1
1	Door	24.00	72.00	Double Door	Unit	2
1	Window	24.00	40.00	Acoustic Window	Unit	2

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If you want to remove objects, use **Remove Object from Schedule**. All schedule objects are highlighted; select the windows and doors you want to remove. Select Finish, and the objects are removed from the schedule.

Fill Schedule Wizard



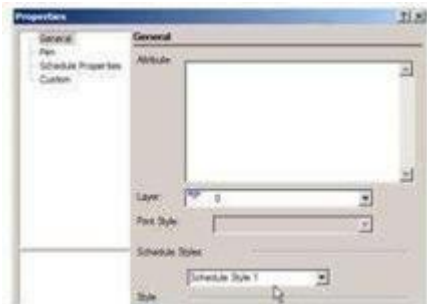
The **Fill Schedule Wizard** enables you to scan your drawing for objects to include in the schedule; you do not have to select objects manually. The wizard scans for specific types of objects on specific layers, which is useful for large drawings which may have objects on invisible layers, or in which slabs are not easy to select because of multiple floors.

This examples uses a model that has walls and slabs. There are two types of windows and three types of doors.

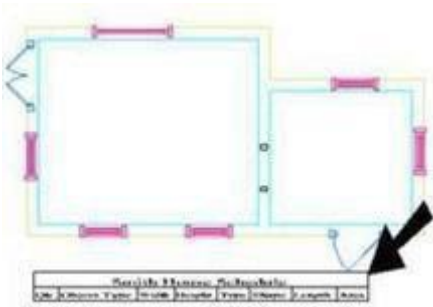
2. Use the Style Manager to create a schedule style.

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To create the schedule with the new style, open the **Schedule** tool's **Properties** to the **General** page.



Switch to **World Plan** view and use the **Schedule** tool to place the schedule, by clicking two corner points.



Select Fill Schedule Wizard.



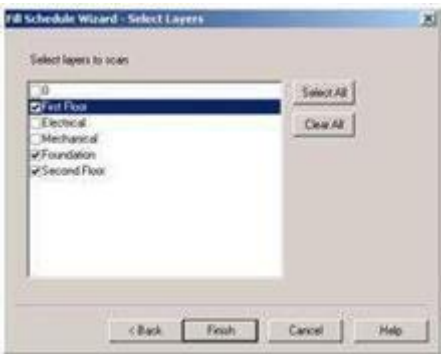
Click the schedule you want to fill.

In the wizard, check the types of objects to include. (The list that appears in the window depends on what was defined as part of the schedule style.) Then click **Next**.

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8. Select the layers you want to include in the scan, and click **Next** .



Note: For details on layers, see [Snap Modes](#)

The schedule is filled with all objects found on the selected layers.

Sample House Schedule						
Object Type	Width	Height	Type	Shape	Length	Area
Wall	6.00	100.00		T	120.00	11400.37
Wall	6.00	100.00		T	102.00	9600.00
Wall	6.00	100.00		T	120.12	11412.18
Wall	6.00	100.00		T	102.00	9600.00
Wall	6.00	100.00		T	80.00	7412.50
Wall	6.00	100.00		T	75.00	6900.00
Wall	6.00	100.00		T	80.12	7412.80
Door	16.00	72.00	Double-Opening	Half-Round		
Door	10.00	84.00	Double-Opening	Rectangular		
Window	24.00	40.00	Awning Transom	Rectangular		
Door	24.00	70.00	Double-Opening	Rectangular		
Slab	?	?				
Window	42.00	62.00	Awning Transom	Oval		

Slab Styles

(Available in Platinum and Professional)

Default UI Menu: Tools/Palettes/Style Manager/Slab Styles

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Ribbon UI Menu:

The Style Manager can be used to set various styles for slabs. For details on creating slabs, see [Slabs](#)

Note: If you want to save styles to a template, see [Saving as Templates](#)

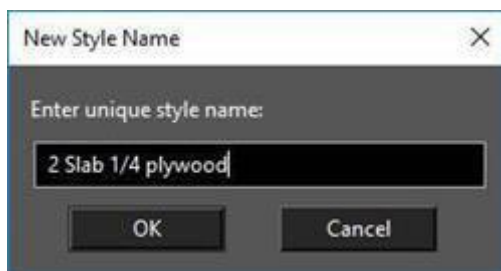
In the Style Manager, there is one style, "Standard," listed under "Slab styles." A preview showing a previous of the slab style is on the lowest pane.

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Note: In the Preview area, you can click to zoom part of the graphic. Double-click to fit the graphic in the window. You can also preview dynamically by using mouse wheel.

You can change the "Standard" style, but if you want to preserve this style, make sure "Standard" is highlighted and click **Create New Style**.

2. Assign a name.



This creates a new style which is a copy of "Standard." By default, a slab consists of one component, for which you can specify a name, thickness, offset, and material.

In this example, there will be two components. Change the number and press Enter, and an additional component is added.

Component 1 is a 2" concrete slab with zero offset, which means the slab will be created flush with the bottom of the walls. Component 2 is 1/4" wood with an offset of 2", so that it will sit directly atop the concrete slab. To define material, click inside the field, then select a category and material.

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Note: For details on materials, see [Materials](#)

These are the icons for **Add Slab by Click** and **Convert to Slab**. Right-click on either icon to set the tool's **Properties**, and click the relevant icon to create the slab.

1. To create the slab with the new style, open the tool's **Properties** to the **General** page.

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2. Create the slab using one of the slab tools.

3. In this example, if you remove the walls, both components of the slab can be seen.



Rendering Slab Preview:

You can also render the slab preview in style manager. Right click the preview pane and select the desired rendering from local menu. You can also change view by right clicking in the preview pane and selecting the desired view from the local menu.

You can now view the actual Slab setup including all the components and materials.

Demo Video:

<https://youtu.be/e7YphZglURk>

Custom Blocks for Windows and Doors

TurboCAD 2023 User Guide Publication

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Tools/Palettes/Style Manager/Door Styles, Tools/Palettes/Style Manager/Window Styles

Ribbon UI Menu:

This functionality provides the means to attach one or several blocks to window and doors to provide additional geometry e.g. a window sill, door knob, shutters. You can attach custom block to entire window/door or to different parts.

To make a door with components you must create the block first. The block used for this must be composed only of ACIS solids.

Multiple muntin blocks can be assigned to each style. This allows you to have different patterns of muntins for each panel of glass, or to combine muntin patterns. To setup muntins component please do the following:

Open the Style Manager Palette.

Navigate to the Door or Window styles.

Select the style you wish to add blocks to.

Expand the Design properties category and click Blocks.

The Custom Blocks dialog will appear.

This dialog contains list of all blocks assigned to the door or window style.

Custom Block Dialog

Disable Blocks: When ON - custom blocks are ignored.

Add: displays Custom Block dialog to add new custom block.

Edit: displays Custom Block dialog to edit selected custom block

Remove: deletes selected custom block

Blocks list: contains list of blocks attached to the window style.

Custom Blocks Editor

Block combo-box: Used for selecting the block (from the list of blocks) that is attached to the window/door style.

Scale To Fix Controls Group

Width check box: when ON - block is scaled to fit window/door width.

Height check box: when ON - block is scaled to fit window/door height.

Depth check box: when ON - block is scaled to fit window/door depth.

: The Plus (+) fields allow you to add additional width, depth or height to a block in surplus to the scaled fitting.

Lock XY Ratio check box: works in combination with one of the (Width, Height, and Depth) check boxes.

For example, when Width is ON and you click Lock XY Ratio the block is scaled in a way to keep XY ratio.

Mirror In Controls

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Mirror X check box: mirrors block relative X axis.

Mirror Y check box: mirrors block relative Y axis.

Mirror Z check box: mirrors block relative Z axis.

Insertion Point controls group

combo-box: Defines how the block's origin is attached relative to window/door in X Axis (to Left Side, Center or Right Side)

combo-box: Defines how the block's origin is attached relative to window/door in Y Axis (to Back Side, Center or Front Side)

combo-box: Defines how the block's origin is attached relative to window/door in Z Axis (to Bottom Side, Center or Top Side)

Y: Z: edit boxes: define the offset from block origin point

Component Controls Group

Frame Component:

Out side - block is attached to entire window

Inside - block is attached to inside frame

Window Component

All - block is attached to all window component **Single**

- block is attached to single window component

Materials Controls Group

These controls allow define material that will be used for this custom block in render.

Example of use

Suppose you would like to create custom sill for our window style.

Create a block.

First, we create a geometry for our sill (just a box with 10x2x0.2 in size

Note: Block used for Custom Blocks must be composed of ACIS solids. For best results align your block with the origin of the drawing.

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Make Block

Select the box and select **Format | Create Block**.

Enter "Custom Sill" as the Block name and click OK.

Navigate to the style manager palette, select Standard window style and click the Block property in the properties of the window style.

The Custom Block dialog opens. It contains a list of all the blocks that are attached to the window style.

5. Click the **Add** button to add new block to the window style.

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6. The Custom Block Editor dialog opens.

7. Click the Block combo-box and select the desired block.

Right click on the preview pane to display pop-up menu and select Isometric_SE item from it (to have better point of view)

Now let's adjust the block size and position.

Check Width check box (to scale the block width to the width of window).

Select Back from Y: combo-box (to place the block to back side of window)

Click OK.

Click OK again.

Now create a wall in drawing and insert a window (of Standard style) into it.

Profile Styles

(Available in Platinum)

Default UI Menu: Tools/Palettes/Style Manager/Profile Styles

Ribbon UI Menu:

Profiles are used to define custom shapes for doors, and windows. Once you have created the profile, the profile is selected as part of the window or door's profile.

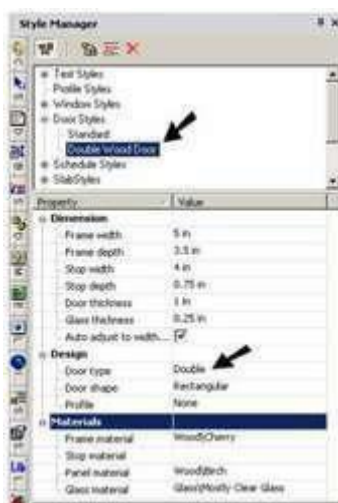
Note: If you want to save styles to a template, see [Saving as Templates](#)

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The first example will show how to use a profile for a door shape.

Example 1- Door Profile with One Boundary

Before creating the door, create a new door style. This one is a double door with a rectangular shape.



Note: For details on doors, see [Door Styles](#)

Create walls and insert the door. Use the door's **Properties** window to set the overall size and elevation of the door.

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This is the door that will be modified with a profile.

First, look at the **Delay Style Modification** icon; if it is enabled, you cannot create a new profile. Make sure **Delay Style Modification** is off.



Highlight Profile styles and click Create New Style.



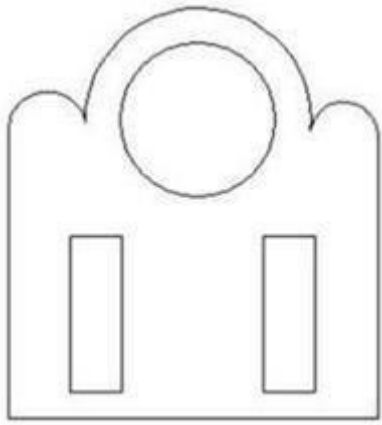
Assign a name to the profile.



When you create a profile, you switch to **Edit Geometry** mode, and anything you have in your file is removed from view. Because you are creating a 2D profile, switch to **World Plan** view.

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Use **Polyline**, **Arc** or **Circle** tools to create closed curves for boundaries and holes. In this example, the outer shape and the two rectangles were created using Polyline. The circle was created using **Circle Center and Point**.



You can have one or more boundaries, but boundaries cannot overlap. Holes must be entirely within a boundary.

8. To define which curves will be used for boundaries, select **Edit Boundary Profile**.



9. Click the boundary curve or curves, which turn blue when selected.

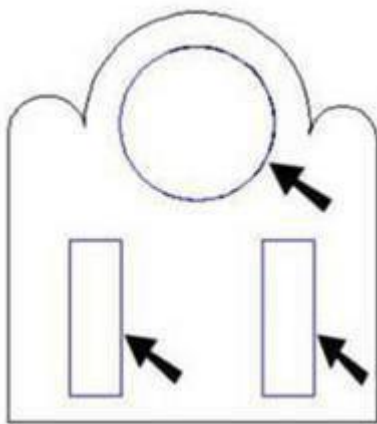
10. Select **Finish** from the local menu or Inspector Bar.

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11. To define which curves (if any) will be used for holes, select **Edit HoleProfiles**.



12. Click the hole curve or curves, which turn blue when selected.



Select **Finish** from the local menu or Inspector Bar.

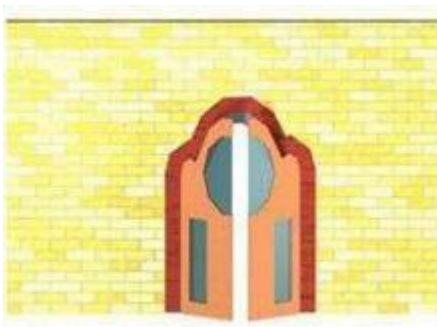
When the boundaries and holes are defined, click **Finish to Edit Geometry** in the Style Manager.



The profile now appears in the Preview area of the Style Manager. Boundary profiles are shown in blue, holes are shown in red.

Now return to the door profile. Under **Design**, set the **Profile** to the one you just created.

The door outline changes to match the profile. The hole profiles define the door glass, and these areas have the material specified for **Glass** in the door style.



If you want to make changes to the profile, highlight it in the Style Manager, and then click **Edit Content**.

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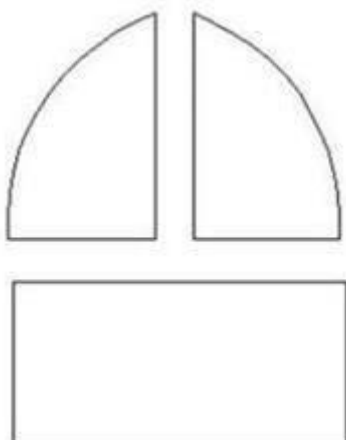
You will return to **Edit Geometry** mode, where you can make your changes. When you're finished editing, click **Finish to Edit Geometry**. The door or window that uses the profile will update automatically.

Example 2- Window Profile with Multiple Boundaries

1. In this example, create a new profile that will be used in a window.



2. This profile has three closed profiles, each created using **Polyline**.

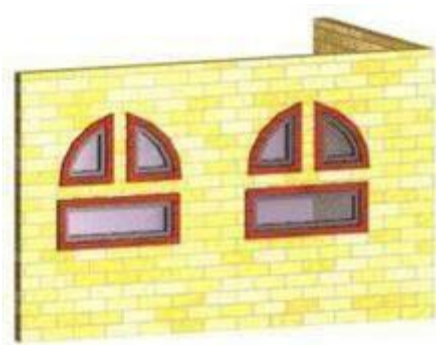


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Hole profiles are not generally needed for windows, since the boundary profiles define where the window glass is. If you define holes in window profiles, holes will be placed in the glass.

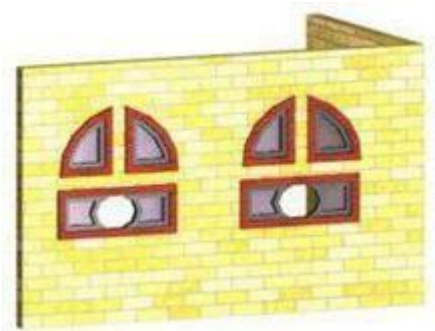
Now create a window style that uses the profile. This example uses a **Picture** window.

Use the window **Properties** to define overall dimensions, and insert a window or two in a wall. Each window consists of three parts, defined by the boundary profiles.



To see what hole profiles would do, use **Edit Content** edit the profile. Add another profile within one of the boundaries. Use **Tools / Architecture / Profile / Edit Hole Profile** to define the new curve as a hole. Then click **Finish to Edit Geometry**.

The glass now has a hole where the new profile curve was created.



Saving as Templates

(Available in Platinum, Professional and Deluxe)

Default UI Menu: File/Save As

Ribbon UI Menu:

You can save styles in template files, so that you don't have to create styles from scratch each time. To do this, set up the styles you want for doors, schedules, slabs,

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etc. Then use **File / Save As** to save the file as a **.tct file (TurboCAD Template)**. Place the template file in the "Template" folder of the TurboCAD root directory. Then when you want to open the template, use ***File / New**, and select **New from Template**.

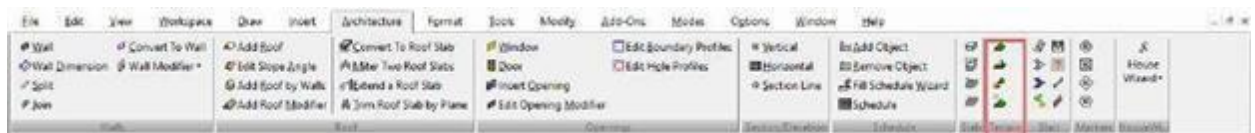
Note: This also applies to tool **Properties**; set the properties you want, such as door dimensions or wall width and height, and save them as part of the template file as well.

11.10. Terrain

(Available in Platinum and Professional)

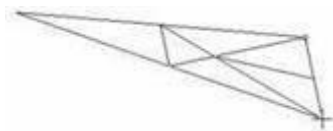
Default UI Menu: Architecture/Terrain

Ribbon UI Menu:

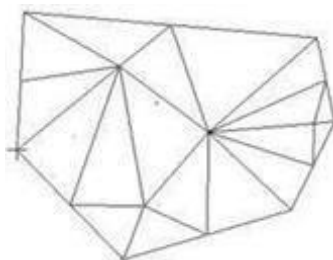


Creates a topographical terrain, represented by triangulated network. You can create a terrain from scratch, or import coordinates from a file.

Activate **Terrain**, and click points to define the outer boundary (or enter coordinates in the Coordinate Fields). As you click points, the surface triangulates.

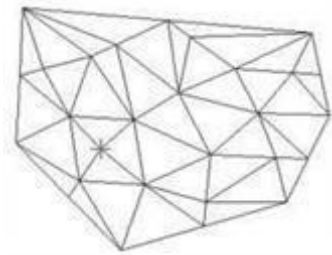


Continue defining the outer boundary.

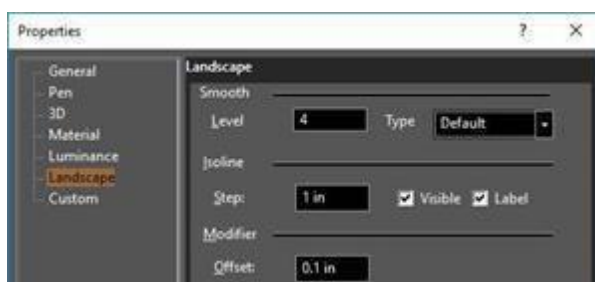


Define interior points as well. Each point you define will become a node you can edit. You do not need to define all nodes now; you can add nodes later.

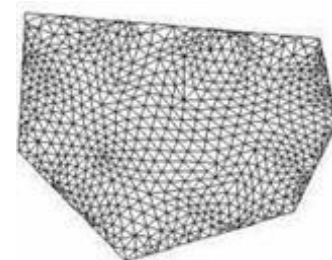
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Open the terrain's **Properties** to the **Landscape** page. Under **Smooth**, increase the **Level**.



This increases the level of triangulation, resulting in a more accurate surface.



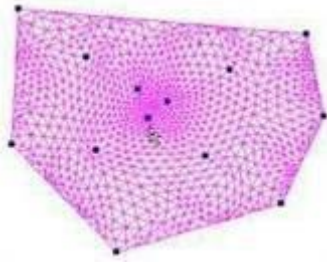
Activate the **Edit Tool** on the terrain. Each point you defined when creating the terrain is represented as a node.



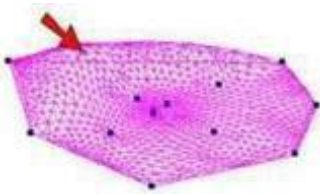
Note: For details on this tool, see [Edit Tool](#)

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If you want to add more nodes, right-click where you want the new node and select **Add Node**.



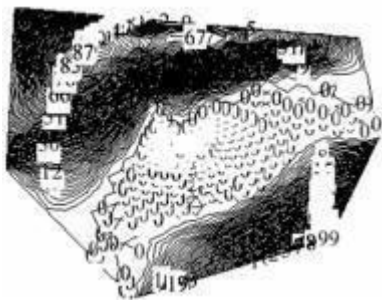
To modify the terrain, click a node and use the Coordinate Fields to change its Z position. You can also drag a node while in a Front or Side view. Adjacent nodes will remain in place.



You can also change the elevation of nodes along the boundary.



Switch to **World Plan**. This view shows the terrain as a series of isolines, each labeled with its elevation.

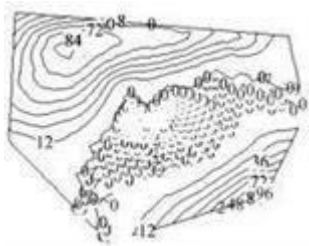


In the **Landscape** page, change the **Step**. In this example, an isoline will be displayed every 12 inches.

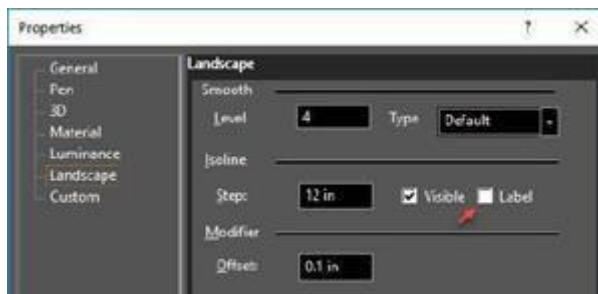
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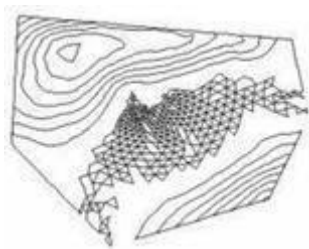
This makes the isoline display less dense.



11. Uncheck **Label**.



This removes the elevation value from each isoline.

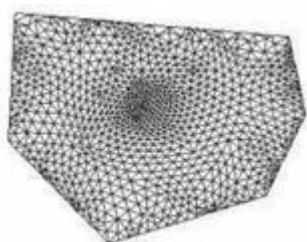


12. Uncheck **Visible**.

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This removes the isoline display.



If isolines are displayed, when you move out of World Plan view, they will not be visible.

Import Terrain



Default UI Menu: Architecture/Import Terrain

Ribbon UI Menu:



Terrain data you can import must be listed in a .txt or .xyz file. The format for each coordinate should be X, Y, and Z values, separated by commas or spaces. A ctivate **Import Terrain**, and define the original point. This is where the center of the terrain will be placed. Then select the data file.

Terrain from Selected Points

Default UI Menu: Architecture/Terrain from Selected Points

Ribbon UI Menu:



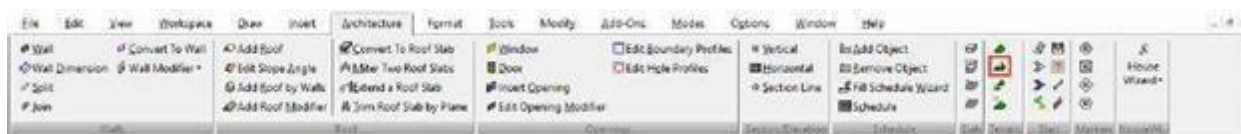
Creates a terrain from a set of selected points. The points must all have the same Z coordinate.

For details on points, see [Point](#). To create the terrain, first select the points, then activate **Terrain from selected Points**.

Terrain Modifier

Default UI Menu: Architecture/Add Terrain Modifier

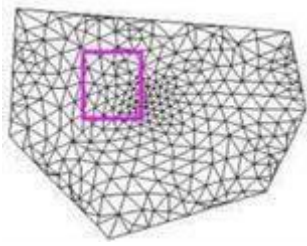
Ribbon UI Menu:



You can add or remove material from a terrain around a closed, 2D profile.

1. Start with a terrain, and create a closed 2D shape, such as a rectangle.

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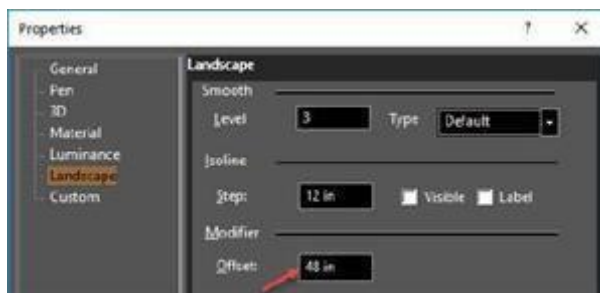
In this example, the shape is placed vertically so that part of it is below the terrain, and part of it is above.



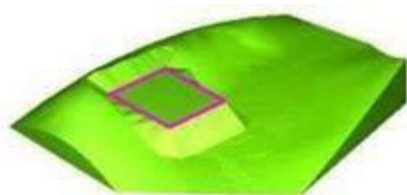
Activate **Terrain Modifier**, and select the terrain to modify. Then select the closed shape. Material is added or removed as needed to meet the shape.



To slope the terrain around the modified shape, add an **Offset** value.



This is the result:

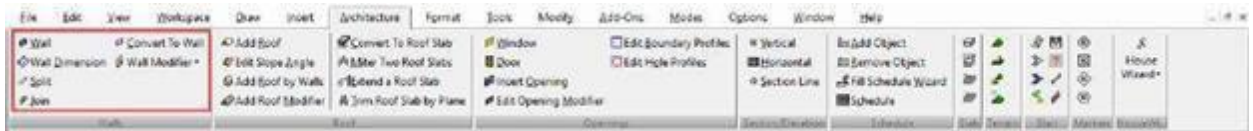


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11.11. Wall Tools

Default UI Menu: Architecture/Walls

Ribbon UI Menu:



The **Wall** tools help you quickly work out a wall design, by creating smart objects. Walls heal at intersections, and they can take block insertions that automatically align correctly.

The typical sequence to create a floor plan starts with a wall layout. When the walls have been drawn, insert doors, windows and/or openings. After that, you can adjust the wall layout by moving or rotating any wall. When you adjust a wall, the walls that are attached to it will adjust to it. Any inserted doors, windows, and openings also adjust.

Add Bottom Wall Modifiers

(Available in Platinum)

Default UI Menu: Architecture/Walls/Add Bottom Wall Modifier

Ribbon UI Menu:

Enables you to change the bottom height of a wall by trimming or extending it to a single-line object. The object can be a line, arc, or polyline. It must be at least as long as the wall, and cannot extend below the bottom of the wall. In addition, it must lie in the same workplane as the wall.

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Note: The way to assign height to a wall is to open its **Properties** to the **3D** page and enter a **Height**.

Set the workplane to the wall you want to modify. The easiest tool is **By Facet**.

Create the object you want to use as a modifier. Be sure that the object meets or

extends past the wall ends.

Activate **Add Bottom Wall Modifier**, select the wall to modify, and select the 2D modifier. The wall is trimmed or extended to meet the modifier.

The results may be easier to see in **Hidden Line** mode.

Note: If the modifier length is insufficient, you can edit it. The modifier will be applied once it reaches the required length.

If you add another Bottom modifier, it will cancel the effect of the previous one. You can use Top and Bottom modifiers on the same wall.

Add Top Wall Modifiers

(Available in Platinum)

Default UI Menu: Architecture/Walls/Add Top Wall Modifier

Ribbon UI Menu:

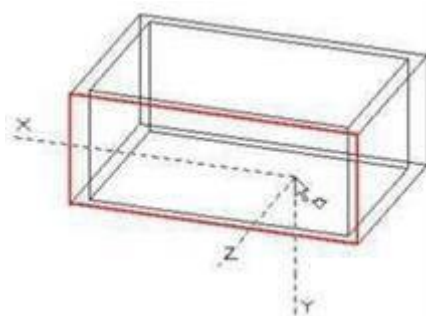


Enables you to change the top height of a wall by trimming or extending it to a single-line object.

The object can be a line, arc, or polyline. It must be at least as long as the wall, and cannot extend below the bottom of the wall. In addition, it must lie in the same workplane as the wall.

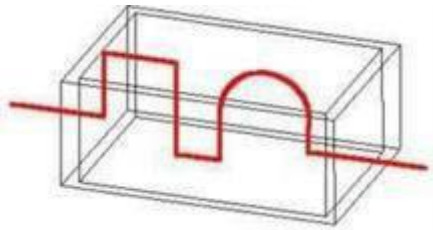
Note: The way to assign height to a wall is to open its **Properties** to the **3D** page and enter a **Height**.

1. Set the workplane to the wall you want to modify. The easiest tool is **By Facet**.

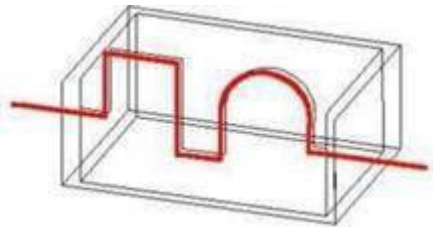


Create the object you want to use as a modifier. Be sure that the object meets or extends past the wall ends.

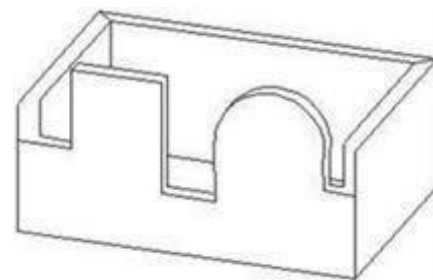
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Activate **Add Top Wall Modifier**, select the wall to modify, and select the 2D modifier. The wall is trimmed or extended to meet the modifier.



The results may be easier to see in **Hidden Line** mode.



Note: If the modifier length is insufficient, you can edit it. The modifier will be applied once it reaches the required length.

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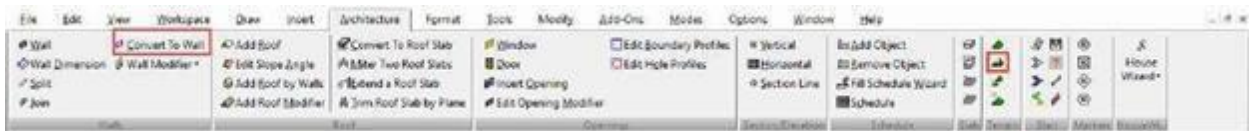
If you add another Top modifier, it will cancel the effect of the previous one.

Convert to Wall

(Available in Platinum)

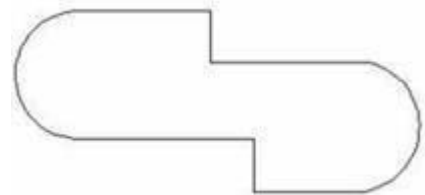
Default UI Menu: Architecture/Walls/Convert to Wall

Ribbon UI Menu:



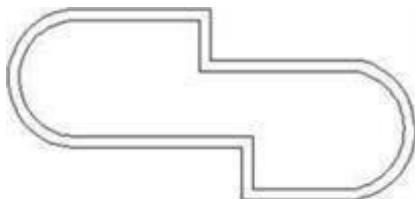
Converts single-line objects into walls. This is useful for creating curved walls.

Select the object to convert. You can select lines, arcs, circles, polylines, etc.



Multiple objects can be selected.

Select **Finish** from the local menu or Inspector Bar. The wall is created with the default thickness, but you can modify its **Properties**,



Note: The way the wall is created in relation to the original object depends on the **Reference** parameter in the **Wall** page of the **Properties** window.

Editing Walls

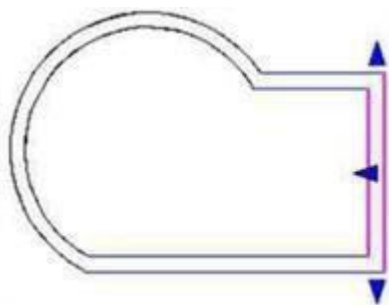
(Available in All TurboCAD Variants)

The **Edit Tool** can be used to move or resize walls.



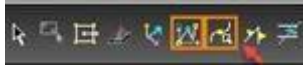
Note: For details on this tool, see [Edit Tool](#)

This example shows a set of walls with one arc segment. Activate the **Edit Tool** and click the segment you want to change.

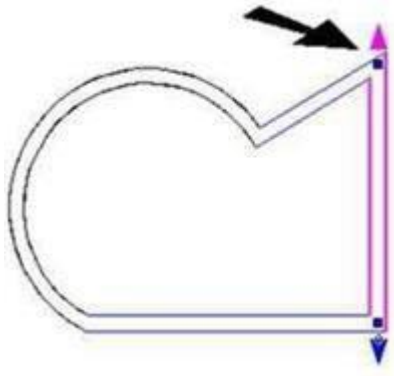


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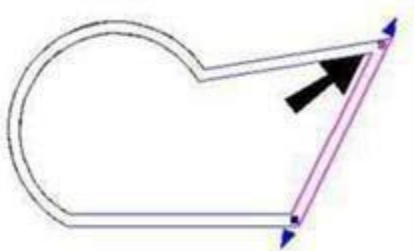
Select Edit Nodes.



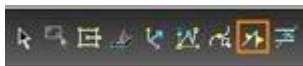
With this options, you can move the node at either end. If you click and drag the arrow, you are constrained to the line of the wall.



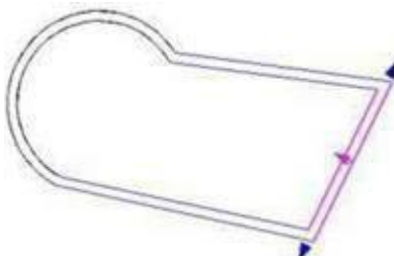
4. If you click and drag a node, you can move it anywhere.



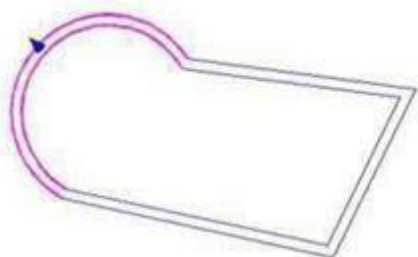
Switch to Edit Segments.



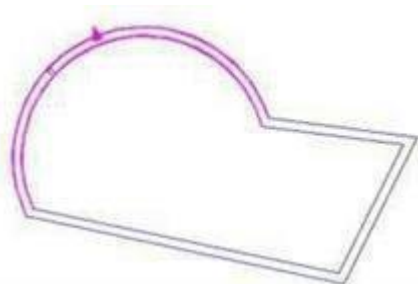
Now you can move the entire wall, constrained to its current orientation



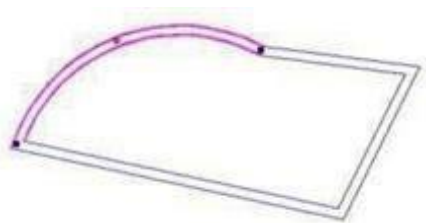
7. You can also change arc segments with the **Edit Tool**.



8. With **Edit Segments**, the arc center stays the same and the arc radius changes.



9. With **Edit Nodes**, the arc endpoints stay the same.



Inserting 2D Blocks in Walls

(Available in All TurboCAD Variants)

When a 2D block is inserted into a wall, it rotates to align itself with the wall. The direction of the rotation depends upon the direction in which the wall was drawn; the top of the block will align to the right side of the selected wall. (The right side is determined by facing the wall's start point from the end point.)

Note: This method of inserting blocks applies to 2D blocks, and 2D (plan) representation of walls. To create walls and doors that you can see in 2D as well as 3D, see [Openings](#)

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Create the 2D representation of the desired insertion (door, window, etc.). Create a block from these objects. You can do this by selecting the objects and dragging them into the Blocks Palette, or by selecting **Format / Create Block**.



For more information on blocks, see [Blocks](#)

Drag the block from the Blocks Palette onto the wall at the point where you want to insert it. The block will automatically align itself with the wall, and a cut out will be created.



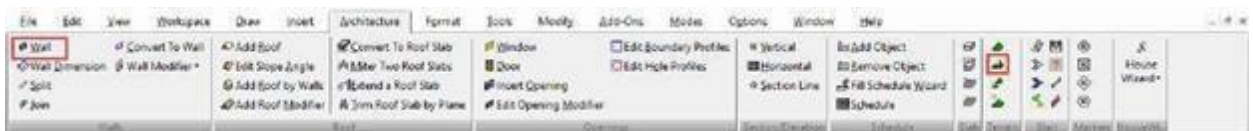
Inserting Walls

(Available in All TurboCAD Variants)



Default UI Menu: Architecture/Walls/Wall

Ribbon UI Menu:



Creates segments of double lines that represent wall layouts. The difference between this tool and the **Double Line** tools is that walls automatically heal at corners and intersections. In addition, you can attach walls to existing walls which will also heal.

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Justification of walls:

There are four types of justification Left, Center, Right and Baseline. Justification allows you to select the alignment of every new wall. You can select the options from local menu or inspector bar.

Demo Video:

<https://youtu.be/-gVoySw0KGo>

1. Select two points to draw the first wall segment.



Tip: Use Shift to activate **Ortho** mode if you want to draw horizontal and vertical walls.

Select the endpoint for the next wall segment. The corner is automatically healed; no intersection lines are shown.

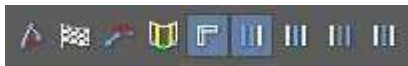
TurboCAD 2023 User Guide Publication



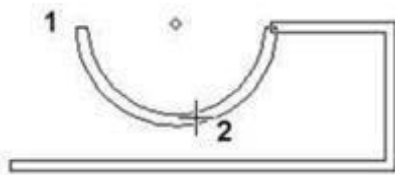
3. Select more wall endpoints as needed.



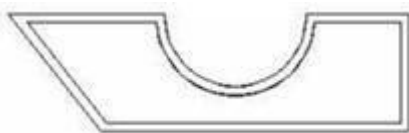
To add a circular wall segment, select **Arc Wall*** from the local menu or Inspector Bar and.



To create the arc segment, first select the endpoint, then select a point along the circumference.



To return to linear walls, turn off **Arc Wall**. Continue selecting wall endpoints. Select **Finish** from the local menu or Inspector Bar. If you want to close the wall, select **Close** from the local menu or Inspector Bar.



Any wall segment can be deleted or moved like any other object.

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You can now choose to enter an Angle (which is absolute in relation to the world coordinates) or a **Relative Angle** which is the angle from the immediately previous segment (arc or line). For the Wall tool, the Relative Angle is always enabled. At the first state we draw a straight wall (the relative angle is enabled)

The Relative Angle is also enabled with Curved Walls.

Demo Video:

<https://youtu.be/nIWELbMwCXs>

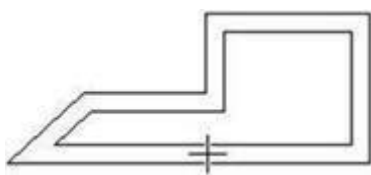
Linking and Healing

Walls have two aspects which define how they interact with one another: Linking and Healing. Linking means that walls are associative (attached). If you move or modify a wall, walls that are linked to it will modify themselves to remain attached. Healing means that the intersection of walls is cleaned to show a joint without overlapping elements.

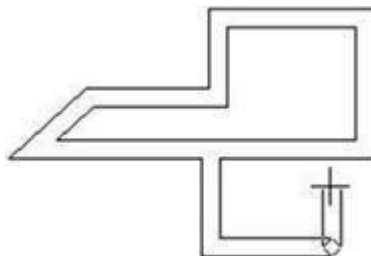
Attaching Walls

Any wall you add to an existing wall will be automatically healed at their intersections.

1. Click inside an existing wall at the point where you want to add another wall.



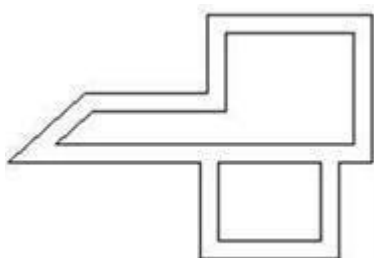
2. Select segment endpoints.



3. If you extend a segment to an existing wall, this intersection will be healed as well.

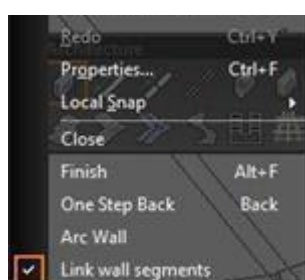
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If you extend a segment using the T-Meet tool or Meet 2 Lines tool into an existing wall, this intersection will be healed and linked.



Link Wall Segments

The Link Wall Segments (local menu) option controls whether walls become linked as they are drawn. It is on by default, but If it is turned off walls will not become linked. You can tell if a wall is linked by selecting it. Any other segments linked to that wall will be highlighted. Since walls automatically linked when you draw then in proximity this option can be useful for drawing a wall close to another wall without having them link. For example, this can be useful for drawing cubicle walls in an office setting.



Drop Link between Wall Segments

You can break the link between any two walls segments.

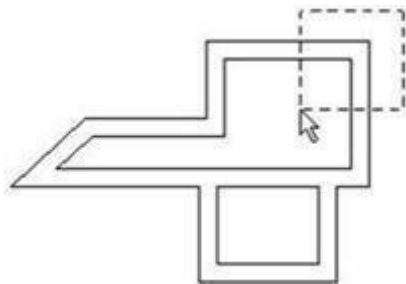
Select the two segments

Right click to open the local menu and select Drop Link.

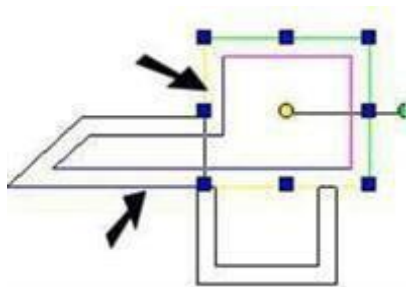
Moving Walls

To move a single wall, select it as you would any other object and use the **Select Edit** tools to move, scale, or rotate it. (See [Select Edit](#)) Moving a wall will affect its adjacent walls.

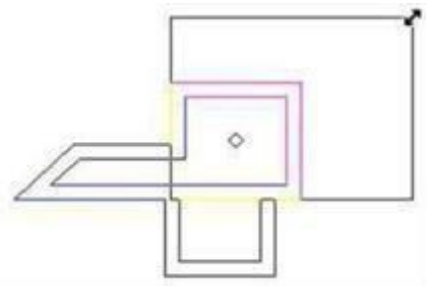
In **Select** mode, select the wall you want to move. To select multiple walls, use Shift or drag a selection window.



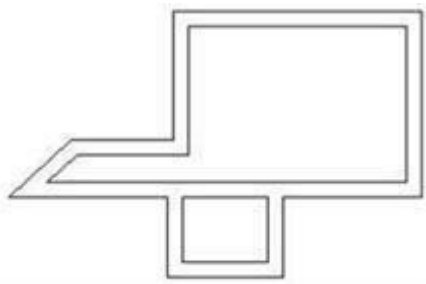
In addition to the walls you selected, adjacent walls are marked in blue, so that you will see all affected walls before performing the edit.



2. Use the **Select Edit** tools to move, rotate, or scale the selected wall or walls.



The affected walls extend or shrink to meet the new walls, but their orientation does not change. If the walls contain inserted windows, door, or other blocks or openings, they will adjust as well.



Roof Wall Modifier

(Available in Platinum)

Default UI Menu: Architecture/Walls

Ribbon UI Menu:

Enables you to extend the top or bottom of a wall to meet the roof (this does not work with Roof Slabs). This is especially important if you have edited the Slope angle of a roof so that you have an open gable end.

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Note: The way to assign height to a wall is to open its **Properties** to the **3D** page and enter a **Height**.

Select the Roof Wall Modifier tool.

From the Local menu choose either



3. Select the wall you want to modify..

4. Select the roof to which you want the wall to extend.

5. The wall will now extend to the roof.

Top/Bottom Wall Modifiers will override Roof Wall Modifiers, and Roof Wall Modifiers will override Top/Bottom Wall Modifiers, depending upon the sequence in which they are added.

Splitting and Joining Walls

(Available in Platinum)

Default UI Menu: Architecture/Walls/Split, Architecture/Walls/Join

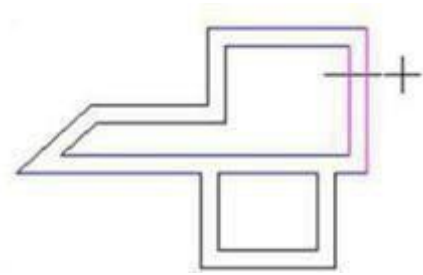
Ribbon UI Menu:



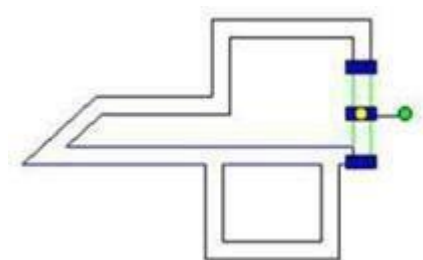
Splits a wall into two separate walls, and joins split walls into one wall. This tool works on straight walls only, not curved walls. To use the **Split** tool:

Select the wall you want to split.

Position the cursor where you want to split the wall, and click to split.

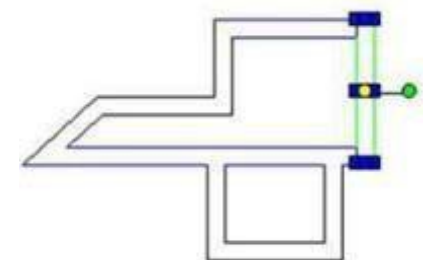


Because walls automatically heal, you will not be able to see the split (unless you use end caps). If you select one of the walls, you can see that the original wall was split.



To use the **Join** tool:

4. Select the two segments you want to join. The walls are combined into one wall.



Note: Most standard Modify tools, such as Trim, Split, and Meet 2 Lines, will work with walls.

Wall Dimension

(Available in Platinum)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Architecture/Walls Dimension

Ribbon UI Menu:



Specialized architectural dimensioning tool for walls. This tool is used in conjunction with the Style Manager. See [AEC Dimension Styles](#)

Wall Properties

(Available in All TurboCAD Variants)

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Options for wall alignment, thickness, end caps, and direction.

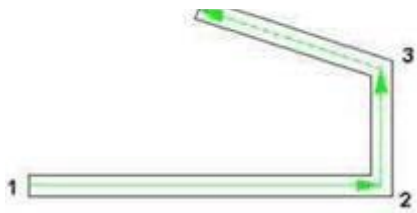
Thickness: The thickness of the wall (distance between the double lines).

Height: The vertical height of the wall.

Autocleanup: This option sets whether the wall will automatically cleanup (heal) at intersection with other walls. This can be useful if you want to show the overlap between walls

Show Miter to close the ends of a wall.

Show Direction: Displays a wall's direction, from its start point to its end point. This is useful when defining end caps, and when assigning materials to different sides of the wall.



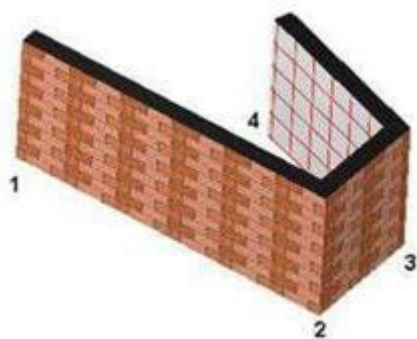
Note: The location of the directional arrow depends on which **Reference** option was used (Left, Right, or Center.)

Wall Side

(Available in Platinum and Professional)

Side: **Left** and **Right** depend on the wall's direction, assuming you are standing at the start point and facing the end point. To show the directional arrow, check **Show Direction** on the **Wall** page of the **Properties**. For details on materials, see [Materials](#)

In this example, the walls were created in the order shown. The **Right** sides of the walls have a brick material, and the **Left** sides have a grid pattern.



11.12. Windows and Doors

(Available in Platinum, Professional and Deluxe)

Default UI Menu: Architecture/Doors, Windows and Openings

Ribbon UI Menu:



These two architectural tools enable you to insert "smart" windows and doors into walls. Large libraries for doors and windows are available, and all features and dimensions can be specified.

Door



Default UI Menu: Architecture/Doors, Windows and Openings/Door

Ribbon UI Menu:



Inserts "smart" doors into walls. Before inserting doors, you first need to use the Style Manager to define door styles. For the entire process of creating doors, see [Door Styles](#). For the various types of doors you can insert, see [Door Styles](#)

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Select the tool and hover over a wall.

Click on the wall to insert the door.

You can use snaps to exactly place the door. You will also be shown dynamic dimensions when you are inserting the window that indicate the doors distance from the two nearest walls.

Dynamic dimensions will also appear when you use the Edit tool on doors.

Window



Default UI Menu: Architecture/Doors, Windows and Openings/Window

Ribbon UI Menu:



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Inserts "smart" windows into walls.

Before inserting windows, you first need to use the Style Manager to define window styles. For the entire process of creating windows, see [Window Styles](#) For the various types of windows you can insert, see [Window Styles](#)

Select the tool and hover over a wall.

Click on the wall to insert the window.

You can use snaps to exactly place the window. You will also be shown dynamic dimensions when you are inserting the window that indicate the windows distance from the two nearest walls.

Dynamic dimensions will also appear when you use the Edit tool on windows.

11.13. Using the Trim Tool with Architectural Objects

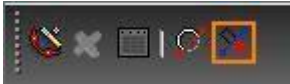
(Available in Platinum)

The Trim tool can be used to trim Roof Slabs, and Walls with closed 2D objects. The 2D object must be on the same workplane as the 3D entity to be trimmed. Use Workplane by Entity to set the correct workplane before drawing the trimming object. Since the workplane for walls is horizontal not vertical the effects of the trim are more limited.

To Trim an architectural object with the Trim tool:

Select the Trim tool.

In the Inspector bar or Local menu select the **Extract closed contour from closed entity** option.



Select the object that will be used as the trimming entity.

4. Select the object to be trimmed.

5. Select the part of the entity that is to be trimmed away.

11.14. AEC Rectangular Grid

(Available in all variants)



Default UI Menu: Architecture/ Architectural Grid/AEC Grid

Ribbon UI Menu:

The Architectural Grid makes it easy to align other architectural objects in the drawing, such as walls and columns. Users can also import rectangular grids from DWG drawings and can change the properties of the grid and bubble representation, via the Selection Info palette.

Create an AEC Grid:

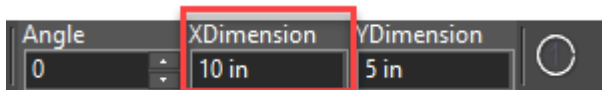
1. Select AEC Grid and specify position with the first click.
2. Specify the second point to set the rotation angle of the Grid.
3. The third click will specify X and Y dimensions.

Local menu options:

Angle: Defines the rotation angle of the Grid to be placed.



XDimension: Defines the x-dimension of the Grid.



YDimension: Defines the y-dimension of the Grid.



Create default block: Creates the default block for bubble Grid.



GRID SETTINGS AND PROPERTIES:

Spacing:

Spacing type is specified separately for the X and Y directions.

Manual: you specify the position of each line on the grid. **Even:** you specify equidistant spacing between lines on the grid.

The number of lines is determined by the size of the grid. As the size of the grid changes, lines are added or subtracted accordingly. The distance between lines remains fixed. Note: by default, grids are inserted with equidistant spaced lines.

On a manual grid, you can add or remove each line individually. Select the grid to which you want to add or remove lines by edit tool. A manual grid has grip points for every line.

To remove line select the grip point of line that you want to remove and choose "Delete Grid Lines" from local menu. To add line choose "Add Grid Lines" from local menu on the place that you want to add this. Also you can use grip points of lines to specify line positions on the grid moving (dragging) them.

Dimension(X/Y): Increase or decrease the overall grid size.

Bay Size(X/Y): Increase or decrease the size of the bays.

Default Bubble Block:

When creating a grid and selecting "Create default block" from the local menu, it creates labels with bubbles. You can change your current grid label to the bubble block by selecting the block created in the Bubble Properties section.

The "**Keep Label**" option adds the ability to save the axis name without changing it, regardless of the order of the axes in the grid.

11.1. AEC Radial Grid

(Available in All TurboCAD variants)



Default UI Menu: Architecture/ Architectural Grid/AEC Radial Grid

Ribbon UI Menu:

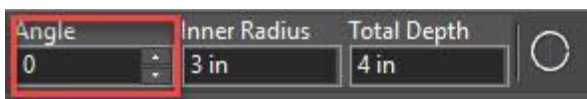
The Architectural Radial Grid makes it easy to align other radial architectural objects in the drawing, such as walls and columns. Users can also import rectangular grids from DWG drawings and can change the properties of the grid and bubble representation, via the Selection Info palette.

Create an AEC Radial Grid:

1. Select AEC Radial Grid and specify position with the first click.
2. Specify the second point to set the rotation angle of the Grid.
3. The third click will specify the inner radius of the Grid.
4. The fourth click will specify the total depth of the Grid.

Local menu options:

Angle: Defines the rotation angle of the Grid to be placed.



Inner Radius: Defines the inner radius of the Grid.

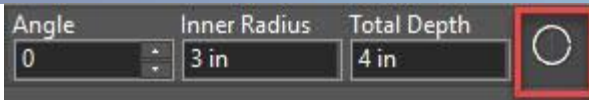


Total Depth: Defines the total depth of the Grid.



Create default block: Creates the default block for Bubble Grid.

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GRID SETTINGS AND PROPERTIES:

Spacing:

Spacing type is specified separately for the X and Y directions.

Manual: you specify the position of each line on the grid. **Even:** you specify equidistant spacing between lines on the grid.

The number of lines is determined by the size of the grid. As the size of the grid changes, lines are added or subtracted accordingly. The distance between lines remains fixed. Note: by default, grids are inserted with equidistant spaced lines.

On a manual grid, you can add or remove each line individually. Select the grid to which you want to add or remove lines by edit tool. A manual grid has grip points for every line.

To remove line select the grip point of line that you want to remove and choose "Delete Grid Lines" from local menu. To add line choose "Add Grid Lines" from local menu on the place that you want to add this. Also you can use grip points of lines to specify line positions on the grid moving (dragging) them.

Dimension(X/Y): Increase or decrease the overall grid size.

Bay Size(X/Y): Increase or decrease the size of the bays.

Default Bubble Block:

When creating a grid and selecting "Create default block" from the local menu, it creates labels with bubbles. You can change your current grid label to the bubble block by selecting the block created in the Bubble Properties section.

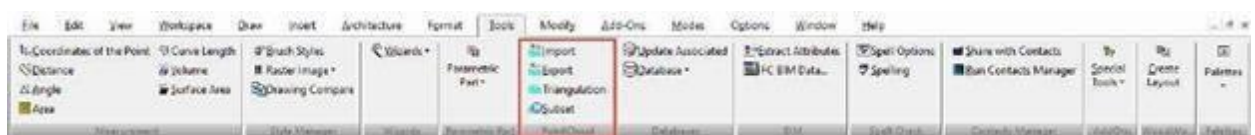
By default, the label value is generated according to the order of the axes. The value of the axis bubble changes if the axis moves backwards or forwards behind a fixed axis. The "**Keep Label**" option adds the ability to save the axis name without changing it, regardless of the order of the axes in the grid.

12. Point Clouds

(Available in Platinum and Professional)

Default UI Menu: Tools/Point Cloud

Ribbon UI Menu:



A point cloud is a set of data points in some coordinate system. In a three-dimensional coordinate system, these points are usually defined by X, Y, and Z coordinates, and often are intended to represent the external surface of an object.

12.1. Exporting Point Clouds

Exporting Point Clouds



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/Point Cloud/Export Point Cloud

Ribbon UI Menu:



To Export a point cloud:

1. Select the Export Point Cloud tool

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Click on the point cloud to be exported.

From the file dialog, select a name and location for the file. (Only *.PCD is

supported)

Click save. The point cloud will be saved.

Repeat step 3 and 4 as many times as needed. Press ESC to drop the tool

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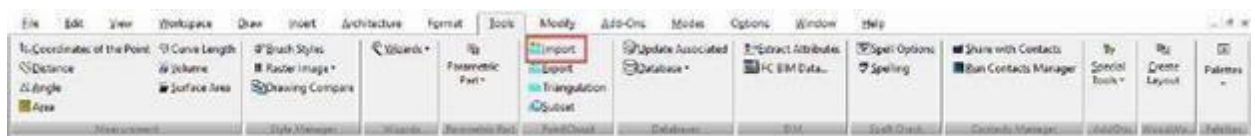
12.2. Importing Point Clouds



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/Point Cloud/Import Point Cloud

Ribbon UI Menu:



This tool allows you to import a point cloud:

To import a point cloud:

Select the Import Point Cloud tool

Click to set the location for the cloud.

From the file dialog select an appropriate point cloud file. (the file types that work are *.PCD, *.XYZ, *.ASC) Then include ASC files with color data.



4. Click open. The point cloud will be inserted into your model.

5. Repeat step 3 and 4 as many times as needed. Press ESC to drop the tool

12.3. Point Cloud Properties

Path: File name of the source, containing the set of points. Available file extensions - "pcg", "pcd" and "asc" - text file with format (x, y, z) or (color, x, y, z) **(cannot be modified)**

Width: Specifies the width of the point cloud dataset in the number of points. Width has two meanings: **(cannot be modified)**

it can specify the total number of points in the cloud for unorganized datasets;

it can specify the width (total number of points in a row) of an organized point cloud dataset

Height: Specifies the height of the point cloud dataset in the number of points HEIGHT has two meanings: **(cannot be modified)**

it can specify the height (total number of rows) of an organized point cloud dataset;

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It is set to 1 for unorganized datasets (thus used to check whether a dataset is organized or not).

Count: Total number of points in the set. Count = Width*Height (**cannot be modified**)

Pixel size: point size when drawing; 1x1 or 2x2 pixels. Use for GDI draw only.

Density: Use for GDI draw performance, shows each 1/density points.

Show Frame: Shows the bounding frame around the point cloud.

Scale: Sets the scale of the point cloud. Decimal values are accepted.

12.4. Subsetting Point Clouds



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/Point Cloud/Subset Point Cloud

Ribbon UI Menu:



This tool allows you to create a subset point cloud from an existing point cloud. To work a sphere or box which surrounds the subset must be drawn first. To subset a point cloud:

1. Select the Subset Point Cloud tool.

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Click on the point cloud from which the subset is to be derived.

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Click on the sphere or box which bounds the desired subset.

Click Finish.

From the file dialog, select a name and location for the file. (Only *.PCD is

supported)

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Click save. The point cloud will be saved.

the subset will also be in your drawing.

12.5. Triangulating Point Clouds

Default UI Menu: Tools/Point Cloud/Point Cloud Triangulation

Ribbon UI Menu:



This tool is used to create a triangulated mesh from a point cloud.

The method works by maintaining a list of points from which the mesh can be grown ("fringe" points) and extending it until all possible points are connected. It can deal with unorganized points, coming from one or multiple scans, and having multiple connected parts. It works best if the surface is locally smooth and there are smooth transitions between areas with different point densities.

Select Point Cloud object

Define triangulation method parameters

3. Click OK.

The point cloud is retained:

Parameters:

Triangulation is performed locally, by projecting the local neighborhood of a point along the point's normal, and connecting unconnected points. Thus, the following parameters can be set:

Maximum Nearest Neighbors and **Mu** control the size of the neighborhood. The former defines how many neighbors are searched for, while the latter specifies the maximum acceptable distance for a point to be considered, relative to the distance of the nearest point (in order to adjust to changing densities). Typical values are 50-100 and 2.5-3 (or 1.5 for grids).

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Maximum Edge Length is practically the maximum edge length for every triangle. This has to be set by the user such that to allow for the biggest triangles that should be possible.

Minimum Angle and **Maximum Angle** are the minimum and maximum angles in each triangle. While the first is not guaranteed, the second is. Typical values are 10 and 120 degrees (in radians).

Maximum Surface Angle is meant to deal with the cases where there are sharp edges or corners and where two sides of a surface run very close to each other. To achieve this, points are not connected to the current point if their normals deviate more than the specified angle (note that most surface normal estimation methods produce smooth transitions between normal angles even at sharp edges). This angle is computed as the angle between the lines defined by the normals (disregarding the normal's direction) if the normal-consistency-flag is not set, as not all normal estimation methods can guarantee consistently oriented normals. Typically, 45 degrees (in radians) and false works on most datasets.

Database, Tables, and Reports

Database, Tables, and Reports

This section covers the creation, editing and use of databases, tables, and reports.



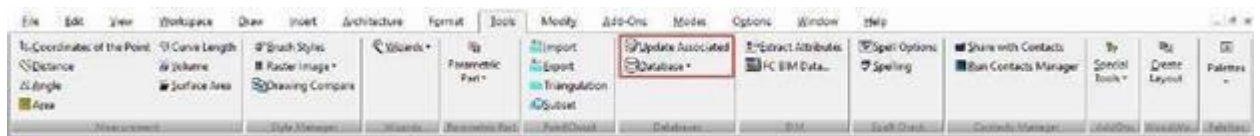
Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

13.1. Custom Properties, Database, and Reports

(Available in Platinum and Professional)

Default UI Menu: Tools/Database

Ribbon UI Menu:



All properties of objects are elements of a database. Standard database property items include line width, color, coordinates of points that comprise the object, etc.

The database management feature enables you to add information to objects and to create reports using this information. For example, you can add custom fields to provide information on suppliers, manufacturers, price, etc. The information can then be presented as a Bill of Materials, Parts List, or other type of report.

Database management and report creation generally follows these four steps:

Step 1: Define Custom Fields

Step 2: Attach Custom Properties to Objects

Step 3: Define Fields for the Report

Step 4: Create the Report

The following example shows how to create some custom properties, then create a report from the custom and standard properties. The objects in the report are shown below: a large plastic disc, two cubes on top, and six steel cylinders below.

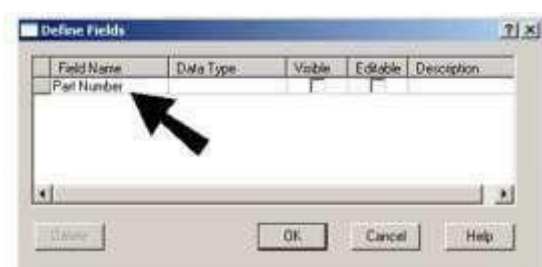
Step 1- Define Custom Fields

Default UI Menu: Tools/Database/Define Fields

Ribbon UI Menu:

The first step in creating a database is to define the types of information to be recorded.

Select **Define Fields**. In the **DefineFields** window, enter the first property: "Part Number."



2. Fill in the remaining fields in this row



Data Type: Text: Alphanumeric information which does not have to be calculated.

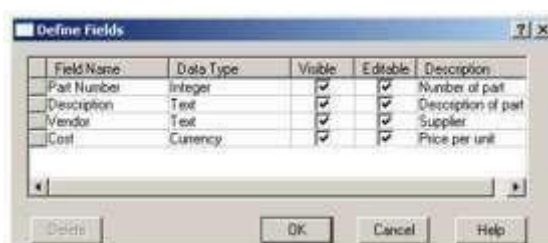
Integer: Whole numbers. **Double:** Decimal numbers. **Currency:** Dollar amounts. This type will not appear in an object's **Custom** page of the **Properties** window. **Visible:** If checked, the field will be visible and available for use in the **Edit Object Data** and **Report** windows.

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Tip: If you know you will not be using certain fields in reports, it is helpful to turn off visibility.

Editable: If checked, the field can be edited in the other database dialogs. Auto fields are always non-editable. **Description:** An optional field for notes or other comments.

To add the next property to the database, click in the first empty field under **Field Name**, and type the new field name. This example uses "Description," "Vendor," and "Cost." Then continue to fill in the other fields.



Note: To delete a record, click the square at the beginning of a row, and select **Delete**.

4. Click OK when the custom properties are defined.

Step 2- Attach Custom Properties to Objects

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Select the object whose custom properties you want to define. If multiple objects have the same custom properties, such as the cubes below, select them all.

2. Open the **Properties** to the **Custom** page. Click **Attach** at the bottom of the window.

3. Select all of the custom properties and click OK.

4. Now the properties appear on the **Custom** page. Fill in the **Value** for each part.

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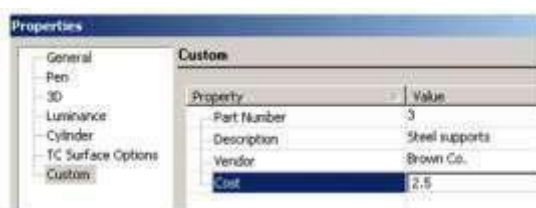


You can also enter or change custom properties in the Selection Info palette.

The previous values were for the two cubes (Part Number 1); the values below are for the large disc (Part Number 2).



6. These values are for the six support cylinders (Part Number 3).



7. To verify that the custom properties were properly attached, select one of the objects.

Then select **Tools / Database / Edit Object Data**. You can update values in this table, if needed.

Step 3- Define Fields for the Report

Default UI Menu: Tools/Database/Report

Ribbon UI Menu:

1. To begin defining the report, select **Tools / Database / Report**. Then click **Create**.

Enter a report **Title** and **Comments** (optional). If you choose **Detail**, only values from editable fields may be edited before the report is created. With **Summary**, all values may be edited before printing, even if they come from a non-editable field.

Note: "Object ID" is a unique identifier, and is generated automatically when attaching fields to objects.

The custom fields appear under the "Custom" category. Click the "plus" sign to expand this category.

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For each field you want to appear in the report, click it under **Available Fields**, then click the **Add** button. The field should then appear under **Report Fields**. Repeat these steps for each of the custom fields.

Note: If you have block in your drawing with attributes you can add those attributes to your report.

Non-custom (standard) fields can also be added to a report. In this example **Material** is added, and **Material** is found in the "Common" category.

You do not have to create a report using custom fields; reports may contain only standard fields.

The order of the Report Fields list is the order that will in the report. To change the order, highlight the field you want to move and click **Up** or **Down**. (You will also be able to change the order later.)

You can use a query if you want to filter the objects to be included in the report. If you want to run a query, click **Edit** . Otherwise, all objects in the model will be scanned while creating the report.

An example of a query in this example would be to search for objects that have "Part Number" fields greater than one. This is how the syntax would look:

8. If you created a query, click OK to return to the **Report Create** window.

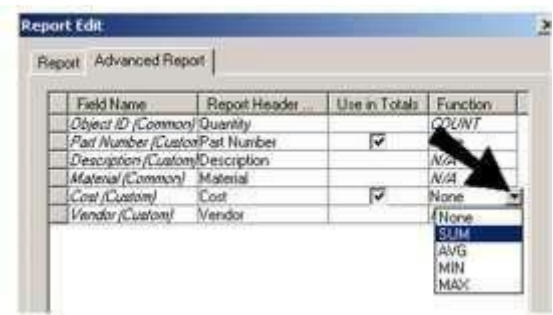
Note: For details on query searches, see [Select by Query](#)

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Open the **Advanced Report** table.

The appearance of this tab depends on whether you are creating a **Detail** or **Summary** report. For a **Detail** report, **Field Names** can be edited. For a **Summary** report, **Report Header Names** may be edited.

For the "Copy" row, click the cell in the **Function** column. From the drop-down menu, select **SUM**. This will show the total cost of all objects, not the unit price per object.



The other function options are **AVG** which calculates the average of all values, and **MIN** / **MAX** which displays the smallest or largest of the values.

11. When all fields and queries are defined, click OK to return to the **Reports** window.

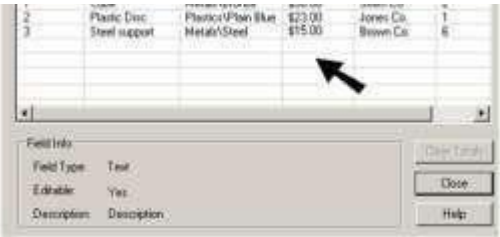
Step 4- Create the Report

Default UI Menu: Tools/Database/Report

Ribbon UI Menu:

The **Reports** window now shows the report you have just created. (You can return to the **Reports** window anytime by selecting **Tools / Database / Report.**) Click **Report**.

The **Summary Report** window shows the fields and values that will go in the report. If you want to re-order a column, drag its header to the new location. You can also click on a header to change that column's sorting order. To add a row for cost sums, click in a



cell below the last "Cost" items.

3. Then click **Grand Total**.

The sum of the cost items is now listed.

The other available options at the top of the **Summary Report** window are: **Export:** Saves the report as a Microsoft Excel Spreadsheet (XLS), Microsoft Access Database (MDB), a Comma Separated Value (CSV), or a Tabbed Text File (TXT). **Page Setup:** Sets the format of a printed version of the report. You can add the page number, number of total pages, date, and/or time to the header or footer of each report page. You can also set the margins, alignment, page order, and scale of the report pages, and whether grid lines, row headings, and column headings will be printed. **Print:** Prints the report using the **Print Setup** options.

4. To place the report in the file, click **Insert Table into the Drawing**.

In the **Insert Table** window, set the sizing options for rows and columns. You can also set the table style here.

6. Click to place the report as a table in the drawing.

The table contains the custom fields, as well as the **Materials** field.

List of Parts					
Part Number	Description	Material	Cost	Vendor	Quantity
1	Blocks	Metals/Bronze	\$30.00	Smith Co.	2
2	Plastic Disc	Plastics/Plain Blue	\$28.00	Jones Co.	1
3	Steel supports	Metals/Steel	\$15.00	Brown Co.	6
Totals:			\$73.00		

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Note: If you want to edit or add text to a table, see [Modify Table](#). For other changes such as adding or removing rows / columns, or merging cells, see [Changing Rows and Columns, Merging Cells](#)

If you want to change the style of the report table, make sure the style is defined under **Table Styles** in the Style Manager.

Note: For details on table styles, see [Table Styles](#)

8. The style is assigned to the table in the **General** page of its **Properties**.

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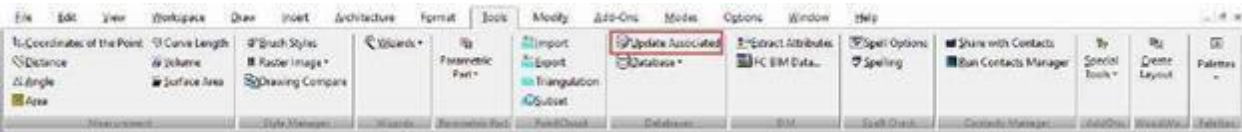
If you change the style, the table will update automatically.

List of Parts					
Part Number	Description	Material	Cost	Vendor	Quantity
1	Blocks	Metals/Bronze	\$30.00	Smith Co.	2
2	Plastic Disc	Plastics/Plain Blue	\$28.00	Jones Co.	1
3	Steel supports	Metals/Steel	\$15.00	Brown Co.	6
Totals:					
14			\$73.00		

Updating the Database and Report

Default UI Menu: Tools/Database/Update Associated

Ribbon UI Menu:



The table generated by report has the ability to update contents.

Create Table

Insert table in the drawing

Update Association:

To update the table with new entities. Select table and update the table using “Update associated”.

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Demo Video:

<https://youtu.be/latRrmSWSzU>

13.2. Database Connection Palette

(Available in Platinum)

Default UI Menu: Tools/Palette/Database Connect

Ribbon UI Menu:

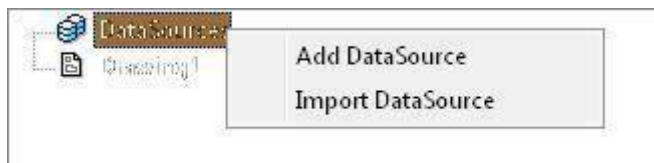
TurboCAD has the ability to connect to a database and associate the data with objects. Database connectivity and management is handled through the Database Connection Palette.

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To Create a Database Connection:

Open the Database Connection Palette.

Right click on the **DataSources** icon and click **Add DataSource**.



In the **Add Datasource dialog** type in the name of the new datasource and click **OK**.

The **Data Link Properties** dialog will open.

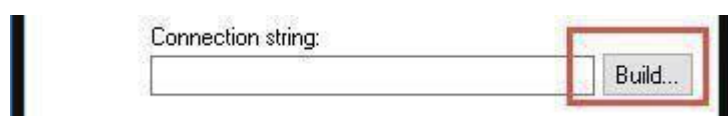
On the **Provider** page select the appropriate **OLE DB Provider** type for the datasource to which you wish to connect. Then click **Next** .

6. The **Connection** page will appear.

Specify the database source. or

Use connection String

Click Build



2. Select a File Data Source or Machine Data Source

3. Set login info if needed, then press OK.

4. Select the Database.

Setup the Server login information.

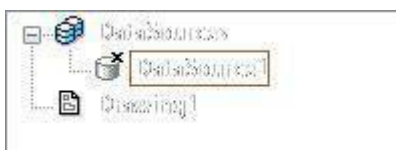
Click the **Test Connection** button. If the connection is valid the **Test Connection Succeeded** message will appear.

11. At this point you can move on to the **Advanced** and **All** pages, or click **OK**.

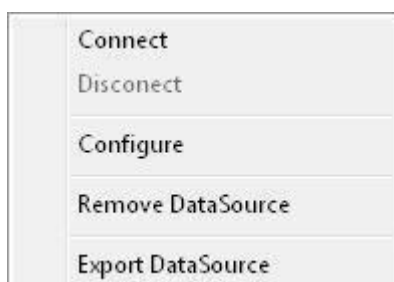
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The features and functions of the Advanced and All pages are network and database specific, and are dependent upon your network and ODBC/OLE DB protocols. Only a person and experienced with your network and database protocols should alter these settings.

After these operations you will see a new datasource object in **DataSources** tree.



The datasource is not yet connected. Right-click on the datasource and select **Connect**.



Your database is now connected to TurboCAD as a datasource.

When connected a datasource shows all available tables and views in database.

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To Disconnect a Datasource:

Select a connected datasource in the **DataSources** tree.

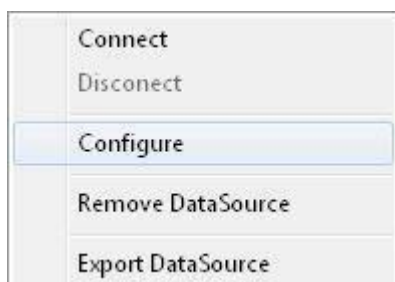
Right-click on the datasource and select **Disconnect**.



To Re-configure a Datasource:

Select a datasource in the **DataSources** tree.

Right-click on the datasource and select **Configure**. This will open the **Data Link Properties** dialog.



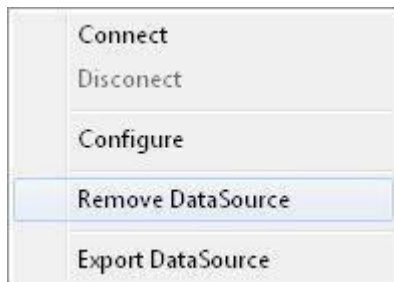
3. Make the required changes and click **OK**.

To Remove a Datasource:

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Select a datasource in the **DataSources** tree.

Right-click on the datasource and select **Remove DataSource**.



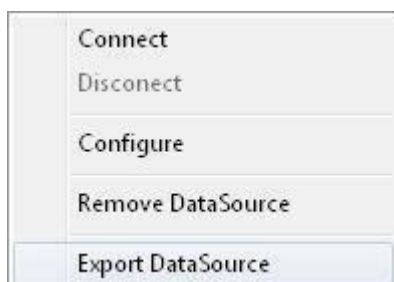
You will be prompted with a warning dialog.

If you are clear to proceed click **Yes**.

To Export a Datasource:

Select a datasource in the **DataSources** tree.

Right-click on the datasource and select **Export DataSource**.



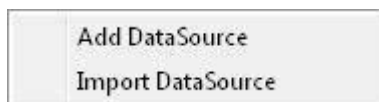
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3. This will open Save As dialog.

4. Select a location and name for the datasource, and then click **Save**.

To Import a Datasource:

Right click on the **DataSources** icon and click **Import DataSource**.



This will start the **Open** dialog.

Note: Connect is only available when a datasource is disconnected. Disconnect is only available when a datasource is connected.

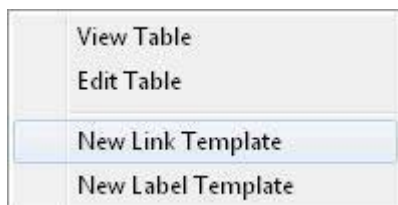
Creating Data Link Templates

In order to connect data from tables to graphic objects it is necessary to create Link Templates. Link templates set a pattern that defines which the data elements will be linked from the table and associated with designated graphic objects.

To Create a New Link Template:

1. Select a table in the **DataSources** tree.

Right-click and select **New Link Template**, or click the . The **New Link Template** dialog will appear.



3. Type in a name for the link template, and then click **OK**.

The **Link Template Edit** dialog will appear.

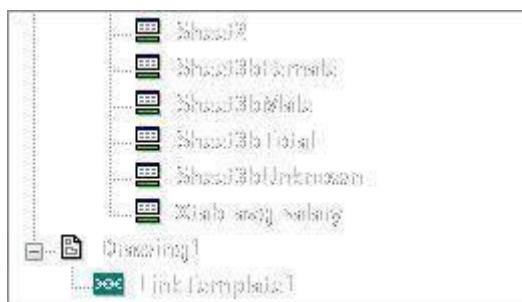
Select the **Key Columns**. These are used to define link information data for linked graphics.

Select the **Value Columns**. These values will be set as custom properties of the object.

The created link template will be attached to current active drawing.

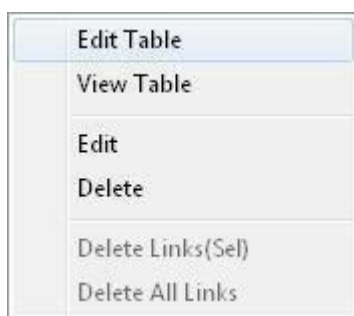
To Edit a Link Template:

1. Select the template in the **DataSources** tree.



2. Right-click and select **Edit**. The **Link Template Edit** dialog will appear.

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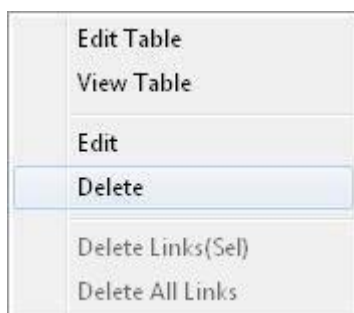
3. Make the required changes, and then click **OK**.

To Remove a Link Template:

Select the template in the **DataSources** tree.

Right-click and select **Delete**.

Warning: The link template will be immediately deleted. Removing a link template leads to removing all graphic links bound to it.



3. You will be prompted to confirm the deletion.

Linking Data to Graphics

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Data is linked to graphics by connecting a table to specific graphics via link templates.

To Connect Data to Graphics:

1. In a drawing, select one or more graphics.

In the **DataSources** tree, double click on the desired table to select it for linking. This will also open the view of the table in the lower frame of the palette.

3. In the table view, in the lower frame of the palette, click on the record you wish link.

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Use the **Link Template** drop-down box located on the palette toolbar to select the desired link template.

Click the **Create Link** button on palette toolbar.

The link between the selected graphics and the table record is created.

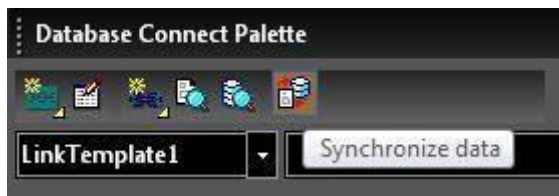
Go to the Custom Properties of the object to verify the link.

Synchronizing Data and Graphics

To get data into linked graphics it is necessary to synchronize the data. The synchronizing process copies data from the datasource into all linked graphics and places it in the custom properties of linked graphics.

To Synchronize Data:

1. Click the **Synchronize Data** button on toolbar to run synchronizing process.



Verify that the data has been synchronized by opening the properties of a graphic that is linked and going to the **Custom** page.

To Remove Links from Selected Graphics:

Select the graphic/s you wish to de-link.

In the **DataSources** tree select the link template you wish to de-link from the graphic/s.

3. Right-click and select **Delete Links(Sel)**.

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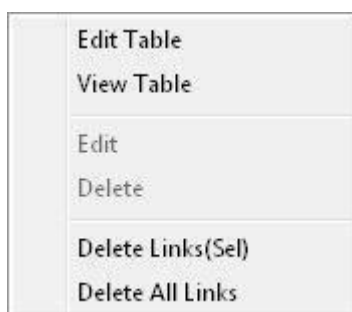
Synchronize the data.

The links and data have been removed.

To Remove All Links to a Link Template:

1. In the **DataSources** tree select the link template you wish to de-link.

2. Right-click and select **Delete All Links**.



3. Synchronize the data.

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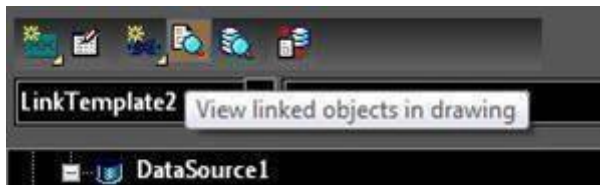
4. The links and data have been removed.

To Find Graphics Linked to the Datasource:

Select the datasource.

Click the **View linked objects in drawing** button on palette's toolbar.

Right-click and select **Find Graphics**.



To Find Table Record Linked to a Selected Graphics:

Select a graphic.

Click the **View linked records in dataview** button on palette's toolbar.



All records which are linked to the selected object will be highlighted in the table view.

Viewing and Editing Data in Tables

When connected a datasource shows all available tables and views in database. Available tables and views appear as child nodes of the datasource.

To Show a Table's Content:

Select the table in the **DataSources** tree.

Right-click and select **View Table**, the table will be opened in the lower frame of the **Data base Connection** palette in Read-Only mode.

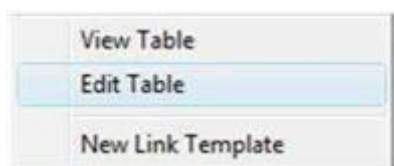


To Edit a Table's Content:

Select the table in the **DataSources** tree.

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Right-click and select **Edit Table**, the table will be opened in the lower frame of the **Data Base Connection** palette in Edit mode.



13.3. Tables

(Available in all TurboCAD Variants)

Default UI Menu: Draw/Table

Ribbon UI Menu:

Insert Table enables you to insert an empty table, and **Modify Table** enables you to add or edit text in the table.

Insert Table is available on the **Draw** menu, and **Modify Table** is available on the **Modify** menu. Both tools are also available on the **Text** toolbar. You can display the **Text** toolbar by right-clicking on any toolbar area and selecting **Text**.



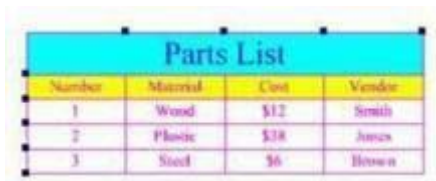
Changing Rows and Columns, Merging Cells

Other than adding text or changing individual cell properties, which are done using the **Modify Text** tool, table changes are made with the **Edit Tool**.

Note: For details on the **Edit Tool**, see [Edit Tool](#)

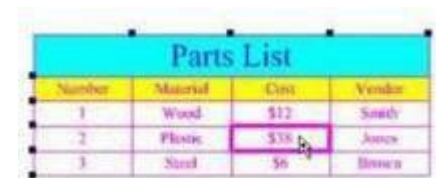
To start editing, activate the **Edit Tool** and click the table. You can move any of the nodes to change sizes of single rows or columns.

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Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

To add a row or column, press Shift and click the cell to the left or above where the new item will go.



Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

Note: If you want to Shift-select a new cell, you must first use Shift and click to de-select the current cell.

Insert Row and **Insert Column** are available on the local menu or Inspector Bar.



The new row is added below the selected cell.



Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

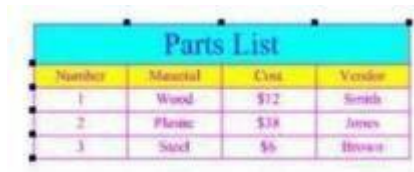
4. To remove a row or column, Shift-select a cell in that row or column.



Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

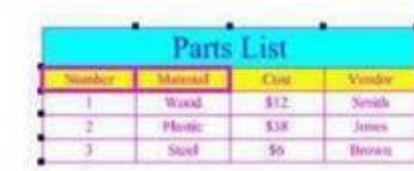
The row is removed.

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Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

To merge cells, Shift-select each cell you want to merge.

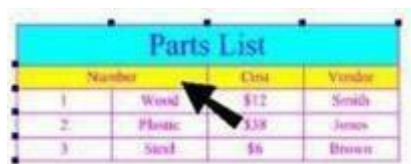


Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

Select **Merge Cells** in the local menu or Inspector Bar.



The selected cells are now one cell.



Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

To separate them again, Shift-select the cell and select **Unmerge**.

Local Menu & Keyboard

The local menu provides additional controls and lists the complementary keyboard commands for those functions. You can change and expand or contract the cells which

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are selected. most importantly you can opt to edit the cells text or block.

Insert Table

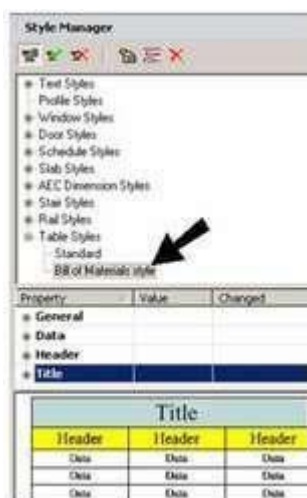


Default UI Menu: Draw/Table/Table

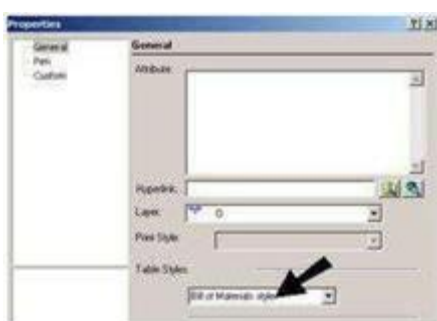
Ribbon UI Menu:

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Before creating a table, you can define a table style (see [Table Styles](#)). Styles are useful for creating tables with specific text properties, cell colors, and header / title rows.



To apply the style to the table, open the tool's **Properties** and select the style on the **General** page.



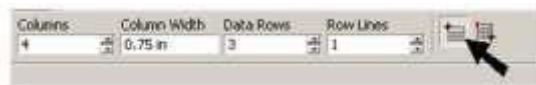
You can also change an existing table's style in its **Properties**, or in the Selection Info palette. There are two methods of inserting a table: **Specify Insert Point** and **Specify Window**.

Specify Insert Point

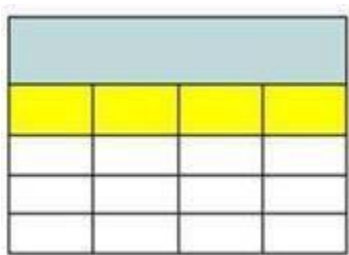
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With this method, you click the top left point of the table.

Make sure **Specify Insert Point** is selected in the local menu or Inspector Bar. The number of columns and rows, as well as column width and row lines (number of lines of text) is set in the Inspector Bar.



Click once in the file, and the table is inserted.



If you don't want to insert another table, press Esc or start a new tool.

If you need to change the size of an individual row or column, use the **Edit Tool**.

See [Changing Rows and Columns, Merging Cells](#)

Specify Window

With this method, you insert the table by clicking two corner points. You can either fix the cell dimensions, or you can fix the number of rows / columns. (You can also mix options for sizing and numbers for rows and columns.)

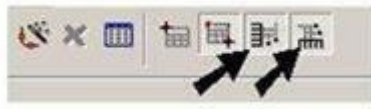
Make sure **Specify Window** is selected in the local menu or Inspector Bar.



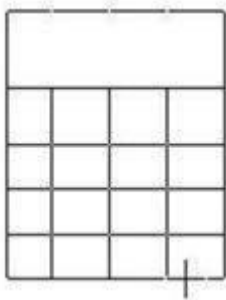
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Fixed Number of Rows - Columns

Activate both Calculated Row Height and Calculated Column width.



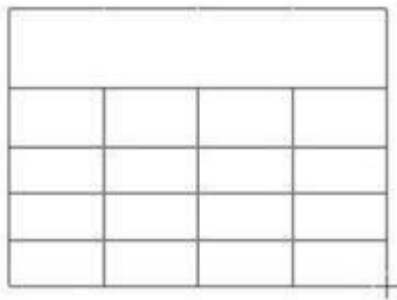
Click once to set the top left corner of the table.



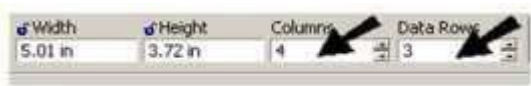
Set the number of columns and rows.



Move the mouse to size the table.

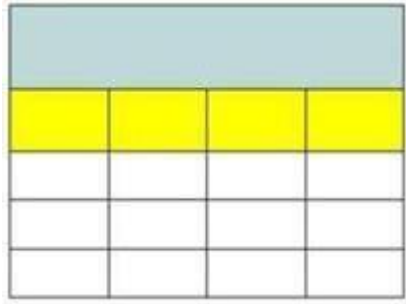


The number of cells remains the same, no matter how the table is sized.



Click the second corner to insert the table. If you specified a table style, it will be applied.

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If you need to change the size of an individual row or column, use the **Edit Tool**.

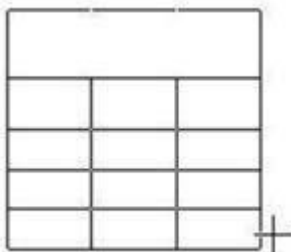
See [Changing Rows and Columns, Merging Cells](#)

Fixed Cell Size

Turn off both Calculated Row Height and Calculated Column width.



Click once to set the top left corner of the table.

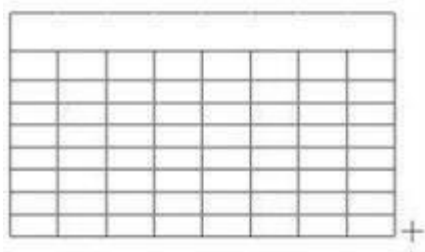


Set the column width and number of text lines per row.



Move the mouse to size the table.

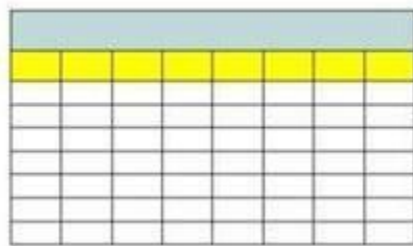
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The cell size remains the same, no matter how the table is sized. Cells are added or removed as needed.



Click the second corner to insert the table. If you specified a table style, it will be applied.



If you need to change the size of an individual row or column, use the **Edit Tool**.
See [Changing Rows and Columns, Merging Cells](#)

Modify Table

Default UI Menu: Draw/Table/Modify Table

Ribbon UI Menu:

Modify Table is used to add text to cells, or to edit existing cell text. It can also be used to change properties of individual cells.

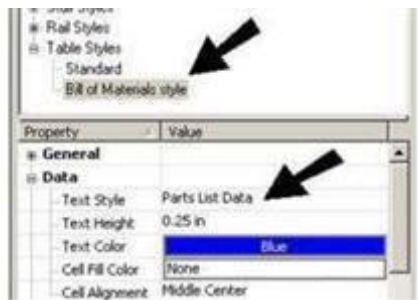
Adding or Editing Cell Text

If you are using styles, the table style will refer to a text style for each type of text (data, header, and title). So unless you want to use standard text, define text styles first.

Note: For details on text styles, see [Text Styles](#)

Open the table style you want to use, and specify the text style you want to use for **Data** text. You can also set text color and height here.

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There are also categories for **Header** and **Title**, which can each have their own text style.



To add text to the table, activate **Modify Table**.

Make sure **Edit Cell Text** is active in the local menu or Inspector Bar.



Select the table you want to modify, then click in a cell where you want to place text.



Click in each cell and select **Edit Text** from the Inspector bar or Local menu. Then type the text. If your drawing has blocks use **Insert block** you can also opt to insert a block into the cell.

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Parts List			
Number	Material	Cost	Vendor
1	Plastic	\$12	Smith
2	Wood	\$19	Jones
3	Steel	\$3	Brown

If you want to remove text from a cell, click it and select **Clear Cell Content** from the local menu or Inspector Bar.



When the table text is complete, end **Modify Text**. The properties of the cell text will match their text styles.

Parts List			
Number	Material	Cost	Vendor
1	Plastic	\$12	Smith
2	Wood	\$19	Jones
3	Steel	\$3	Brown

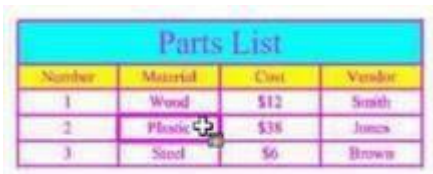
Changing Cell Properties

If you want to use **Modify Text** to change properties of an individual cell, make sure **Edit Cell Text** is not active.



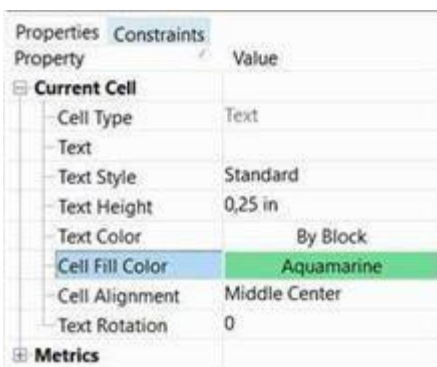
Click the cell you want to change.

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Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

The properties of the cell can be changed in the Selection Info palette. In this example, the cell's fill color was changed.



Properties		Constraints
Property		Value
Current Cell		
Cell Type		Text
Text		
Text Style		Standard
Text Height		0,25 in
Text Color		By Block
Cell Fill Color		Aquamarine
Cell Alignment		Middle Center
Text Rotation		0
Metrics		

4. When finished, the cell has the new properties.



Number	Material	Cost	Vendor
1	Wood	\$12	Smith
2	Plastic	\$38	Jones
3	Steel	\$6	Brown

Text Rotation:

TurboCAD table now supports the "Text Rotation" property. Now table cell has editable property - Text Rotation.

Also, cell editing has been improved. Now the cell selection is not lost after changing the properties of the cell using the "Selection Info Palette".

Demo Video:

<https://youtu.be/XJ76UKqiSI0>

Table Export (*.xlsx)

Default UI Menu: Draw/Table/Table Export

Ribbon UI Menu:

This tool allows you to save a table as an Excel (*.xlsx) file.

To save a XLSX file:

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Select the tool.

Click on a table.

3. Type in a **File name**.

Click the **Save** button.

Press the **Space bar** to finish.

Table Import (*.xlsx)

Default UI Menu: Draw/Table/Table Import

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Ribbon UI Menu:

This tool enables you to create a table based on a sheet from an Excel (*.xlsx) File.

To Create a Table:

Start the tool and the dialog appears.

Click the down arrow in the **Choose an Excel Data Link** dropdown.

Click on **Create New Data Link**.

Type in a name and click OK.

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Click

the down arrow in the **Choose an Excel file** dropdown. Click
on **Browse for a new Excel (*.xlsx) file**.

Select a file.

Click Open.

Click the down arrow in the **Select Excel sheet to link to** dropdown.

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Select a sheet.

Click OK.

Position the cursor and click to insert the table.

The dialog will reopen.

Change the parameters and insert another sheet, or click Cancel to terminate the operation.

To Delete a Data Link

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Click the down arrow in the **Choose an Excel Data Link** dropdown.

Select an existing link.

Click the Delete button.

13.4. TurboCAD Coordinates Export Palette

(Available in Platinum and Professional)

Default UI Menu: Tools/Palettes/TurboCAD Coordinates Export Palette

Ribbon UI Menu:

This palette tool provides the ability to export the point data from your drawing.

TC Coordinates Options

Before exporting your point data it is best practice to configure the data output. Click the Options button to open the Options dialogue. Once open you can specify what data to export and the format of the data.

The basic options allow you to set which types of points you wish used in the export. These include: Points:

- Points (including 3D Points)

Lines:

- End points of lines

- MidPoints of Line Segments

Polylines:

- End points of polylines (including 3D Polylines)

- Vertices of Polylines (excluding endpoints)

- Midpoints of Polyline Segments (including 3D Polylines)

Circles:

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Center points of circles

Quadrant points of circles

Arcs:

End points of arcs

Center points of arcs

Included points of arcs

Midpoints of arcs

Quadrant points of arcs (only those on the drawn segment of the arc)

Curves:

End points of curves (including 3D curves)

Control points of curves (exclude end points)

TC Coordinates Advanced Options

Clicking the Advanced button in the TC Coordinates Options dialogue opens the Advanced Options dialogue which allows you to configure how points are to be

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analyzed and how they are to be exported. These options include:

Point CS mode: specifies what coordinate system will be used to determine the point location. There are four options, World is the default.

World

User (Current drawing workplane)

View

Entity

Object processing mode:

Use explode to export coordinate of non-default objects: specifies whether blocks, groups and other compound object should be exploded (internally) to compute the point data within them.

Output result mode:

Add point's type: Appends the points object type to the location data

Add point's CS mode: Appends the points Coordinate System info to the location data.

Automatically save to file: Prompts to save the data to file every time you click the Export button.

Automatically copy to clipboard: Automatically adds the data to the system clipboard when you click the Export button.

Output result format:

CSV Delimiters symbol: Specifies the data divider (delimiter) to be used in output. There are three options.

Semicolon

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Tab

Comma

End of Line symbol(s): Specifies the symbol that will be used to designate the end of line in the data file. There are three options.

CR+LF

CR (Carriage Return)

LF (Line Feed)

Using TC Coordinates

To use TC Coordinates simply click the Export button.

The data area of the TC Coordinates palette will be filled with the relevant data.

Depending upon the settings in Advanced options, the system clipboard will be filled with the data and/or the Save As dialog will open so that you can save the file.

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Note: It is important to delete the contents of the data area of the TC Coordinates palette after each use, or before subsequent use. The palette will not automatically delete data and will simply add new data to previous data each time you click the Export button.

To delete the content of the TC Coordinate Palette:

Click in the palette.

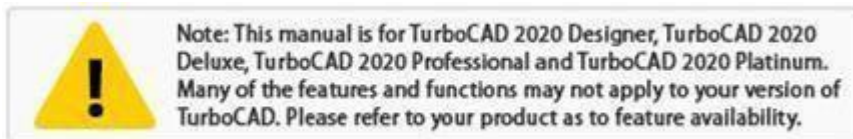
Press CTRL+A keys to select everything in the palette.

Press the Delete key.

Groups, Blocks, and the Library

(Available in all TurboCAD Variants)

Groups, blocks, and symbols are features that enable you to combine and store multiple objects for easy access, and for additional and future use.



14.1. Auto-Naming of Groups, Blocks, Symbols

By default, each new group, block, or symbol is assigned a name automatically. These names are numbered, and the number increases by one whenever you add a new group, block, or symbol to the drawing, ensuring that each has a unique name. By default, the generated names are "Group 1", "Group 2", "Block 1", "Block 2", "Symbol 1", "Symbol 2," and so forth.

You can change the default naming settings in the **Auto-Naming** page of the **Program Setup**. See [Auto-Naming](#)

The "@" symbol is a placeholder where the actual number will be inserted. For example, if you type "Part @" in the **Group name prefix** field, the first group you create will automatically be named "Part 1", and the next group will be named "Part 2". If the **Group name prefix** is "#@ gear," the groups will be named "#1 gear," "#2 gear," etc.

Note: The first '@' that you include in the name is a placeholder for the automatic number. Any '@' characters that occur after the first one will be part of the actual name.

The name of a group or symbol (not a block) is listed on the **General** page of the **Properties** window, in the **Attribute** field. You can use this field to change the name (or path, for a symbol).

14.2. Blocks

(Available in all TurboCAD Variants)

One or more objects can be combined and stored as a block. A block is treated as a single object for purposes of selecting and editing.

Each block is stored in the drawing's internal library, and each instance of the block is a reference to this source. This means that numerous instances of a block can be added to the model without significantly increasing the file size. Groups are similar, but they are not linked to sources; each group contains its own drawing data. See [Groups](#)

Note: A drawing's block library is internal to the drawing, and is stored with the file. Symbol libraries are similar but are stored separately, and can be accessed while in any drawing. If you need to create a group of objects that will be used in multiple drawings, create a symbol. See [Loading Symbol Folders into the Library](#). If you want to import the entire contents of another file (TurboCAD or other format) as a block, see [External References](#)

Because blocks can contain individual objects, groups, and other blocks, they can be complex hierarchical structures. For block manipulation, use the Blocks Palette (**View / Blocks**).

Tip: You can use the TC Explorer Palette to view blocks of any open drawing, and to drag blocks to and from drawings. See [Drawing Settings](#).

You will find details on using block and the Blocks palette on the following pages:

Additional Block Controls



Show Selected: Toggles the Show Selection option. When on the result will be that anytime a single block is selected into the drawing space, that block will be selected and highlighted in the Blocks palette.

Block name prefix: If names are automatically generated, you can enter a string that appears before the item name. The "@" character is a placeholder for the automatic number.

Prompt for name: You will receive a prompt each time a new item is created.

Generate block names: Names will be automatically assigned.

Insert blocks when creating: Each block will be inserted into the drawing once it is created.

Compensate for the offset of the base point at the block references: Prevents updating of the reference points for inserted blocks, when a the block reference point is relocated.

[Editing a Block](#)

Show Selection any time an inserted block is selected: Toggles the Show Selection option. When on the result will be that anytime a single block is selected int the drawing space, that block will be selected and highlighted in the Blocks palette.

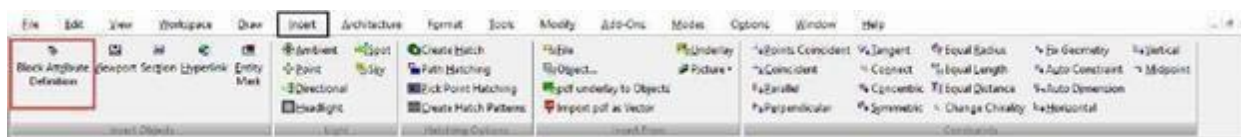
Block Attributes



Default UI Menu: Draw/Block/Block Attributes

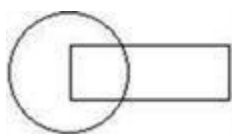
Ribbon UI Menu:

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A block attribute is AutoCAD- informational text associated with a block, that you can enter whenever you insert a block. TurboCAD reads and displays block attributes from AutoCAD drawings (DWG) and DXF files.

1. Create the objects that comprise the blocks.



(You can also add a block attribute after a block has been created, in **Edit** mode. This is done the same way as adding another geometric object. See [Editing a Block](#).)

Select **Block Attribute Definition**. Select the start point for the text, preferably on or near the block objects.

Type the "tag" name for the block attribute, such as "COST." This name is used to uniquely identify the attribute within the block, since more than one attribute can be created. If the drawing will be sent to AutoCAD, do not use spaces (use underscores instead).



Note: This tool works like the text tool, in terms of alignment and local menu options.

See [Inserting Text](#)

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Enter the prompt and default value in the Inspector Bar, or you can enter these properties later. For example, the **Prompt** can be "How much does it cost?" and **Default** can be \$0.00.



Press Enter to finish the definition. You can create multiple attributes, such as Part Number, Owner, etc.

Once the block attribute is created, you can open its **Properties** to add or change the **Tag**, **Prompt** and **Default** values, as well as **Mode**.

Mode: Affects how the dialog appears when the block is inserted:

Invisible: The text of the block attribute is not displayed in the drawing. You can see the information in the Selection Info Palette or edit it by displaying the block's properties.

Warning: When you explode a block containing invisible block attributes, the information is lost.

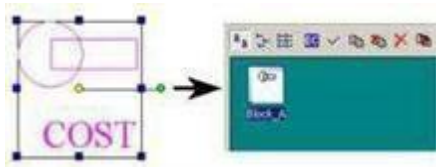
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Constant: The attribute value is fixed and unchangeable. It is shown to you during block insertion. Exploding the block will turn the block attribute into text that can then be edited.

Verify: Prompts you to verify that the attribute value is correct when you insert the block. In AutoCAD, the attribute value is shown for your verification during block insertion if the AutoCAD variable ATTDIA is set to 0. This flag has no effect in TurboCAD as it takes place in AutoCAD when the variable ATTDIA is set to 1.

Preset: Sets the attribute to its default value when you insert a block containing a preset attribute. In AutoCAD, you are not prompted for a value if the AutoCAD variable ATTDIA is set to 0. This flag has no effect in TurboCAD as it takes place in AutoCAD when the variable ATTDIA is set to 1.

When the block attributes are defined, simply include them in the selection of objects that will make up the new block.

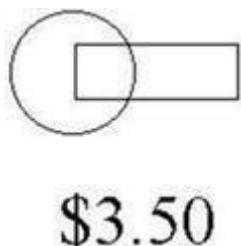


Setting Block Attributes

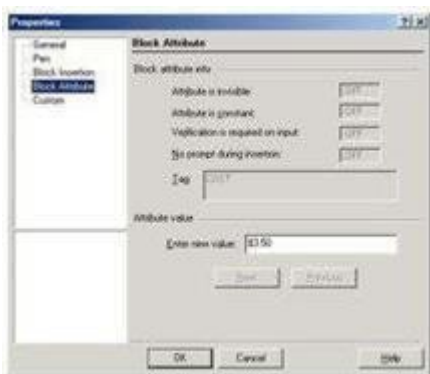
When you insert a block that has one or more attributes, a window will appear that gives the prompt and offers the default value. For example, the prompt says "What does it cost?" the default value is \$0.00. The value can be changed to another value, in this case, \$3.50.

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For multiple attributes, use the **Next** and **Previous** buttons to set values for all of them. When the block is inserted, the value is included with the objects.



When a block has attributes defined, its Properties will contain an additional page - **Block Attribute**. You can edit each attribute value in this window, using the **Next** and **Previous** buttons to scroll through multiple attributes.



Sync Attributes

Sometimes it is necessary to add attributes to blocks after many instances of the blocks have already been inserted into the drawing. In this case the new attributes are not automatically added to the prior insertions. The Sync Attributes button will add the new attributes to the older blocks.



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The values for these newly synced attributes within the older blocks will be the default value you assign to the attribute when you create it and it will blank if no value is assigned. For blank attributes you will have to select the existing blocks and assign values to those attributes.

Extracting Block Attributes

If you attach attributes to blocks, you can extract these attributes and display them in a table. You can also export the attributes to a report in an external file.

Note: If you want to create a schedule for architectural elements in your file (walls, slabs, windows, or doors), without defining block attributes, see [Schedule Styles](#)

This example has three blocks used to mark windows, doors, and slabs.

Here are the three blocks in the Blocks Palette.

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Select **Extract Attributes**. In this window you can select the blocks and attributes that will be included in the schedule or report.

Note: You can re-order a column by dragging its header to the new location.

Scan Entire Drawing: Attributes will be extracted from all paper spaces and model space.

Scan Model Space: Attributes will only be extracted from model space.

Scan Current Space: Attributes will be extracted from the current model space or paper space.

Scan Selected Entities: Attributes will be extracted only from currently selected objects.

Scan Groups: If any groups contain blocks, these blocks will be scanned for attributes.

Scan Nested Block: If blocks contain nested blocks, these nested blocks will be scanned for attributes.

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Include Xrefs: The content of Xrefs will also be scanned. The **Blocks** list contains all blocks that have attribute definitions. The **Properties** list all attributes found for the blocks checked in the **Blocks** list.

Show Summary List: The **Properties** list contains all attributes for all blocks checked in the **Blocks** list.

Show Selected Block Properties: The **Properties** list contains attributes only for the block that is currently checked in the **Blocks** list.

Show Visible Properties Only: If selected only attributes that are visible will be shown.

You can select attributes for each block that will be included. For example, click **Show Selected Block Properties** at the bottom, and select the "Door Mark" block. Check only the "COST" and "TYPE" attributes.



Select the "Room" block and check "AREA," "COST," and "TYPE."

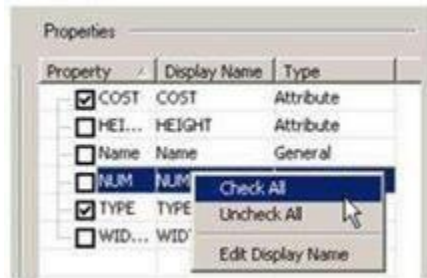


Select the "Window Mark" block and check "COST" and "TYPE."

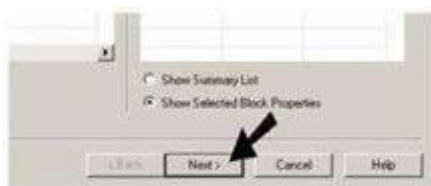


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You can right-click on any field under **Blocks** or **Properties** to get a popup menu in which you can check or uncheck all, or change the display name.



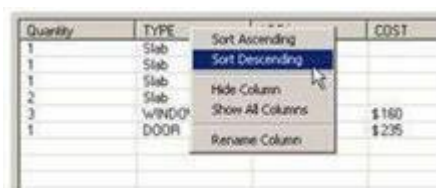
5. When the blocks and properties are defined, click **Next**.



TurboCAD scans the file, and the **Preview** window displays the results.

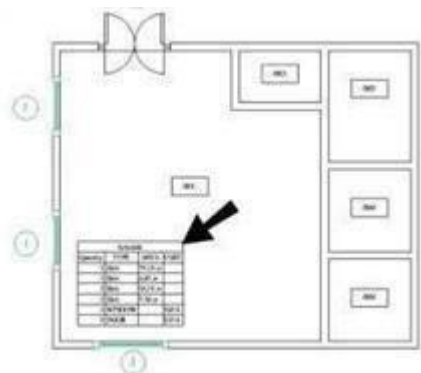


If **TurboCAD Table** is checked, the report will be inserted into the file. If you want to export the results, click **External File**. You can click on any column header to change the sorting order, or hide or rename a column.



Click **Finish**. If the table is to be inserted into TurboCAD, you will see the **Insert Table** window. Here you can define the column and row sizes.

7. Click OK, and then click where you want to place the table.



You can make changes to the table formatting in the Selection Info palette.

Creating a Block



Default UI Menu: Draw/Block/Create Block

Ribbon UI Menu:

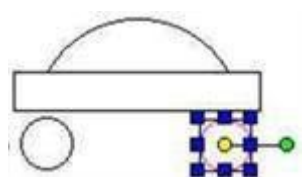


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Combines all selected objects into a block. Objects can include drawing objects, images, OLE objects, groups, and other blocks. Blocks can be comprised of 2D and 3D objects.

Warning: You should move all objects to be used in a block to Layer 0, before creating the block. Many features will not work properly otherwise.

Create the objects you want to combine into a block. Each object is separate, as you can see in **Select** mode.

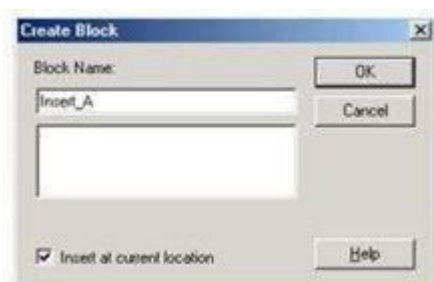


You can set the reference point of the objects before creating the block (see [Components of Select Edit Mode](#)), or you can change it later.

Select the objects you want to include in the block.

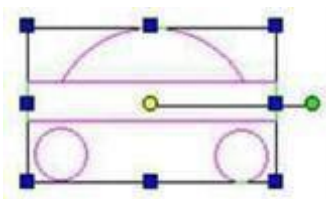
Click **Create Block**.

If auto-naming is not used, you must assign the block name. (See [Auto-Naming of Groups, Blocks, Symbols](#).) Check **Insert at current location** to create the block in place. Otherwise, the block will be removed from the drawing and stored in the library for future insertion.

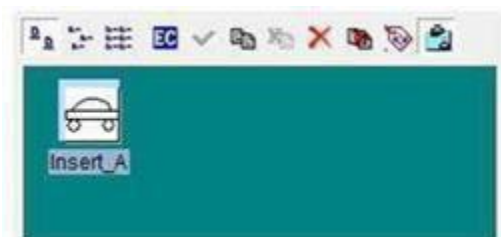


The block is created. If the block is inserted in the drawing, you can select it as one object.

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Blocks are created on Layer 0, even if their components are on other layers. To insert the block, see [Inserting a Block](#). If you open the Blocks Palette, you can see each block you have created. You can also add blocks to the drawing, edit blocks and create new blocks using this palette.

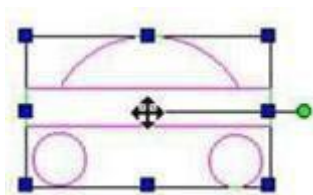


Warning: Do not include lights in a block. If you do, the lights will remain in their original locations regardless of where you place the block in your drawing.

Creating a Block by Drag and Drop

To create a block this way, the Blocks Palette must be open. Select **Blocks palette**, or the Blocks Palette tab.

Select the objects that you want to combine into a block. Click and hold the reference point (the yellow circle). The cursor turns into a double arrow.



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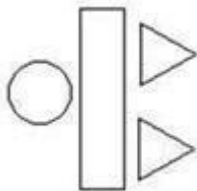
Drag the objects into the Blocks Palette. If auto-naming is not used, assign a block name.

The block appears in the Blocks Palette.



Editing a Block

(Changing a block's contents) You can use any editing or node editing tool to modify, move, copy, add, etc. objects that comprise a block. This example use the block shown below, consisting of four objects. The block name is "Logo."

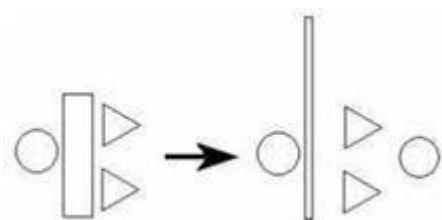


1. In the Blocks Palette, select the block, and click **Edit Content**.



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In edit mode, the screen contains only the block components, which are available for editing. You can add, delete, or edit objects. Whatever appears on the screen will become part of the edited block.



When finished, right click and select **Finish block/group editing**, or click **Finish Edit Content** in the Blocks Palette.



The icon for the block is updated to reflect the edited components. Any instances of the block already in the drawing are updated, including resized and scaled blocks.



when a block is edited, an editing mode indicator bar is displayed.



When block editing is done, click "Finish to Edit Block" in the indicator bar.

Relocating a Block Reference Point

Hotkey: Ctrl+Shift+R

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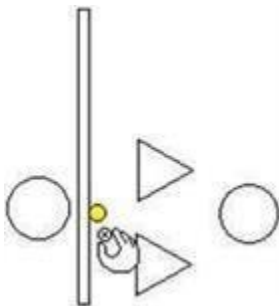


A block is inserted by placing its reference point. By default, the reference point is at the center of extents of the entire block, but this location can be changed.

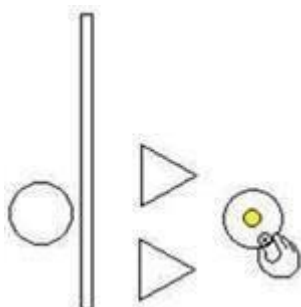
1. In the Blocks Palette, select the block, and click **Edit Content**.



Activate **Relocate Block Ref Point**. The default reference point appears, as a yellow circle.



3. Click to specify the new location for the reference point.

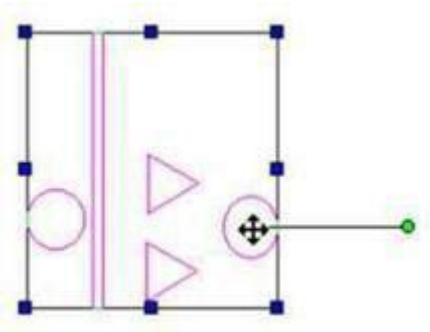


Select **Finish to Edit Block**, or click **Finish Edit Content** in the Blocks Palette.

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Now when you insert the block by dragging, you will drag it by the new reference point.



Note: For any instance of a block you can move its reference point and rotation bar. (See [Components of Select Edit Mode](#)) However, this change only affects the block itself and any copies of it. New instances of the block will use the block's defined reference point.

Changing a Block Name and Description

To change a block name or add a description, double-click the block icon in the Blocks Palette.



Make the necessary changes in the **Edit Block** window.

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After the changes are made, any description you add will be displayed as a tool tip on the block icon.



Copying a Block

This section is on copying a block to create a new block. To create copies of the same block in the drawing, you can use any of the standard editing tools. See [Copying Objects in Select Edit](#) or [Copying Objects](#)

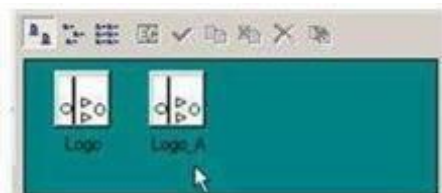
1. To copy a block, select it in the Blocks Palette and click **Copy**.



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Depending on the **Auto-Naming** settings, a copy will be created with a generated name, or you will be prompted for a name. See [Auto-Naming of Groups, Blocks, Symbols](#)

The new block appears in the Blocks Palette, and can be edited like any other block. See [Editing a Block](#).



Deleting a Block



Press Delete a Block button.

Remove block with all references: Deletes the block and all instances. Be careful when using this option, because it cannot be undone.

Replacing Blocks

One convenient feature of blocks is that they can easily be replaced by other blocks. You can replace all blocks in the drawing, or just selected blocks.

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To replace instances of a block, select the block in the Blocks Palette and click **Replace References**.



In the **Replace Block Reference** window, select the replacement block. You can also choose to replace all instances in the entire drawing, or just those in the active space (visible).

Click OK to implement the change.

To replace an instance of a single block, open its **Properties** to the **Block Insertion** page. Use the **Replace with** button to replace it with another block.

Exploding a Block

Click the **Explode** icon.



If you explode a block that contains nested groups or blocks, the nested groups will remain intact. Each sub-block must be exploded separately. See [Exploding Objects](#)

External References

Default UI Menu: Insert/Create External Reference

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Ribbon UI Menu:



An external reference (xref) is a kind of a block in that it is stored in the current drawing's block library. However, unlike a block, the objects associated with an xref definition are not stored in the current drawing; they are stored in another drawing file. When you create an xref, the entire contents of this other file are imported as a block. See [Blocks](#)

Note: You can also access An external reference via Block palette.

Xrefs are usually used to display the geometry of a common base drawing in the current drawing, such as a frame. They can be taken from files any formats readable by TurboCAD. Only files that have objects in Model Space can be added as Xref's.

To import another drawing as a block (xref), select **Insert / Create External Reference**.

In the **Ext ernal Reference File Location** window, select a file type and locate the desired file.

The selected file is added to the block library of the current file. You can view it and insert it using the Blocks Palette. However, you cannot edit an xref in the Blocks Palette - you must change the original file.

Note: If you edit the original file from which the xref was created, the block in the current library will not change. You will need to recreate the xref.

External References Panel

If you click on the **Ext ernal References** button at the top of the Block palette you will see the **External References** panel at the bottom of the palette.



When xref's are nested in other drawings they are shown in a tree format listed below that referenced drawing. Right-clicking on any of the xrefs listed in the external References panel will open the following local menu:



Open: will open the xref in TurboCAD.

Reload: will reload the reference file, including any updates.

Detach: will detach the referenced file, and any insertions of that file in the drawing will be deleted. You cannot detach nested xrefs. You must open the file to which they are attached to remove them.

Bind: embeds the selected XREF as a Block in the drawing. All attachment to the external drawing is lost.

VISRETAIN: Through the Design Director it is possible to edit the various properties of the layers within an XREF. These changes do not affect the original drawing. Even if the external referenced drawing is altered these layer changes will be retained. However, if the XREF is reloaded the changes will be lost. It is possible to disable this feature by changing the \$VIZRETAIN variable through the DCExplorer Palette.

Exploding XREFs: When an XREF has been bound to a drawing it becomes a block.

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Instances of that block in the drawing can be exploded so that you can edit the geometry directly in the drawing.

Automatic Updating of Xrefs

When using a file with Xrefs, if any of those external files are updated, TurboCAD will prompt you if you want to regenerate the file with the updated Xref(s).

Editing Xrefs Block Properties:

Editing Xref Blocks is similar to editing other blocks. The only difference is that Xref blocks have an option "Status", with two values: "Loaded" and "Unloaded" has been added to the Edit Xrefs dialog box. When a referenced drawing (Xref) is Unloaded from the current drawing, the drawing opens much faster and uses less memory. When the Xref definition is unloaded from the drawing file, the internal pointer to the referenced drawing remains. You can therefore restore all the information by Loading the Xref. In order to maximize the performance of TurboCAD, it is recommended that you Unload any referenced files if they are not needed in the current drawing session but may be used later for printing/plotting.

In Place Editing of Groups and Blocks

You can edit groups or blocks in place within the drawing.

1. Select the block or group.

Right click to open the local menu and select Edit tool. All other elements in the drawing will fade-out, while the selected entity remains clear.

Proceed by making your desired changes to the object. This can include changing properties, moving geometry, adding geometry, editing geometry, and deleting geometry.

If you are editing a block, other instances of the block will show the changes you are making simultaneously.

To finish, right click and select **Finish block/group editing**, from the local menu, or **Finish Edit Content** from the Block Palette toolbar, or **Finish to Edit Block/Finish to Edit Group** from the **Tools** menu.

Block attributes that are edited, added or deleted will not be updated in exiting block insertions, including the one that you selected. Only the "Original" block in the palette will reflect attribute changes.

As of TurboCAD latest version, you can now snap to objects outside the block or group while in block/group mode.

Inserting a Block

To insert a block into the drawing, simply drag it out of the Blocks Palette and drop it into your drawing. The inserted block will still be selected after you place it, so that you can move, scale, or rotate it. See [Select Edit](#)

Blocks are placed on Layer 0, even if their components are on other layers. Layer 0 should always be left visible, or blocks will instantly "disappear."

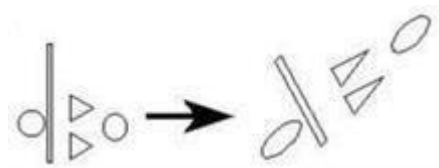
Tip: You can use the TC Explorer Palette to drag blocks to and from drawings. See [Drawing Settings](#).

The Selection Info palette shows the Block Insertion structure. This feature does not allow the editing, but only visualizing the structure.

Block Insertion Properties

These properties can be used if you want to change any aspect of the block instance - its location, scale, angle, or the block reference itself. For any block, open the **Properties** window and open the **Block Insertion** page.

For example, a block was inserted, then moved, rotated, and resized.



Its **Block Insertion** page contains the current values for **Position**, **Rotation**, and **Scale**.

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You can change the values in this window, or use the **Select Edit** tools and see the updated values in these fields. To replace a selected block with another block, select the replacement block from the list and click **Replace with**. Click OK to implement the change.

Inserting Blocks into Another File or Application

You can also use the drag-and-drop technique to insert blocks into another open file. Dragging a block into another drawing accomplishes two things: it inserts the block into the target drawing, and it places the block into the library of the target document. The target drawing must be open and its window must be visible on the screen. (Use **Window / Tile** to see all open windows.) After you drag the block, the target file becomes the active window.

Drag-and-drop can also be used to place blocks, symbols, or any selected objects into other Windows applications, such as Microsoft Word or graphics programs.

Note: You can also use **File / Extract To** to export all blocks into another file.

Inserting Blocks from Another File

The **Insert / File** tool can be used to insert some or all blocks from another file into the current drawing (see [Inserting a File](#)). If both drawing have blocks with identical names, you can choose whether to ignore or replace them.

Tip: You can also use **File / Extract From** to insert selected components like blocks (or layers or other settings) from another file into your drawing. However, this method will insert **all** blocks, without enabling you to pick and choose.

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Select **Auto Naming** and make sure that **Prompt for Name** is checked for **Blocks**.

Select **Insert / File** and choose a file containing one or more blocks you wish to insert.

Use the **Add Blocks** window to select the blocks to import:

The left panel displays the blocks found in the selected file, and the right panel displays any blocks that exist in the current drawing. Select the mode (**Add**, **Replace**, or **Ignore**) and click the relevant button at the top right (**Add**, **Add / Replace All**, etc.) to generate the blocks. If you want to pick and choose the blocks to add, make sure **Process all additional blocks** is not checked.

Generate name: Assigns a new name to a block you wish to add.

Modes: The options here depend on the selected block, and whether a block with the same name already exists in the current drawing. **Add block(s):** Adds the selected block.

Replace block(s): The blocks from the external file will replace those in the current drawing.

Ignore block(s): Click **Ignore All** and the blocks will not be added.

Options

Process all additional blocks: Adds and/or replaces all blocks found in the source file.

Generate blockname with prefix: Assigns a name automatically, with the specified prefix, to the inserted blocks.

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Using **Insert / File** also adds all drawing objects found in the source file. However, you can press **Undo** (Ctrl+Z) immediately after using **AddBlocks** to clear the imported objects, leaving only the imported blocks. You may have to undo twice, to remove objects both in Model Space and Paper Space. Other source file components like layers, lights, and views will also be inserted, but they can be deleted manually if needed.

Warning: If the source file and current drawing have layers or other components with identical names, the layers will be replaced with those of the inserted file.

There are other ways to import blocks from another drawing, without importing other components:

Open both the source file and new file, and select **Windows / Tile** so that you can see both drawing windows. Use the Blocks Palette to drag blocks from the source file to the new file. This method imports the blocks only.

In the source file, select the blocks you want to export (select in the drawing area, not in the Blocks Palette). Copy the blocks (Ctrl+C or Edit / Copy), and paste them (Ctrl+V) into the destination file. The **AddBlocks** window will appear. This method imports both the blocks and the layers the blocks are on.

14.3. Groups

(Available in all TurboCAD Variants)

Two or more objects can be combined as a group. A group is treated as a single object for purposes of selecting and editing.

Each group contains its own drawing data. This means that making a copy of a group increases the size of your model. If you need to create several groups of numerous or complex objects, you may want to use blocks instead. Blocks are stored in a library, and each instance of the block refers to this external source, without significantly increasing the file size. See [Blocks](#)

Because groups can contain groups and blocks, they can be complex hierarchical structures. The structure of a group can be seen in the Selection Info Palette (**View / Selection Info**), which can also be used to edit a group's contents. See [Selection Info Palette](#)

Creating a Group



Default UI Menu: Format/Create Group

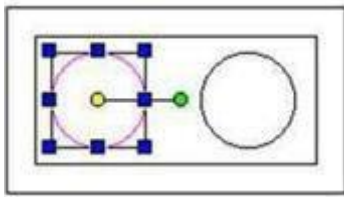
Ribbon UI Menu:



Combines all selected objects into a group. This tool is available on the Groups and Blocks toolbar, or on the icon in the **Drawing Tools**.

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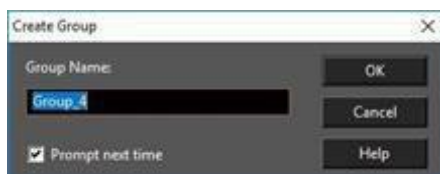
Create the objects you want to combine into a group. Each object is initially separate, as you can see in **Select** mode.



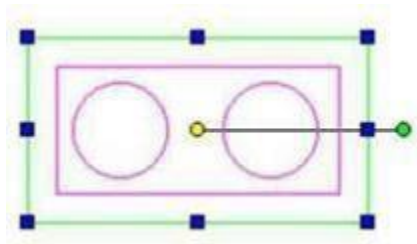
Select all the objects you want to include.

Click Create Group.

If **Auto-naming** is not used, you must assign the group name.



The group is created, and the objects are now selected as one object.

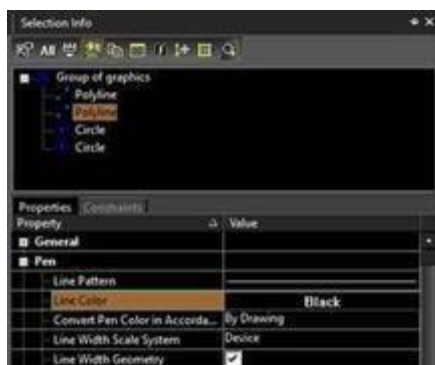


Note: The new group will be placed on Layer 0 if its components are originally on different layers. See [Snap Modes](#). If you explode a group, the objects will return to their original layers. Layer 0 should always be left visible, or newly created groups will instantly "disappear."

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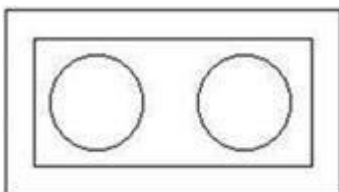
If you open the Selection Info Palette, you can view the contents and properties of a selected group.

You can highlight any component of the group to view its individual properties as well.



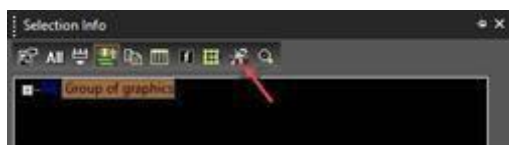
Editing a Group

This example will use the following group, consisting of four objects:

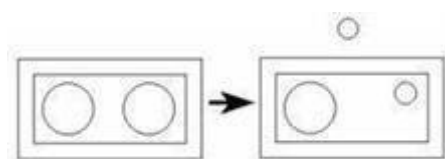


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Select the group, and select **Edit Group Content**. If the Selection Info Palette is open (see [Selection Info Palette](#)), you can also click the **Edit Content** button on the palette toolbar.



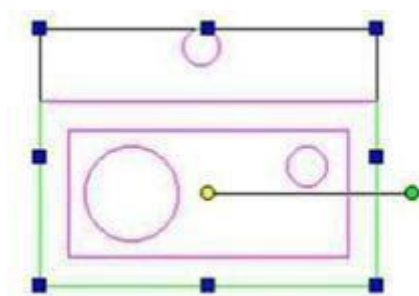
You are now in edit mode, and the screen contains only the group's objects. You can add objects, delete objects, or edit existing objects. Whatever appears on the screen will become part of the edited group.



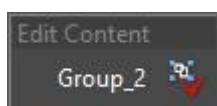
When finished, select **Finish to Edit Group**, or click **Finish Edit Content** in the Selection Info Palette.



The group now contains the edited and new objects.



when a group is edited, an editing mode indicator bar is displayed.



When editing is done, click "Finish to Edit Group" icon.

Exploding a Group

Default UI Menu: Modify/Explode

Ribbon UI Menu:



Click the **Explode** icon.



If you explode a group that contains nested groups or blocks, the nested groups will remain intact. Each sub-group must be exploded separately. See [Exploding Objects](#).

14.4. Library

(Available in All Variants)

Default UI Menu: Tools/Palettes/Library

Ribbon UI Menu:

The TurboCAD library is where you can access symbols and parts. Symbols and parts, like blocks and groups, are objects available for repeated use. But while groups and blocks are internal to a drawing, library items are external files. Generally, each symbol or part is stored as a file, and libraries of files are stored in specified library folders. Any Windows folder can be recognized as a library folder, and any vector drawing file (not just TurboCAD files) can be used as a symbol.

A symbol or part is placed into a drawing as a group; it has a unique name and is selected and manipulated as a single object. Because they are placed as groups, each instance of a part or symbol creates additional drawing data, and inserting numerous library items can increase your drawing size significantly. If you need to use many instances of a particular symbol or part, you can reduce your file size by making it into a block. A block is stored internally in a drawing, and references to it are inserted into the drawing rather than the objects themselves. See [Blocks](#)

TurboCAD provides several symbols and parts, organized in folders by category. Each symbol is a separate *.tcw file, and parts are *.ppm files. These files are installed, by

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default, to the "Symbols" folder in the TurboCAD installation folder.



You can modify these symbols (though it is preferable to modify copies and leave the originals intact), and also create and save your own symbols. Symbol manipulation is done via the Library Palette.

Favorites

The **Favorites** tab of the Library palette is where you can store symbols and parts you use often.

You can place an entire folder in **Favorites**. To do this, click the folder, then click **Add to Favorites**.



2. If there is no folder currently in **Favorites**, the folder will be placed in the main folder.

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3. Open the **Favorites** tab, and the folder is listed.



You can also add individual symbols or parts to **Favorites**. Return to **All Symbols** and locate the symbols you want to add. Another way to add a symbol is to right-click on it and select **Add to Favorites**.



You can choose to add the symbol to the main folder, or to an existing folder within **Favorites**. In this example, the symbol will be placed in the main folder.



Now **Favorites** contains the single symbol and the folder.

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6. If you want to save the **Favorites** as a library, click **Save Favorites as New Library**.



Inserting a Symbol from the Library into the Drawing

This section explains inserting symbols, which are standard TurboCAD objects saved as their own files.

Note: For inserting parametric parts. See

For an example of a symbol, open the "Fasteners" folder. Click on one of the symbols, and the **Description** tab at the bottom shows the name and location of the symbol's file.

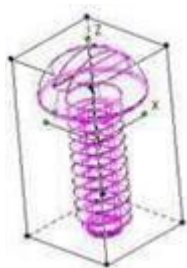
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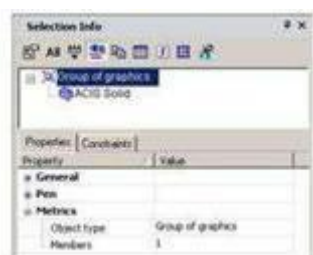
There are three ways to insert a symbol. You can click and drag the symbol from the palette to the drawing. Or you can double-click the symbol thumbnail. Or you can click the **Insert Symbol** icon.



The symbol appears in the drawing, in Select Edit mode.



In the Selection Info palette, you can see that the symbol is grouped. If you want to edit the symbol, you first need to **Explode** it.



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Library Folders

To see which folders will appear in the Library palette, look in the **Symbol Libraries** page of the **Program Setup**.



Folders that do not appear here can be added by clicking **New**, or you can load a file from within the Library palette.

2. Open the Library palette to see the folders it contains.



If you want to change how the folders are displayed (icon size, or detailed list), click this icon:



You can also view the folder structure in tree form by clicking the drop-down arrow for **All Symbols**.

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Note: The other tab is **Favorites**, where you can store symbols and parts you use often. See [Favorites](#)

The library contains symbols as well as parametric parts. Symbols can be 2D or 3D, and are objects saved as their own *.TCW file. For examples of symbols, open the "Bath" folder. (Your thumbnails may look different, or you may not see thumbnails at all.)



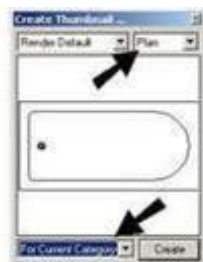
Note: If a symbol is not in *.TCW format, its thumbnail will be a generic icon.

6. To change the way thumbnails are displayed, right-click on one them and select .

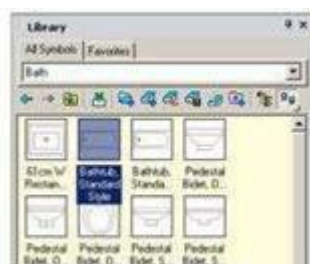
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The top right field defines the standard view (Plan, ISO, etc.). The bottom left field determines if the thumbnail properties will be applied to only the current thumbnail, all items in the current folder, or all items in the entire library.



In this example, all thumbnails in "Bath" now are now shown in plan view.



To maneuver within the Library's folder structure, you can use the **Back** and **Forward** arrows, or the **Up** arrow. Go back to root library folder.



Parametric parts and 3D symbols have more thumbnail options. Open the "Cabinets" folder - these are parametric parts created from scripts.

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Right-click on one of these thumbnails. On the top left field, you can choose Wireframe, Hidden Line, or Default, In addition, you can select one of the "Conceptual" renderings; three colors are available.



In this example, Hidden Line display is now applied only to the selected thumbnail.



Loading an Individual Symbol into the Library

There are two ways to save an individual symbol - saving the entire file or dragging selected objects directly into the Library.

Saving a File as a Symbol

If you create a symbol as its own file, simply save the file in a folder already identified as a library folder. Or you can save it to any folder, then load that folder into the Library palette. For example:

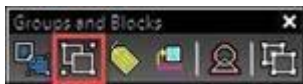
Create a new door and save the file as "FrenchDoor.tcw" in the "Symbols\Sample Symbols\3D Symbols\Door" folder.

Open a new file, open the Library Palette, and open the "Door" folder.

The "French Door" symbol appears in the palette, and you can drag it into the drawing.

Saving Selected Objects as a Symbol

To save objects as a symbol, select them and create a group.



(It is not essential to make a group, but the symbol will be easier to work with later.)
Open the Library palette, open a folder, and drag the objects by their reference point (the yellow circle) into the Palette. You will be prompted for a symbol name.

Loading Symbol Folders into the Library

This section explains how take folders of symbols you've already saved, and load these folders into the Library.

Note: For loading parametric parts into the Library, (See [Loading a Parametric Part into the Library](#))

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In this example, a few boxes, cones, and cylinders were created in TurboCAD, and each saved as its own *.TCW file.

Here is the folder structure for these symbols. The folder "3D Primitives" was created in "My Documents\TurboCAD\Symbols\My Symbols." "3D Primitives" has three sub-folders for boxes, cones, and cylinders. Each of these folders has a few *.TCW files.

There are two ways to add "3D Primitives" to the library. One way is to open **Symbol Libraries**, click **New**, and browse to "3D Primitives."



Another way to load this folder is within the library. Click **Load Folder**.



If you use **Load Folder**, browse to where you saved "3D Primitives" and load it.



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The "3D Primitives" folder now appears in the library.



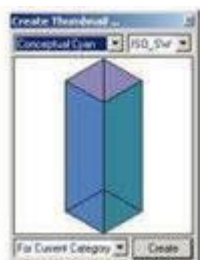
6. Double-click "3D Primitives" to see the three sub-folders inside.



When you first open a new folder that contains symbols, the thumbnails are empty.



Right-click on one of the thumbnails and select **Update Thumbnails**. In this example, an ISO view and "Conceptual Cyan" are used, and applied to the entire category.



This updates the thumbnails.



Use **Up** or **Back** to return to the three sub-folders. If you want to see all of the symbols contained in all three folders, click **Expand Subfolders**.

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Now all symbols from "Boxes," "Cones," and "Cylinders" are displayed.



To return to the folder structure, click **ExpandSubfolders** again.

To unload a folder, click **Unload Libraries**.



Find "3D Primitives" on the list and check the box. Then click **Unload**.



"3D Primitives" is no longer displayed.



14.5. Parametric Parts

(Available in Platinum)

Default UI Menu: Tools/Parametric Part Manager, Tools/Palettes/Parametric Part Script Editor

Ribbon UI Menu:



A parametric part is a group of objects that have parameters you can define before or after the part is inserted. For example, you can insert a bookcase whose height and shelf spacing can be defined.

Creating a Parametric Part

This section explains how to create a parametric part from within TurboCAD.

Note: You can also create a parametric part using a script. For details on creating scripts, check the "Docs" folder of the TurboCAD installation disk.

There are four basic steps in this process:

- creating the object(s) themselves

- saving the objects as a part and defining parameters

defining relationships between parameters

inserting the part

Step 1- Creating the Object(s)

In this example, a part will be created from a polyline with holes, made 3D by extruding it. The objects will be created using different sizes than the actual part that will be saved, to show that parameters can be used for accurate sizing.

In this example, **Auto Constraints** is turned on. This is so that dimensions can be used as parameters, and so that geometric relationships between objects will be maintained when parameters are changed.



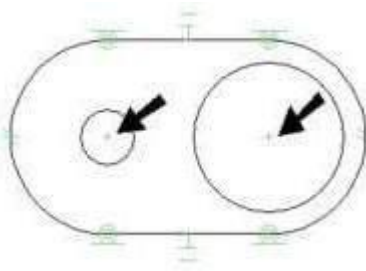
Note: For details on constraining objects, see [Auto Constraint](#)

2. Create a polyline using two identical linear segments and two tangent arc segments.

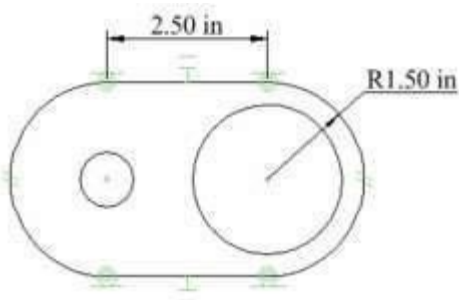
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Create two circles concentric with the polyline arcs. The circles can be different sizes now, but in the eventual parametric part, the circles will be identical.

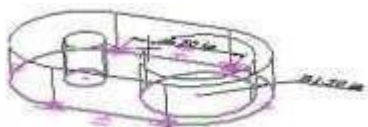
Apply **Concentric** constraints so that the circles will remain concentric with the polyline arcs.



Add a **Linear Dimension** and a **Radial Dimension** to measure the polyline.



Activate **Simple Extrude**, and make sure **Use Compound Profile** is active. Select the polyline, then press Shift and select two circles. Click to define the height, or enter an exact value.



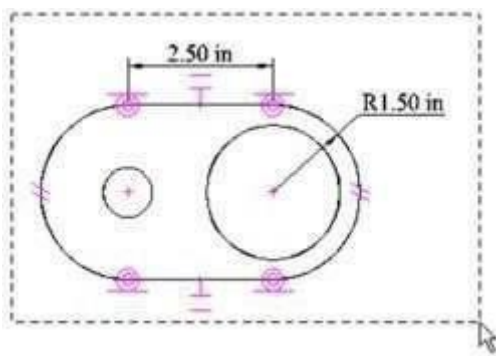
Step 2- Defining the Part and its Parameters

Now a part will be created from this object. There will be five parameters defined for this part:

H1 = radius of one internal circle H2 = radius of the other internal circle R = radius of the polyline arc W = length of the linear polyline segment D = extrude depth

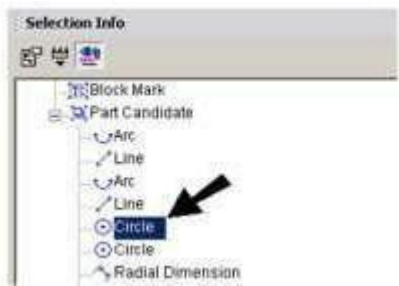
Select Parametric Part Manager.

Use a selection window to select all of the objects.

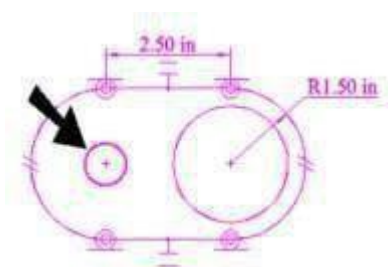


The Selection Info palette opens, and the part is temporarily called "Part Candidate." Expand this item to see what the part contains.

The parameters H1 and H2 are for the two internal circles. Highlight the first Circle in the palette.



The circle is also highlighted on the model.



The radius of this circle is found under "Metrics," at the bottom of the palette. Right-click on the **Radius** field.

Property	Value
Brush	
3D	
Metrics	
Object type	Circle
Area	0.56
Center	x=2.39 in, y=1.58 in
Radius	0.42 in
Circumference	2.65 in

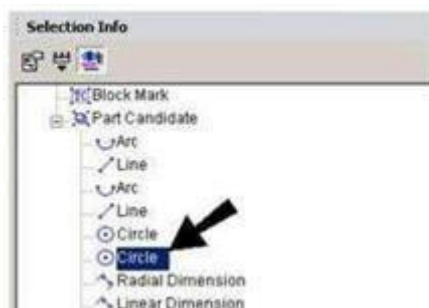
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6. Enter the parameter name "H1," then click OK.



Note: You are not restricted to defining only geometric parameters. You can also define material, pen color, line width, etc. as parameters.

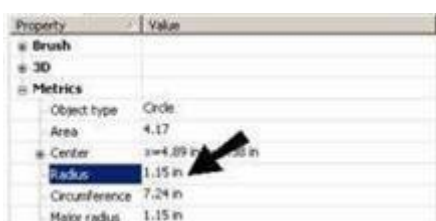
7. Next, highlight the other Circle.



The other circle is now highlighted on the model.

8. Right-click on the **Radius** field.

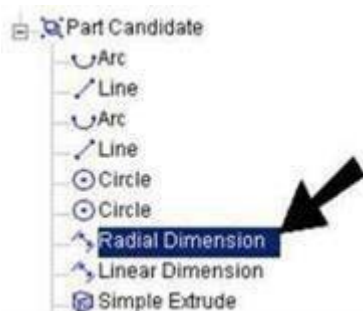
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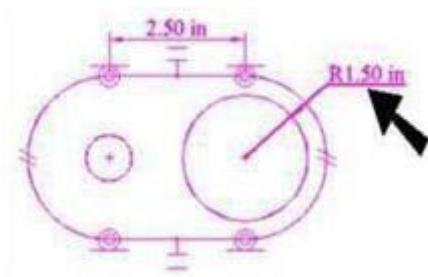
9. Enter the parameter name "H2."



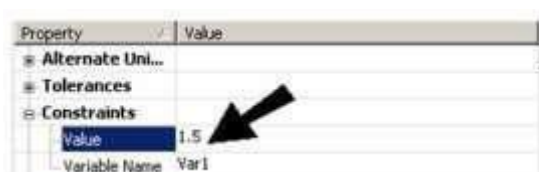
10. For the next parameter, highlight the Radial Dimension.



The dimension of the polyline arc is now highlighted.



The Value field of this dimension is found under "Constraints." Right-click on this field.

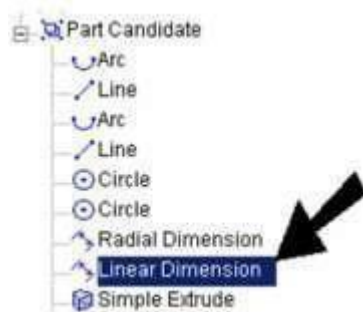


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12. Name this parameter "R."



13. Next is the Linear Dimension.



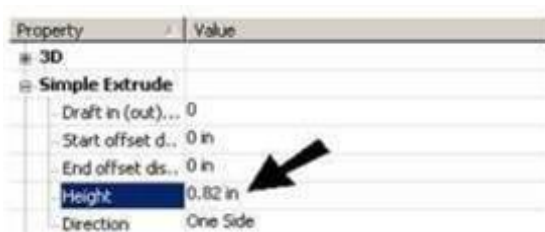
14. This value is also found under "Constraints." Name its parameter "W."



15. The last parameter is the depth of the part. Highlight "Simple Extrude."

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This parameter is called "Height" and is found under "Simple Extrude." Right-click on this field.



17. Name this last parameter "D."



Now that all five parameters are defined, select **Finish** from the local menu or Inspector Bar.



Note: There are other options in the local menu and Inspector Bar. **Relocate Reference Point** enables you to define the point by which the part will be inserted. For circular

parts, **Set Assembly Axis** can be used to set an axis for assembly using **Assemble by Axis** (see [Assemble by Axis](#)).

Step 3- Defining Relationships between Parameters

The **Parametric Part** window is now open, in which you can define relationships between the parameters you've defined. First, enter a name for the part.

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The eventual part will be defined by only three parameters: the hole radius, distance between hole centers, and depth. All other parameters will be functions of these three.

The names of the parameters currently listed don't tell the user exactly what they are for, so we can add parameters with more obvious names. To do this, right-click on the first parameter on the list, and select **Insert Parameter**.



Name this parameter "HoleRad."



"HoleRad" is now first parameter on the list. Under "Value," enter the default radius that will be used when you first insert the part.

6. Under "Type," specify "Linear."

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Create another parameter for the center-to-center distance between the holes. This parameter can be named "CenToCen". Specify an initial "Value" and "Type."

Under "Value" next to "H1," replace the number with "HoleRad." Do the same for "H2."

9. Define "R" as twice the hole radius.

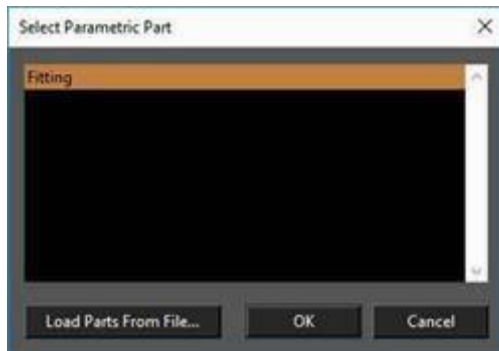
10. Set "W" as equal to "CenToCen."

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Now the three controlling parameters needed for the part are defined. They are "HoleRad," "CenToCen," and "D." Click Finish to close the **Parametric Part** window.

Step 4- Inserting the Part

Continue in the file in which you defined the part. Select **Draw / Parametric Part**. Select the part you just defined.



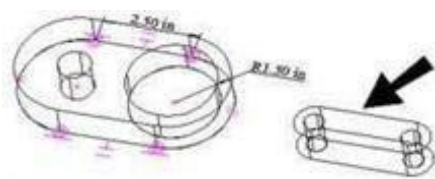
You may get a warning:

If you don't want to see the message again, check the "Don't show this message again." box. Then click OK.

Note: Load from File can be used to insert parametric parts saved in other files.

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Click to insert the part. The inserted part is shown to the right of the original geometry. Note that the two holes are equal, and that the polyline arc's radius is twice as large as the hole radius.



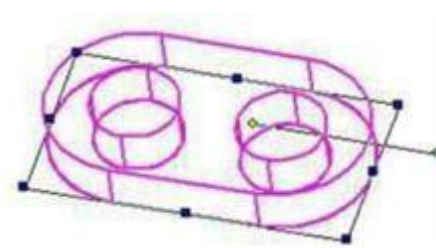
Select the part, and it appears in the Selection Info palette, with its three parameters available for editing. The initial values are the ones you set in the **Parametric Part** window.

4. Increase "HoleRad."



The internal holes updates, as well as the overall radius of the part, which was set to be twice as large as the hole radius. In some instances you may get a warning message "Compound Profile is not up-to-date" when inserting the part.

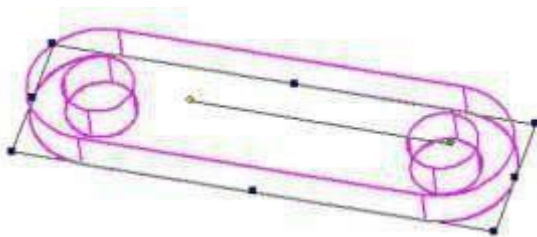
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5. Increase the "CenToCen" parameter value.

Property	Value
General	
Pen	
3D	
Parametric Symbol	
Symbol Parameters	
HoleRad	0.75
CenToCen	7.5
D	0.82 in
Metrics	

The distance between holes updates, while the radius values remain the same.



6. The last value to change is "D."

Property	Value
General	
Pen	
3D	
Parametric Symbol	
Symbol Parameters	
HoleRad	0.75
CenToCen	7.5
D	0.15 in
Metrics	

This changes the depth of the extruded part.



Editing a Parametric Symbol



With the edit tool active you can save all of the Parametric parts in a *.SLW file.

You can also Load Parametric parts from an *.TCW file.

To Edit an exiting part. Click the local menu option **Edit Existing Symbol**.

You can edit either the content or the parameters. Content is the physical geometry.

When finished, click OK and the part is saved with the changes.

Inserting a Parametric Part from the Library

The parametric parts included in the TurboCAD installation were created using scripts.

(If you want to view these scripts, see [Parametric Part Script Editor](#) .)

Note: For details on how to save and insert parametric parts you create yourself, see [Loading a Parametric Part into the Library](#).

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For an example of a parametric part, open the "Books" category. Click on one of the parts, and the **Description** tab at the bottom shows the name and location of the part's file.



2. Open the **Property** tab. These are the editable fields defined in the part's script.



The properties with a drop-down arrow contain a list of pre-set, selectable values.

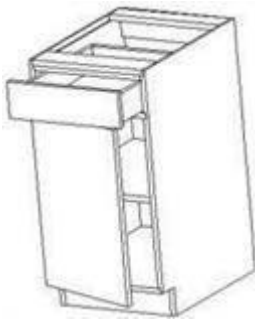


Fields that are not pre-set have a defined range of values you can enter. For example, **Open Angle** below must be between 0 and 90.



There are three ways to insert a part. You can click and drag the part from the palette to the drawing. Or you can double-click the part thumbnail. Or you can click the **Insert Symbolicon**. The part is inserted with the properties set in the palette.

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You can make changes to the part in the Selection Info palette. This palette shows the part's name, as well as fields for each of the properties shown in the Library palette. In this example, **Open Angle** was increased to 70 degrees, and **Open Drawer** was changed to zero.

Now the drawer is closed, and the door is open wider.



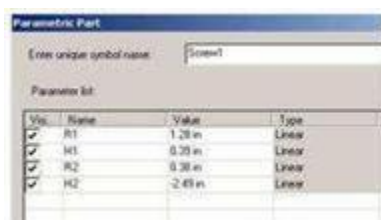
Loading a Parametric Part into the Library

This section explains how to save, and then insert, a parametric part created within TurboCAD. For details on creating parametric parts, see [Creating a Parametric Part](#)

In this example, two cylinders are used to create a parametric part representing a simplified screw.



Use **Parametric PartManager** to create a part from these two objects. There are four parameters defined: radius and height for both cylinders



When the part definition is completed, erase all objects in the drawing.

Select **Parametric Part** to insert the screw.



The part, and nothing else, is now in the drawing.



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Now the file can be saved. In this example, the file is saved as "Screw1.tcw" in the folder "My Parametric Parts."



Start a new file, and open the Library palette. To load the new folder, click **Load Folder**.



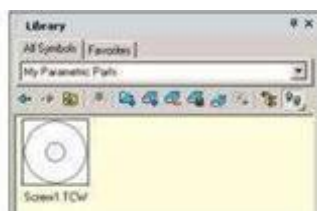
Browse to the "My Parametric Parts" folder and load it.



The folder now appears in the library.

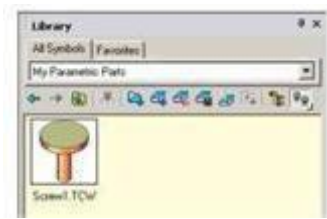


8. Open "My Parametric Parts" to see the "Screw1" part.

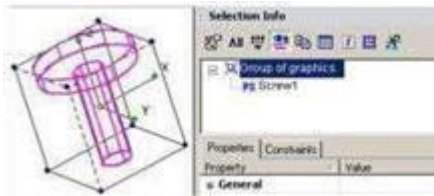


9. Right-click on the thumbnail and update it, if you want.

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Drag the part from the library into the drawing. Open the Selection Info palette, and you can see that the part is a group.



To edit the part, you can **Explode** it, or just highlight its name in the palette and edit its parameters.

Note: If you add more parts to the same folder, they will not automatically appear in the library. You need to use **Unload Library** to remove the folder, then use **Load Folder** to load it again.

Parametric Part Script Editor

Default UI Menu: Tools/Palettes/Parametric Part Script Editor

Ribbon UI Menu:

Other than creating a parametric part from within TurboCAD, the other way to create a part is to write a script. The parts provided in folders such as "Cabinet" are created this way.

For details on creating scripts, check [Parametric Part Script Editing](#)

If you want to write your own script, or edit an existing script, you can use the Parametric Part Script Editor.

To load an existing script into the editor, click **Open**.



Locate one of the scripts in the "Cabinet" folder.

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3. The script appears in the editor.

Note:Scripts can be edited in any text editor as well.

4. A scripted part can be loaded into the drawing by clicking **Insert PPM Symbol**.



14.6 Edit Selected Block

(Available in Platinum and Professional)



Default UI Menu: Tools/Edit Selected Block

Ribbon UI Menu:

User can edit the selected block directly within the TurboCAD drawing without having to select the Edit Content option from the Blocks Palette. This can be done by simply selecting the block and clicking “Edit Selected Block”, now added to the menu.

Steps to Edit Selected Block:

1. Select the block in the drawing you want to edit.
2. Select the "Edit Selected Block" from the menu and it will let you edit the block.

15. Manipulating the Display

This section covers ways to obtain the desired view of your model, and how to save views for future use.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

15.1. 3D View Manipulation

(Available in Platinum, Professional and Deluxe)

In 3D there are many ways you can change your view. The easiest way is to use one of the standard views - top, bottom, isometric, etc. See [3D Views](#).

Other tools for view manipulation include:

Camera movements: Incremental controls for adjusting the camera, and therefore your view of the model. See [Camera Movements](#)

Walk Through tools: Used to dynamically move the camera, presenting a moving viewpoint. See [Walk Through Tools](#)

Camera objects: Used to create and save views with specific parameters. The view for each camera object can be displayed in a separate window. See [Camera Objects](#)

Rendering: Enables you to see your 3D model as shaded. Certain render modes also enable you to view texture and materials. See [Render Styles](#)

15.2. Saving Views

(Available in all TurboCAD Variants)

Once you have used the zoom and pan tools to arrive at the exact view of your drawing that you want, you won't want to repeat all that work to get back to the same view later. You can save views and assign them names so that you can easily return to a saved view at any time.

In addition, saved views must be created if you want to create viewports in Paper Space. See [Viewports](#)

Note: For 3D views, you can also use camera object to save and display views in separate windows and with specific parameters. See [Camera Objects](#)

The **Named Views** tools are accessible from the **View** menu, or from the fly out **World Plan** icon on the **Standard** toolbar.

You can display the **Named Views** toolbar by right-clicking in any toolbar area and select **Named View**.



Note: Saved views can also be accessed and manipulated in the Design Director, in which you can also create new views. Render and camera settings can be manipulated here as well. See [Design Director-Views](#)

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Default UI Menu: View/Named Views/Named View

Ribbon UI Menu:



The **Named View** window can be used to create a new saved view, or to display, modify, or delete a saved view.



Named views are saved with your drawing and remain available unless you delete them.

Draft Rendering: Sets the rendering type when the view is rendered in **Draft** rendering mode. See [Creating a Rendered View](#) and [Render \(Camera\) Properties](#)

Quality Rendering: Sets the rendering type when the view is rendered in **Quality** rendering mode. See [Render \(Camera\) Properties](#)

Perspective: Activates **Perspective** mode. See [The Camera](#)

Angle: The view angle for perspective mode.

Position: The location of the camera position.

Target: The location of the point the camera is facing.

UP Vector: A point defining the up direction of the camera.

Ext ents: The boundaries of the view.

Saving a View

1. Use the zoom and pan tools to arrive at the view you want to save.

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Note: If you want to create a view using a specific area of the screen, see [Creating a View](#)

Select **Named View** from the menu or click the **Named View** icon to open the **Named View** window.

Enter a name for the view. The name can be up to 32 characters in length and can include spaces.

Click **New** to add the name to the list of named views.

Tip: Give the view a descriptive name that will help you remember it, such as "Garden Door." If you use names like "View 1" and "View 2," you'll quickly lose track of which is which.

Displaying a Saved View

In the **Named View** window, select the desired view and click **Go To**. This will leave the window open, in case you want to switch to another view. To immediately go to a named view and close the window, double-click on the view name.

Tip: If you want to display the previous view, use **View / Zoom / Previous View**.

Modifying a Saved View

Use the zoom and pan tools to arrive at the view with which you want to replace a saved view. Open the **Named View** window, select the view you want to replace with the current drawing view, and click **Modify**.

Deleting a Saved View

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In the **Named View** window, select the view you want to delete, and click **Delete**.

Previous View



Default UI Menu: View/Zoom/Previous View

Ribbon UI Menu:



Displays the previous view, whether or not it was saved.

Note: If you want to display a saved view other than the previous one, select the view in the **Named View** window and click **Go To**.

15.3. Views

(Available in all TurboCAD variants)

The view is the area of the drawing space that appears on your screen. As your drawing becomes larger and more complex, it's important to find views that let you work at the right location in your drawing, at an appropriate level of detail.

Tip: Layers are a useful way to change how objects. You can place objects on various layers and then manipulate layer visibility. See [Snap Modes](#)

This section covers views in 2D. When working in 3D, there are additional viewing tools and other ways to examine your model. See [3D Views](#) and [Examining the 3D Model](#)

Aerial View



Default UI Menu: View/Aerial View

Ribbon UI Menu:



Opens an additional window to provide a second view of the drawing. You can use **Aerial View** to create a close-up view of one small part of the drawing, or of the entire drawing. **Aerial View** is especially useful if you do not have a large monitor. You adjust the view in the drawing screen, without changing the aerial view. Use a rectangle to define the area you want to include in the aerial view. This view appears in the small drawing window.

The red rectangle represents what appears on the main drawing window.

The first three tools in the aerial view window control the view inside the aerial view:

Aerial Zoom In: Magnifies the aerial view.

Aerial Zoom Out: Shrinks the aerial view.

Aerial Zoom Extents: Zooms to an aerial view containing all objects in the drawing. The last two tools are used inside the aerial view window, but affect the overall drawing.

Aerial Zoom Window: Sets a new drawing view by using a selection window in the aerial view.

Aerial Pan View: Sets a new drawing view by moving the red rectangle that appears inside the aerial view window.

You can also pan the aerial view with the cursor. Place the cursor just inside one of the borders of the aerial view window. After a second or two, the cursor will change to a small solid arrow and the aerial view will move in the indicated direction.

Move the cursor away from the edge to end panning.

Panning

Default UI Menu: View/Pan

Ribbon UI Menu:



Panning means to move, or scroll, the view to another location across the plane of the drawing.

Two simple ways of panning are to use the scroll bars or arrow keys. The scroll bars work the same way that they do in other Windows programs. You can click the scroll bar arrow buttons to scroll one step, click in the body of the scroll bar, or drag the body of the scroll bar itself.

The arrow keys are also simple and intuitive: press the up, down, left, or right arrow key to pan a single step.

Tip: To quickly pan to a specific point, place the cursor at the point, then press Ctrl+End. You will instantly move to a view with the point you chose at the center of the screen.

Vector Pan



Pans the drawing based on a vector defined by two points.

Select a base point and a destination point. The drawing moves so that the base point moves to the destination point.

Pan to Point

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Hotkey: Ctrl+End

Pans the drawing so that the selected point becomes the drawing center.

If you use the **View** menu command, select a point and the point will move to the center of the screen.

To use the hotkey, place the cursor at the desired center and press Ctrl+End.

Redrawing

Hotkey: F5

Default UI Menu: View/Redraw

Ribbon UI Menu:



In most cases, the screen will be redrawn after each change. Occasionally, however, you may find that the screen is not completely "cleaned up" after an operation. You can manually redraw the screen using the **Redraw** command.

Zooming

Default UI Menu: View/Zoom

Ribbon UI Menu:

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Zooming means to move "closer to" or "farther from" the drawing space, viewing the drawing at a greater or smaller level of detail.

You can always check the current zoom percentage via the zoom indicator, located, by default, at the lower right corner of the screen.



Using a Wheel Mouse

Zooming with a wheel mouse is a particularly handy way to get around in a drawing.

Move the cursor to where you want to work, and move the wheel backward to zoom out or forward to zoom in.

Some wheel mouse setups require that the scroll bars be turned off for zooming to work. Some wheel mice are set up by default to scroll instead of zoom. If you would like your wheel to zoom, select **Tools / Customize** and open the **Intellimouse** page. You will be able to assign commands to the wheel functions. This feature works with other wheel mice as well.

Zoom In and Zoom Out

Hotkey: <+> or <-> (on numeric keypad)



Magnifies or shrinks the current view. The zoom factor is equal to two by default, but you can change this in the **Advanced Preferences** page of the **Program Setup**.

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Tip: Change the zoom factor to 1.4 to double the view every two zoom steps.

If you zoom using the <+> and <-> keys of the numeric keypad, the view centers on the cursor. If you use the **View** menu commands or icons, the zoom will center on the current view, regardless of the position of the cursor.

Note: As you zoom out, the representation of small objects is simplified, thereby increasing redraw speed. You can set the size at which this occurs using the **Suppress display** field on the **Display** page of the **Drawing Setup**.

Zoom Window

Hotkey: Ctrl+Home



Zooms to an area defined by a rectangle. The rectangle is defined by selecting its two opposite corners, or by dragging the mouse from one corner to the other.

Note: Because the dimensions of the zoom rectangle will probably not be exactly proportional to the screen, the closest view that includes the entire rectangle will be used.

Zoom Extents

Hotkey: Ctrl+Backspace



Zooms to an area that completely contains all visible objects (including lights and camera objects). This is the same area enclosed by the selection rectangle if you select all the objects. Any objects on an invisible layer will not be taken into consideration.

Zoom Full View

Hotkey: Shift+Backspace



Only available in Paper Space, displays the entire drawing sheet. The drawing sheet may consist of multiple printer paper sheets, indicated by dashed lines.

Note: Zoom diagnostic message appears when the user tries to zoom in or zoom out beyond the maximum zoom factor or limit.

Demo Video:

<https://youtu.be/IP8Wgn4c9pl>

Zoom Selection

Hotkey: Alt+Backspace



Zooms to an area that completely contains all currently selected objects.

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Tip: You can also use the Design Director to quickly zoom on an object or group of objects. For a single object, see [Design Director-Graphics](#). For a group of objects, see [Design Director-Categories](#).

Printed Size



Zooms at approximately the size the drawing will be when printed. In this view, one inch of screen space is approximately equal to one inch on paper. The exact ratio of screen space to Paper Space will vary, depending on your graphics display and screen resolution.

Uniform for All Views

Used for multiple windows of the same drawing, enables you to zoom all windows simultaneously.

Drafting Palette-Creating Standard Views

(Available in Platinum and Professional)

Default UI Menu: Tools/Palettes/Drafting

Ribbon UI Menu:

The Drafting Palette enables you to insert standard views of your model into Paper Space.

Note: If you want to insert predefined views (non-standard views or views you created in advance), see [Viewports](#)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

16.1. Creating Drafting Views

(Available in Platinum and Professional)

The section covers topics related to creating drafting views.

Breaks



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Breaks are used to reduce the lengths of objects which will not fit into the drawing.

Determine the area to cut:

Define the gap distance:

The result is a section with a Break.

NOTE: you can drag the break and the section independently. Make sure to move both simultaneously if you wish to retain the original association.

Break Line properties can be changed in Selection Info.

Some examples of different settings:

You can specify that the break be Vertical or Horizontal from the Local menu of the Break tool, or the properties of a break.

Multiple Breaks

Objects can have multiple breaks.

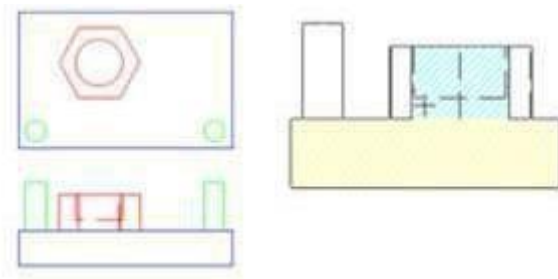


Creating Sectional and Iso Views

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The available standard views also include section views. Create a Section. Add this view to the view list by clicking the **Add** icon.

When this view is inserted, the scale is too large and the orientation is incorrect, relative to the **Plan** view.



Drag the section view over the **Plan** view. This not only sets the scale, but also sets the orientation.

If you press Shift here to break the alignment, the rotation will also change back. Only the modified scale is preserved. Here are the three views created so far. You can change the hatch pattern of section views in the **SectionBrush** page of the view's **Properties**.

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For the last standard view, create an **ISO_SE** view.

Insert this view. Because it is isometric, if you try to set the scale by dragging it over one of the other views, the alignment will be diagonal. Therefore, set the scale manually using the **Scale** field.

To change the appearance of any view, double-click it in **Select** mode to open its **Properties**. For the isometric view, open the **Pen Hidden** page. Set the **Pattern** to

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Invisible so that hidden lines will not be displayed.

Note: For other **Properties** options, see [Properties of Standard Views](#)

Here is the modified isometric view.

Creating Standard Views

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First, select the objects you want to include in the view. In this case, select all the objects. In the Drafting Palette, select the standard view from the lower group of icons. Start with a **Plan** view - the preview appears in the palette window.

Add this view to the view list by clicking the **Add** icon.



The view name appears on the list. You can change it by clicking the **Rename** icon.



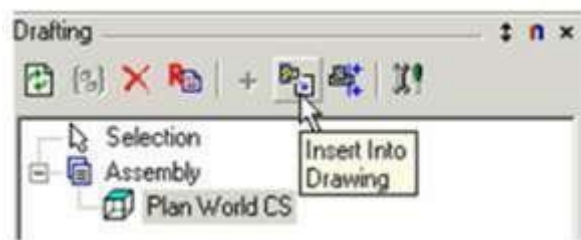
Note: You could have created an assembly of the entire model (see [Parts and Assemblies](#)), but selecting objects, then creating and adding a standard view creates an assembly automatically. Note that the **Plan** view appears under the **Assembly** header.

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Switch to Paper Space. (You can insert views into Model Space as well, but Paper Space is more appropriate.)

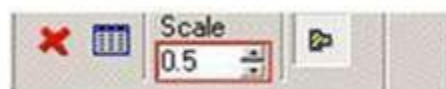


Highlight the view name, and click **Insert into Drawing**.



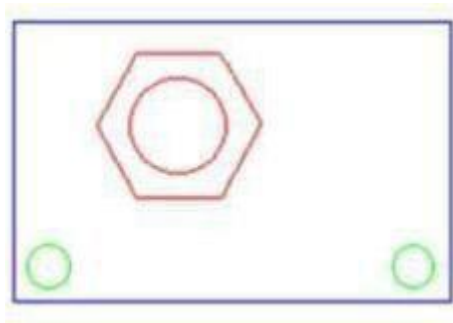
Place the view into the Paper Space. In this case, the scale is too large.

Press Tab to access the **Scale** field, and enter the new scale value.



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The view is now half as large.



Note: You can also change a view's scale in the **Format** page of its **Properties**.

For the next view, create a **Front** view.

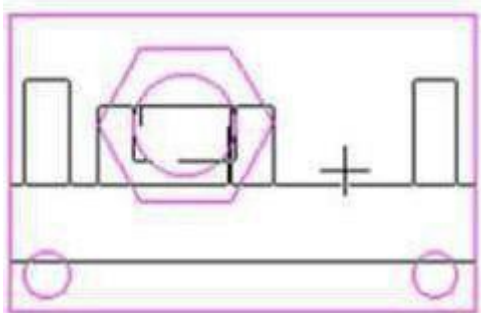
Add it to the list.



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Use **Insert into Drawing**, or simply drag it from the palette (drag either the view's name or drag the preview) into the drawing. Again, the scale is too large.

To set the scale to that of the **Plan** view, simply drag the **Front** view over the **Plan** view. It assumes the same scale.



This also aligns the **Front** view to the **Plan** view. You can move the Front view up and down, but not left or right. (To break this alignment, but preserve the modified scale,

press Shift).

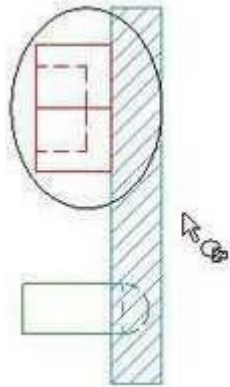
Note: You can always move an aligned view (or any view), by selecting it and dragging its reference point. See [Moving Objects in SelectEdit](#)

Fragmental Views

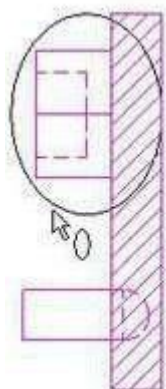
To create a broken-out section view, place a closed polygon or circle overlapping an existing inserted view, then click on the **Fragmental View** icon on the palette toolbar.

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Select the drafting object.



Select the **closed** 2D polygon or circular object that overlaps the drafting object.



The Assembly list will now show a **Fragmental View** at the bottom of the list and in the **Preview** window, when selected.

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You can now insert this view to your drawing. You can edit the 2D shape overlapping the drafting object and the inserted **Fragmental View** will update.

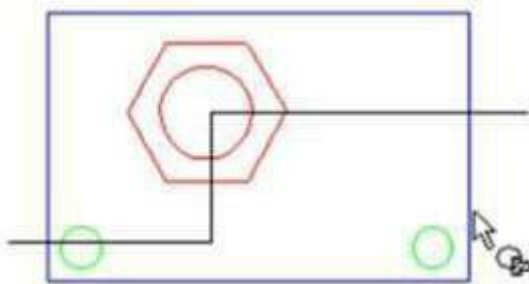
Section by View Line and Aligned Sectional Views

Use a **Polyline** to draw a multi-segmented section line on the **Plan** view.

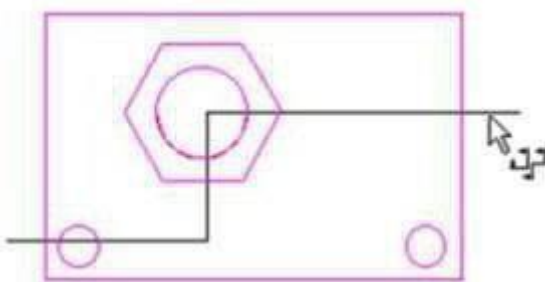
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Click Create by View Line.

If you want to assign text to the view (such as a letter), enter it in the **Text** field of the Inspector Bar. First select the view whose sectional view you want to create.



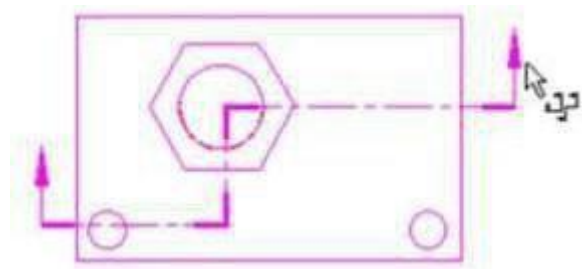
Then select the view line, in this case, the polyline.



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In the next step you define the viewing direction. Move the cursor to switch between the two direction options.

Click when the direction is correct.



Note: You can also change the viewing direction after the section line is created. Open the section line's **Properties** to the **Format** page, and check (or uncheck) **Forward Side**.

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The aligned view is created and is listed in the Drafting Palette, even though it does not yet appear in the drawing.

Insert the aligned view into the drawing, setting the scale according to the **Plan** view. By default, this is an **Aligned** view - what you see is equivalent to "unfolding" the polyline.

To switch to a non-aligned view, open the view's **Properties** to the **Format** page. Deselect **Aligned View**.



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The view is no longer "unfolded."

For another way to change an aligned section view, use the **Edit Tool** on the polyline (see [Edit Tool](#)).

Change the polyline by moving, adding, or deleting nodes. The sectional view updates as you change the polyline.

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If you want to add or change characters on the section line, use the **General** and **Text** pages of the section line's **Properties**. See [Properties of Section Lines](#)

Note: that there is a difference between a section and a sectional view. The 3D function **Section** creates an actual section of a 3D object, whereas a sectional view is what you see when looking in the direction of the section line.

In this example, the lower view is a sectional view looking in the direction of Line A-A.

And this is the section taken at Line A-A.



16.2. Drafting Palette Toolbar



Create Part / Assembly: Enables you to define a part or a combination of parts (assembly) for creating views. See [Parts and Assemblies](#)

Delete: Deletes the selected part, assembly, or view.

Rename: Renames the selected part, assembly, or view.

Add: Adds the selected standard view that appears in the bottom of the palette to the view list at the top of the palette.

Insert into Drawing: Inserts the view selected from the view list into the drawing. (You can also drag the name from the list into the drawing to insert a view.)

Create by View Line: Creates a sectional view from an existing view, by using an existing line or polyline as the section line.

Fragmental View: Creates a Fragmental view (breakout) of a specific part of section or view.

Breaks: Breaks are used to reduce the lengths of objects which will not fit into the drawing.

Drafting Palette Options: Contains options for deleting objects and what appears in view

previews.

Layer Set: Different layer sets could be selected for the drafting palette objects.

Lock When Inactive: When selected Drafting palette objects will not refresh while the Drafting palette is closed.

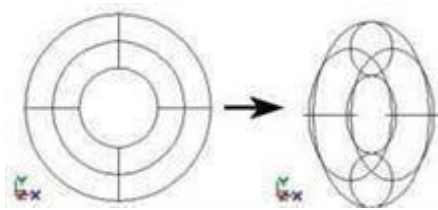
Create Undo/Redo: Add changes to Drafting Palette objects to the Undo/Redo history.

Use ACIS references: Defines that the controlling geometry for dimensions and insertions will be ACIS referenced for ACIS based Drafting palette objects.

Convert curves to polylines: Curves in Drafting palette objects will be displayed as polylines. This increases processing speed but minutely decreases accuracy. **Max ctrl pts:** Specifies the maximum number of polylines will contain for any curve section. The lower half of the palette contains standard views. The top row contains orthographic and isometric views and the bottom row contains sectional views.



Entity CS, World CS: Sets the view based on one of these coordinate systems. As an example, a torus is created and rotated about the Y axis.

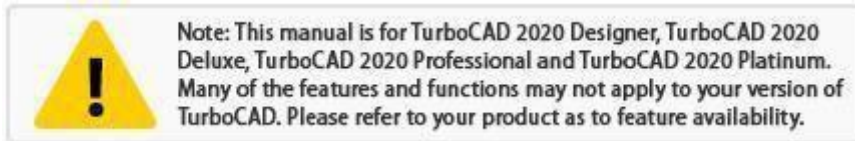


With **Entity CS**, the **Plan** view looks down on the torus in its own CS, even though the torus is rotated.

With **World CS**, the **Plan** view looks down on the rotated torus.



16.3. Drafting Reference Point



Note: It is important that the Create Editing History option is on (Options|ACIS) if you want boolean operations to be recorded in a part or assembly. Otherwise the part/assembly will be deleted since it is essentially a new object.

You can adjust the location of the Reference point for a Drafting object in the following manner. Reference point functionality applies only to the WorldCS and WorkplaneCS, it isn't allowed for EntityCS. WorldCS is the preferred mode.

Note: it is important to understand that using the WorkplaneCS will mean the composite WorkplaneCS of all the source 3D entities for Drafting Part/Assembly. The default Reference point will therefore be the Center of Extents of all these entities, not the original as is the case with the WorldCS.

You can also turn Reference point functionality on and off.

1. Create the drafting object from sphere with followed options turned on.

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With the New drafting object created, the reference point position will be defined automatically according to center of extents of the source entities. The Reference point is defined for the DRAFTING PART (with all of its drafting objects), not for each single drafting object within it:

Add other 3D parts to the sphere to change the drafting object (The position is always related to reference point)

To define new reference point, right click on PART object in the drafting tree and select Define Reference Point:

5. Click to define the new reference point position for the PART in Model Space:

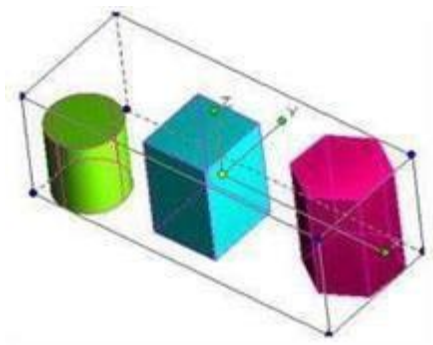
All drafting objects within a part where reference point position was changed will be updated accordingly to new reference point position:

16.4. Parts and Assemblies

Parts and Assemblies

You can create views from individual parts in the model, from selected objects, or from the entire model. This example uses a cylinder, box, and hexagonal prism.

1. For the first assembly, select all three objects.



2. Click Create Part / Assembly.



The header **Assembly** appears in the Drafting Palette.

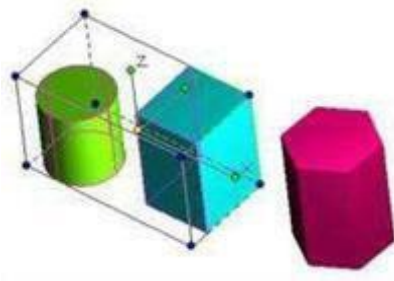


This is the default name, but you can change the name using the **Rename** icon, or simply by clicking in the name field and entering the new name.

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4. For the next assembly, select the cylinder and box.



Make an assembly from these two objects, and assign a name.



To see the difference, first highlight the first assembly, then click the **Plan** view icon.

The preview window shows the plan view for all three objects.

Then highlight the second assembly and look at the preview of the **Plan** view. Only the cylinder and box appear.

You can also create views of single parts. If you click a single object (cylinder, box, or prism) and click **Create Part / Assembly**, the part name and description appear in the



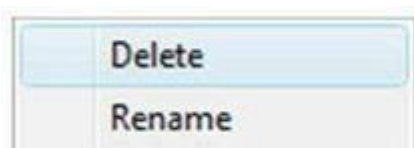
Drafting Palette.

Deleting Assemblies and Views

You can delete any assembly or view at any time.

Right click on the entity in the Drafting Palette and select delete.

When you are prompted if you want to delete the object, select Yes.

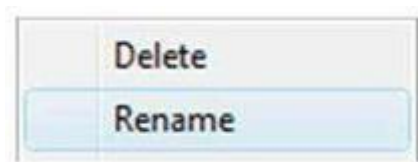


Renaming Assemblies and Views

Right click on the entity in the Drafting Palette and select Rename.

When the name field highlights, type in the new name.

Press Enter to set the new name.



Locking Assemblies and Views for Caching

You can lock assemblies and views so that they do not refresh every time you modify the model or the layout. This is especially useful for large or complex models as it reduces refresh time. **To Lock a View**

Right click on a view in the Drafting Palette.

Select Locked.

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When a view is locked a check mark will appear next to the Locked menu Item.



To Unlock a view

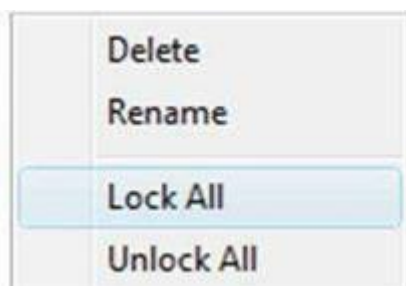
Right click on a view in the Drafting Palette.

Select Locked, and the check mark will disappear.

To Lock an Assembly You can lock all of the views in an assembly.

Right click on an assembly in the Drafting Palette.

Select Lock All



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To Unlock an Assembly

Right click on an assembly in the Drafting Palette.

Select Unlock All



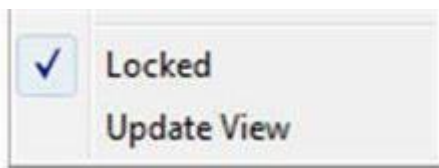
When a view is lock a small lock symbol will appear on the icon of the view. If the view is up to date the lock will be blue. If the model or layout has changed the lock symbol will appear in red.

You can update a view at any time to bring it current.

Right click on a view in the Drafting Palette.

Select Update view.

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16.5. Properties of Section Lines

Properties of Section Lines

A line or polyline can be used to create a section view.

General Properties

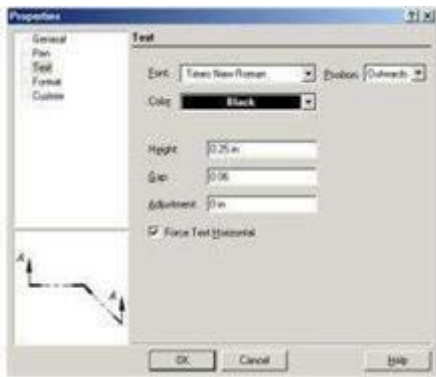
Use the **Attributes** field to enter or edit characters you want to appear on the section line. The **Text** window contains formatting options for this text.



Text Properties

If you entered characters in the **Attributes** field of the **General** page, this window contains text formatting options.

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Position: Sets the side of the arrow where the text will be located.

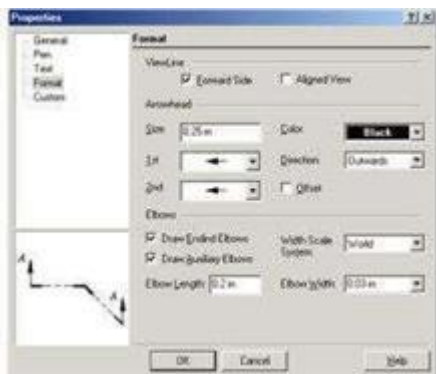
Gap: Moves the text away from the tip of the arrowhead.

Adjustment: Moves the text in the opposite direction of **Gap**.

Force Text Horizontal: Text will be horizontal regardless of the section line orientation.

Format Properties

These are properties that define the appearance of the section line, and the type of section created.



View Line

Forward Side: Enables you to switch the viewing direction of the section line.

Aligned View: Select this if you want the section to be created assuming the section line is "unfolded." If unchecked, the section will be created assuming a uniform viewing direction.

Elbows: These are the thick additions to the section lines that appear at interior corners and at the ends.

16.6. Properties of Standard Views

For each standard view, the Properties window contains options for changing the scale, pen for visible and hidden lines, and section brushes.

Format Properties

This window enables you to change the **Scale** of the view.

Brush Rotation: Select Rotatable Brush or Uniform Brush. This sets whether the brush pattern rotates or not when a section line has internal nodes and offsets. **Angle:**

Specifies the angle of brush rotation, if the brush pattern is rotatable

Suppress tangency hidden lines: Suppresses overdrawing where two or more lines are overlapping because of tangency. Additional format properties are available from the Selection Info Palette.

Curve Segments Amount : This option defines the Number of Segments between adjacent control points and applies to all curves inside of drafting object. This is a memory saving option.

Don't Show Hidden Lines: This option specifies whether all hidden lines of a drafting object are shown or hidden. This allows for more predictable behavior as invisible Hidden Lines will not interact with SNAPs

Curve Segments amount

Layer Set: Manage/select drafting layer sets.

In the Drafting Palette Options select a Layer Set. The Layer Set can contain both visible and non-visible layers.

Drafting Object Scaling:

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Users can now define the Scale of Drafting Object through the local menu Properties of the Object.

Pen Properties

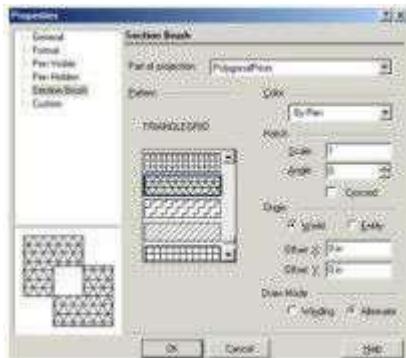
Properties of visible and hidden lines.



Part of projection: enables you set pen properties of hidden and visible lines of different objects in the view. For other options on this page, see [Properties Window](#).

Section Brush Properties

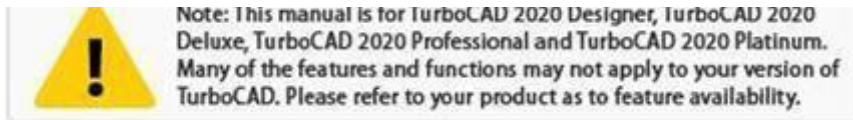
Properties of the hatching of section views.



Part of projection: enables you to set brush properties for different objects in the view. For other options on this page, see [Properties Window](#)

17. Paper Space and Printing

Once your drawing is complete, you may need to view it on paper. This section covers features that are geared toward printing, including Paper Space and viewports. You can also publish your drawing in HTML format for online distribution.



17.1. Paper Space

(Available in all TurboCAD Variants)

Default UI Menu: Insert/PaperSpace

Ribbon UI Menu:



While Model Space is typically used while creating geometry, Paper Space is geared toward creating the final layout of your drawing for printing or plotting on paper. In Paper Space, you typically arrange drawing elements on one or more sheets of paper. The Paper, the large white rectangle in the middle of the drawing window, shows you how your drawing will be laid out on the page when printed. You do not have to draw on the paper; you can draw anywhere in the drawing space and later fit the drawing to a sheet of paper. You can also turn off the paper sheet shown. The pattern of horizontal and vertical lines, which may be shown on the paper, is the grid that marks exact locations in the drawing.

Tip: You can use the TC Explorer Palette to see all Paper Spaces defined for any open drawing. See [Drawing Settings](#)

Manipulating Paper Spaces

By default, a new file contains one Paper Space, named Paper 1. You can create, delete, or rename spaces by using the **Paper Space** menu, or by right-clicking on any Space tab to access the local menu. These menus can also be used to switch between spaces.

Insert: Creates a new, empty Paper Space.

From Template: Allows you to import paper space and layouts from existing TCT and TCW files.

Rename: Enables you to assign a name to the active Paper Space.

Delete: Deletes the Paper Space that was last inserted. You will not be able to delete the last remaining Paper Space of your drawing.

Duplicate: Creates a copy of the Paper Space.

Properties: Opens the **Properties** window (see [Paper Space Properties](#)). Different Paper Spaces can have different properties.

Print Current: Opens the Print Dialog for the current space.

Print Queue: Opens the Print Queue.

Using Named Views

TurboCAD Platinum, Professional, Deluxe only

Named Views can now be associated with a Layer Set, allowing for more efficient management of that Set. Also, when a Named View is inserted as a Viewport into a Paper Space, that space ID (name) is now properly saved.

From Template

To import Paper Space layout from existing files:

Right click on a space tab then select From Template.

The Open dialog will appear.

Select the desired file and click Open.

The Insert PaperSpace(s) dialog will appear.

Select the Paper Spaces you want.

Click OK.

The Paper Spaces will be inserted into your drawing.

It is important to import only paper spaces that are using the same unit system (English/Metric). If differing unit systems are used this will result in incorrect scaling of layout elements.

Print Queue

To import use the Print Queue:

Right click on a space tab then select Print Queue. Or

Select Print Queue from the File menu.

The Print Queue dialog will open. If you opened the queue from the space tab, that space will be selected for printing.

In the Print Queue you can select the spaces you want to print. If you double-click on any of the listed spaces you will open the print dialog for that space. You can configure the print setup for that space including the printer to be used. The order of the items in the queue sets the order in which the spaces are sent to the printers.

Options:

Select All: Selects all of the spaces for printing.

Deselect All: Deselects all of the spaces for printing.

Move Up: Moves the currently selected space up in the queue.

Move Down: Moves the currently selected space down in the queue.

Properties: Opens the Print Properties for the currently selected space.

Copies: Sets the number of copies to be printed.

Paper Space Properties

The **Page Setup** page contains options for the printer, printer paper, worksheet selection, paper size, paper orientation, and printing origin. These settings are saved with the drawing.

The various Paper Spaces can be used for different print setups. For example, you can have one setup for shop drawings on D size paper, and another with renderings on A size paper. The **Print Styles** page enables you to select a print style.

See [Print Style Options](#)

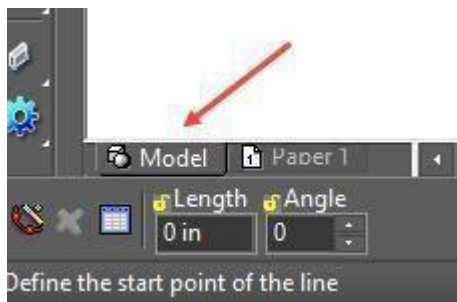
Switching to Paper Space

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You can use the small button at the top left corner, at the intersection of the rulers, to toggle between Model and Paper Spaces.

Note: If the rulers are turned off, this button will not appear. Use **Rulers** to toggle the ruler display.

There are also workspace tabs at the lower left corner of the screen. By default, each file contains one tab for Paper Space. Click the tabs to switch spaces.



Note: If scroll bars are turned off, these tabs will not appear. Use **Scroll Bars** to toggle the scroll bar display.

17.2. Printing

(Available in all TurboCAD variants)

Default UI Menu: File

Ribbon UI Menu:



TurboCAD has a wide range of printing features, enabling you to scale and center your drawing on the page or to tile your drawing across multiple pages. TurboCAD also provides a full range of advanced printing options for experienced users.

Paper Space is the work mode used for layout your drawing for printing. See [Paper Space](#)

Note: Under Windows, plotters are treated as printers.

Page Setup

Default UI Menu: File / Page Setup

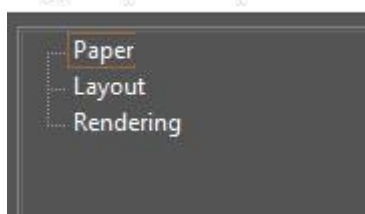
Ribbon UI Menu:

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Controls exactly how drawings will be printed. The three pages of this window are **Paper**, **Layout**, and **Rendering**.

Page Setup: Model Space



Note: **Page Setup** is different than the **Page Setup Wizard** (see [Page Setup Wizard](#)), but any relevant parameters set in the Wizard will appear in **Page Setup** as well.

After setting the parameters in this window, you can either print directly from the **Page Setup** (click **Print**), or click OK to accept the changes and return to the drawing. You can then print using **File / Print**, and the **Page Setup** settings will be used.

Tip: You can also access the **Page Setup** via the **Print** window.

Page Setup - Paper

Sets the size, orientation, and other parameters of the printer paper and the drawing sheet. The printer paper is the physical paper that goes through the printer; the drawing sheet is the area on which your drawing will be printed, which can span multiple sheets of printer paper.

Note: These settings are not retained with the drawing file.

Printer Paper: Sets the size and orientation of the printer paper. These options can also be set via the **Print** window.

Setup: Enables you to choose another printer or access functions that directly control your printer. This window is specific to your printer driver, so check your printer or printer driver documentation for details.

Default Printer: Sets the printer paper parameters to those of the default printer.

Drawing Sheet Size: Controls the size and orientation of the area on which your drawing will be printed. This area can be imposed on a single sheet of printer paper, or it can span multiple sheets of printer paper. Its orientation can also be set separately from the orientation of the printer paper. The page display on the right will reflect the relationship between printer paper and drawing sheet. The drawing sheet size will be shown in white (or a custom color if one has been set), and the dashed lines show individual pages of paper. Changing the drawing sheet size may change the number of columns and rows.

Options: Enables you to set what elements are printed. These options affect only the printing of the drawing and are not reflected in the appearance of the drawing on screen.

Print Margins: Prints a border along the margin of the printer sheets. This is useful for framing a drawing that is printed on a single sheet.

Print Crop Marks: Prints crop marks on the printer sheets. This is useful if you want to print across multiple printer sheets and crop the sheets so they can be pasted together.

You may need to increase your margins to make room for the crop marks. **Print Grid:** Prints the grid, as long as it currently displayed. See [Grid](#)

Print Construction: Prints construction geometry. See [Construction Geometry](#)

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Printing Scale: Scales the drawing so that it fits within the drawing sheet (click **Fit**), or you can choose a custom scale. If the desired scale does not appear in the list, you can type directly using the format 1 in = 1 ft.

Keep Centered: Keeps the drawing centered on the drawing sheet. It may be necessary to turn this option off, then on again, for the desired effect.

Page Setup - Layout

Controls the number of rows and columns (sheets of printer paper arranged vertically and horizontally), as well as other paper parameters.

Margins: Margins are the dotted rectangle lines inside the edges of the paper, that let you know whether you are drawing inside or outside the printable area of your drawing. You can specify the top, bottom, left, and right margins of the printer paper in Paper units. You can also enter units other than the default units.

Get Margins from Printer: Sets the minimum margins for your printer and page size.

Rows / Columns: Sets the number of rows and columns of printer sheet paper for tiled printing. As you increase the number of rows and columns, the **Drawing Sheet Size** (the area on which your drawing will be printed) increases accordingly.

World Height / Width, Paper Height / Width: Sets the dimensions of the drawing sheet in world or paper units. As you change these values, the number of rows and columns will automatically update.

Origin: Enables you to move the paper to a different location in the drawing. Specify the

location of the lower-left corner of the paper in absolute coordinates. This feature is useful if you want to print a particular area of a large drawing by relocating the paper to the area that you want to print.

Page Setup - Rendering

Sets the resolution at which a rendered drawing will be printed.

Printing quality for Rendered Images: Select the resolution for printing rendered drawings.

Note: When the resolution is set to a value that is higher than the maximum resolution supported by the printer, the image will be printed at the printer's maximum resolution.

Apply to All Spaces: It applies settings to all paper spaces in the drawing when checked.

Printer Paper and the Drawing Sheet

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One potentially confusing concept is that of the printer paper versus the drawing sheet. Printer paper means the physical sheets of paper that go through the printer. The drawing sheet is the total surface area on which your drawing will be printed. The drawing sheet can correspond to a single sheet of printer paper, or it can span multiple sheets of printer paper.

Both the printer paper and the drawing sheet have adjustable sizes and orientations. This provides a great deal of flexibility when you want to print a large drawing on multiple sheets. You could, for example, print a poster in landscape orientation three feet wide and two feet tall. You could print this virtual sheet out on 32 sheets of 8.5" x 11" paper oriented in portrait, or on 30 sheets of the same size paper oriented in landscape.

Printing from Model Space

Default UI Menu: File/Print

Ribbon UI Menu:



To get a printed copy of the objects appearing in the current window view, you can use **File / Print** and choose to print the current view.

Print Options

Default UI Menu: File/Print

Ribbon UI Menu:

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Hotkey: Ctrl+P

Icon on Standard Toolbar:



Provides a set of tools for printing a single view of a drawing, or printing specific sheets drawing spanning multiple sheets of paper.

Printer: Select a printer and manipulate its properties.

Name: Select the desired printer from the drop-down list. All printers set up on your system should appear on this list.

Properties: Opens the **Properties** window for the selected printer.

Page Setup: Controls exactly how drawings will be printed. See [Page Setup](#).

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Default Printer: Selects the Windows default printer.

Print to File: Prints the document as a file instead of routing it directly to a printer. You will be prompted to specify the filename and location.

Print Range: Options for setting a specific view of the drawing to print, and for selecting a range of pages when printing a drawing that spans multiple printed sheets.

Tip: This is useful for printing tiled drawings if you only want to print specific pages.

All: Prints the entire current drawing. If you choose **Selection** under **Print Source**, only objects that are selected will be printed. If your drawing spans multiple printer sheets, the total number of sheets will be shown in this option title.

View: Prints a single view of the drawing. You can print the current view by default, or click in the list box and choose any named view.

Pages: Prints a range of pages if the **Page Layout** is set to print multiple pages. The printer sheets are counted starting in the lower-left corner from left to right, and from bottom to top.

Fit on 1 Page: Prints everything on a single page. This option can be used either for printing a view or printing the entire drawing. **Print Source:** Select what you want to print.

Drawing: Prints the current drawing.

Selection: Prints only selected objects.

Color Mode:

Normal: Uses the predefined printer settings.

Gray tint: Used for color printers. Disables color mode and prints in black and white, using grey tints for other colors.

Black only: Disables color mode and prints in black only; no grey tints are used.

Copies: Sets the number of copies and whether you want multiple copies to be collated.

Print Styles: Sets the print style to be used while printing. See [Print Style Options](#)

Full Preview: Previews your drawing before printing.

Print Window

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You can create printouts by arbitrarily specifying a window of the desired area of a drawing.

1. Go to the Print dialog.

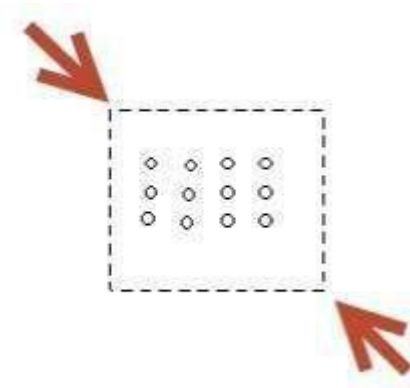
Change the Print Range to View

In the View drop-down select Window.

Press the Define Window button

Click two points to define the print window.

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When the Print dialog reappears click the Ok button. If you wish to review the print area prior to printing click the Preview button.

Previously, it was not possible to zoom in and out using the wheel mouse on the print preview. Now you can zoom in Print Preview windows, prior to printing to be sure things are looking correct Such as line thickness etc.

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Demo Video:

<https://youtu.be/CNPWhAfiFNU>

Print Style Options

Default UI Menu: File/Print Styles

Ribbon UI Menu:



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The Print Styles Option dialog allows you to View and Specify the current print style.

To set the Current Print Style

Open the Print Styles Dialog.

Select the Style from the Print Style Table Name drop-down.

Print Styles

Default UI Menu: File/Print Styles

Ribbon UI Menu:



Enable you to change color, line style, and brush of an object when you send the drawing to print. These settings do not affect the objects on the screen, only how objects are printed.

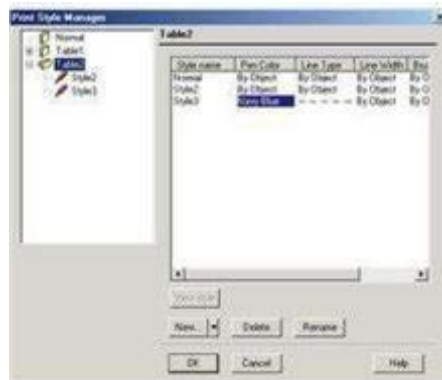
Print Style Manager

Default UI Menu: File/Print Style Manager

Ribbon UI Menu:



Print styles can be created, modified, or imported via the **Print Style** manager.



View Style: Opens the **Properties** window for the selected style.

New: Enables you to create a new table, or a new style within the selected table. You can also duplicate items, and import AutoCAD plot styles.

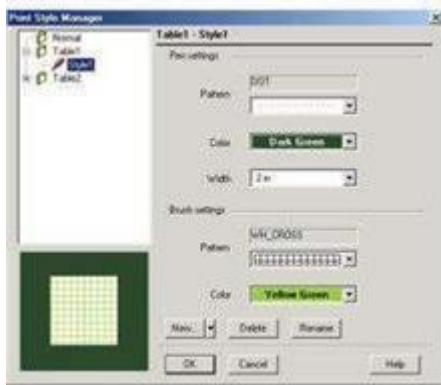
Note: These features are also available by right-clicking in the table list area.

Delete and Rename: Removes or renames the selected table or style. Deleting a style cannot be undone.

Print Style Properties

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To access the properties of any print style, select the style and click **View Style**.



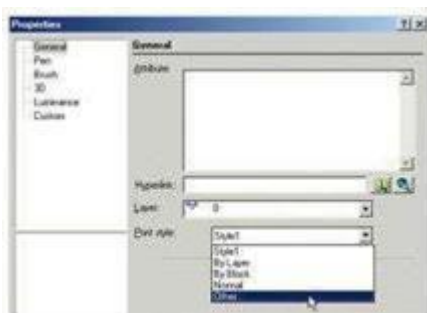
Pen Settings: Sets the pattern (line style), color, and width for the pen that will be applied to an object during printing.

Brush Settings: Sets the pattern and color for filling objects during printing. This only applies to objects to which a brush has been assigned.

Applying Print Styles

Once print styles have been created and/or modified, you can add them to your drawing via the **Print Style** page of the **Drawing Setup (Options / Print Styles)**. While in Paper Space, you can also access this page through the **Properties** window. An object's print style is set in the **General** page of its **Properties**.

Select "Other" from the drop-down menu to open the **Print Style** manager.



Note: Certain types of objects cannot display print styles.

Selecting Print Style Type

You can use two different Print Style type: Named Based and Color Based. Only one type of styles can be used in a drawing at the same time.

STB - Name Based Print Styles

Name based print styles correspond to AutoCAD STB files. They map a named style to a specific print presentation and group of settings.

CTB - Color Based Print Styles

Color based print styles correspond to AutoCAD CTB files. They map each indexed colors to a specific print presentation and group settings.

To Specify the Type of Styles Used

Open the Print Styles Dialog.

Use the option button to specify whether you want named or color based print styles.

Simple Printing and Tiled Printing

To make a simple print of a drawing that fits on one sheet:

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Select **File / Print** to open the Print window.

For **Print Range**, click **All** to print your entire drawing.

Check **Fit on 1 Page**.

Select the printer, and click **Properties** to set the paper size and orientation.

Return to the **Print** window, and click **OK** to print your drawing. Your drawing will be centered and printed on a single sheet of paper.

If your drawing is larger than the printer sheet, as it is in most cases, you need to use tiled printing. This method prints sections of your drawing on separate pages, which can then be pieced together. The **Page Setup** is used to set parameters for tiled printing (see [Page Setup](#)).

Tip: When tile printing, it is usually best to turn the **Print Crop Marks** option on, unless your printer is capable of printing with zero margins. Crop marks enable you to easily trim the edges of the paper so that the sheets can be pasted onto a backing for display.

An example of tiled printing is as follows:

Select **File / Page Setup**.

On the **Paper** page, use the **Printer Paper** field to specify the paper and orientation.

In the **Drawing Sheet Size** fields, specify the size and orientation of the area on which you want to print the drawing. For example, if you want a printout 2 feet tall and 3 feet wide, you would type 2 ft x 3 ft in the list box, and choose the **Landscape** option. Or you can select a standard size.

Click **Fit** to place your drawing on the tiled sheets.

Click **Print** to print your drawing immediately, or click **OK** and print later using **File / Print**.

Another way to tile print is to specify the number of rows (the number of pages from top to bottom) and columns (the number of pages from left to right). This is done on the **Layout** page of the **Page Setup**. You can then return to the **Paper** page to fit your drawing onto the tiled printer paper. TurboCAD will automatically adjust the dimensions of the drawing sheet to accommodate changes in the number of rows and columns.

17.3. Publish as HTML

(Available in Platinum, Professional and Deluxe)

Default UI Menu: File/Publish To HTML

Ribbon UI Menu:



Enables you to export your drawing as an HTML file. You can export the whole drawing or any of its workspaces.

Save: Select the folder path or click the arrow button to browse. The *.htm file will be created in this folder, as well as two subfolders to store the graphic images for Paper and Model Spaces.

Publish: Click to publish the selected workspaces to HTML.

Format: Sets the export format associated with each workspace. Click the column to open the format list. You can choose from *.gif, *.jpg, *.png, *.wrl, and *.dwf.

Note: To display *.wrl and *.dwf formats, you need to have an application that will transfer such drawings to your Internet browser. Otherwise you may receive a warning message.

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Layout: Click the column to open the format list, and select either **Default** or **Custom**.

If **Custom** is selected, you can define the custom layout after clicking **Publish**.

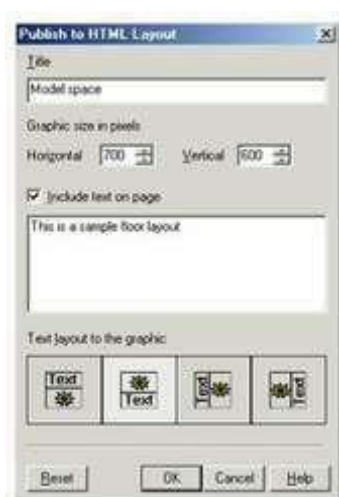
View in Browser: Launches the default Internet browser to show the HTML you created.

All spaces in one page: All selected workspaces will be placed on the same HTML page.

Add full sized image: The graphic will be created with the settings defined in the **Maximum size in pixels** control group of the **Options** dialog. **Options:** Opens the **Publish to HTML Options** window.

Publish to HTML Layout

Sets the HTML page layout. This window appears when publishing HTML files for one or more workspaces that have a **Custom** layout.



Title: Use the default (workspace name) or add a custom title for the graphic.

Graphic size in pixels: Sets the size of the graphic image.

Include text on page: Adds the text you type in the box to the HTML file.

Text layout to the graphic: If text is added, select the text layout position relative to the graphic.

Publish to HTML Options

Parameters for the exported file.

Thumbnail size in pixels: Sets the size of the graphic image. You can change these settings if you use a **Custom** layout.

Maximum size in pixels: Sets the size of the linked graphic image that will be accessible if **Add full sized image** is checked in the **Publish to HTML** window.

Colors: Sets the background and text color. You can define text if you use a **Custom** layout.

17.4. Viewports

(Available in all TurboCAD Variants)



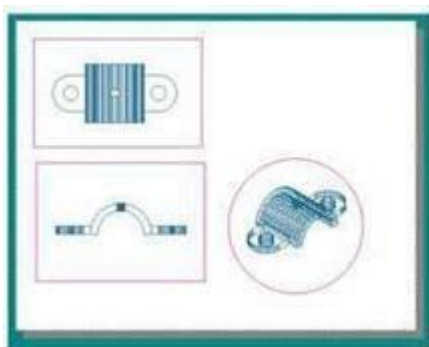
Default UI Menu: View/Viewports

Ribbon UI Menu:

Viewports are used in Paper Space to display one or more views of your model. A viewport has two components: a boundary and the view that it contains. You can only insert views that have already been created - see [Creating a View](#)

Note: If you want to insert standard views (views you do not have to pre-define), see [Drafting Palette-Creating Standard Views](#)

Typically, multiple viewports are created so that you can show several view of the model. If you make changes to your model, any relevant views in the viewports will automatically update.



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Once a viewport is created, you can use 2D tools and annotation tools (**Text** , **Dimensions**, **Hatching**, etc.) to enhance the Paper Space.

You also have the option of saving a viewport's contents as an image. This is handy for rendering objects in viewports, because it is faster to display an image than to render an object. See [Cache Properties](#)

Cache Properties

Options for storing images in viewports.

Use Cache Image: The viewport image will be stored as a picture, so that the image does not have to be regenerated. This is handy for large, rendered images that can take **Use**

Hidden Line Cache: Paper Space viewport processing (redraw, move, zoom) becomes significant faster for large drawings.

time to generate.

Update: If **Manual** is selected, the generated picture will be updated after you select **Update Viewport Cache**.

The remaining options on this page control the quality and size of the generated picture.

Exploding Viewports

(Available only in Platinum)

You can explode a viewport. Wireframe views and Hidden line views are turned into 2D entities. If the viewport is in a rendered (Draft, Quality, advanced) mode the result will be a 2D image.

To work Non-Renderable objects must be turned off in the viewport rendering mode you have selected.

Select the viewport.

Click Explode.

Floating Model Space

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Default UI Menu: View/Viewports/Model Space(Floating), Modify/Model Space(Floating)

Ribbon UI Menu:



Enables you to use Model Space tools within a viewport in Paper Space. This is useful for making minor adjustments to your model within Paper Space, without having to switch to Model Space. Any substantial changes, however, should be done in Model Space.

Select the viewport you want to use to edit the model.

Select **Model Space(Floating)**.

The viewport is outlined in bold. You can now make minor edits using most of the Model Space tools.

Click outside the viewport to finish editing and return to Paper Space.

Warning: If you rotate the view in a floating viewport, the view will **not** return to its original position when you return to Paper Space.

Inserting Viewports

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Default UI Menu: View/Viewports/Viewport

Ribbon UI Menu:

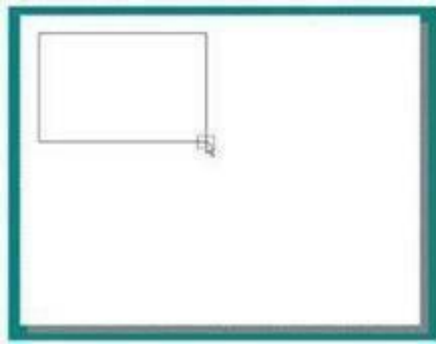
The following example will be used to demonstrate the use of viewports. To try it yourself, open the file **clamp.tcw** in the **Samples\3D Samples** folder.

Switch to Model Space, and select **View / Named View**. Several views have already been defined. (For details on named views, see [Saving Views](#).)

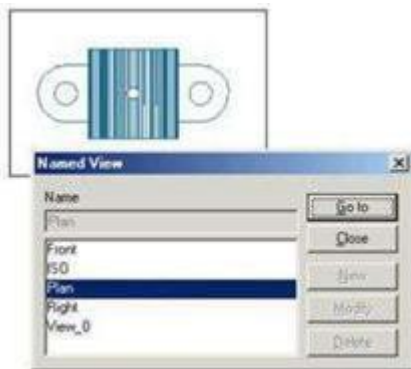
There is already one Paper Space tab, containing several viewports. Right-click on the tab and select **Insert** to create a clean Paper Space.

3. Select **Insert / Viewport** (or use the **Insert Viewport** icon) and define a rectangular boundary in one corner.

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In the **Named View** window, select the desired view (in this case, **Plan**). Clicking **Go To** will display the view without closing the window; this is a good way to check that the view is correct. If you double-click a named view, the view will fill the viewport and the window will close.



5. Insert another viewport containing the **Front** view.

Once a viewport is created, the view inside it can be changed (as well as other parameters) by accessing its **Properties**. See [Viewport Properties](#)

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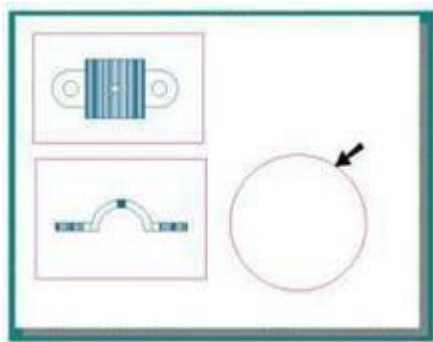
Tip: If you want to create a viewport of the same size as an existing one, you can copy it (see [Copying Objects](#)). You can then open its **Properties** to select another view.

Local menu option:

Shaped Viewport: Enables you to use any closed 2D object as the viewport boundary.

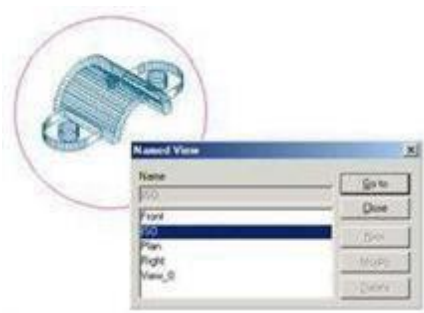


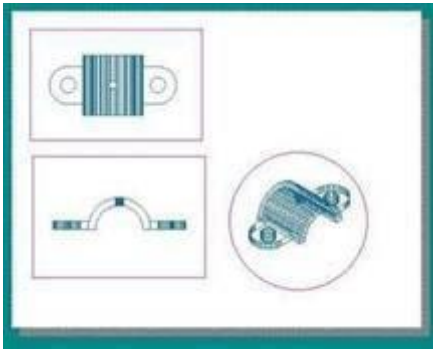
In the Paper Space drawing sheet, use the 2D tools to create the closed boundary, in this case, a circle.



Activate **Insert Viewport** with the **Shaped Viewport** option. Select the closed boundary you just created.

Select a view as before, in this case, ISO.





Overlapping Viewports

Viewports behave like standard 2D objects, in terms of their stacking order and overlapping. You can adjust overlapping viewports using the **Format** menu (**Bring to Front**, **Send to Back**, etc.). See [Stacking Objects](#)

Rotate View with Viewport

You can rotate views by rotating the containing viewport.
First make sure that the Scale is Fixed in the Viewport properties.

Select the viewport.

Rotate the Viewport.

The result is rotated content.

Update Viewport Boundary

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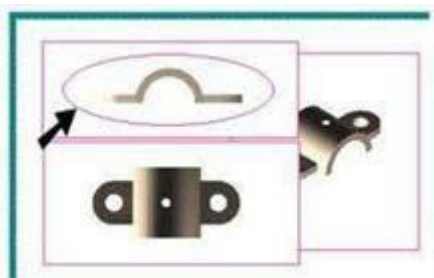
Default UI Menu: View/Viewports/Update Viewport Boundary

Ribbon UI Menu:

A viewport's boundaries can be replaced with any closed 2D object. This is useful in cases like the one shown below, in which the viewport on the right is partially hidden beneath the other two viewports.



Create the closed 2D object you want to use as the new boundary. In this case, the top left viewport boundary will be replaced with an ellipse.



Activate **Update Viewport Boundary**, and select the boundary to change. Then select the new boundary.



If the new boundary is too large or too small for the view inside, you can open the viewport **Properties** and change the **Scale**.

Tip: The conventional way to select the viewport is to click its boundary. It can sometimes be hard to select the viewport if its boundary is invisible. In this case, use F6 or F7 to scroll through the viewports.

Viewport Properties

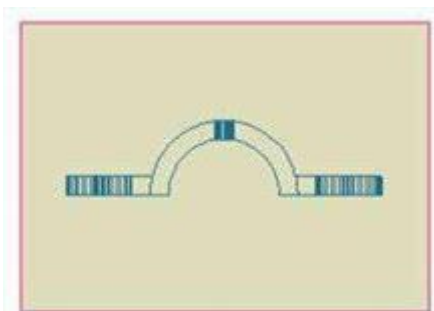
Options for the viewport appearance and the objects it contains.

View Name: Shows all available views. You can use this list to select another view to fill the viewport.

Layer Name: Shows all drawing layers. You can check all layers you want displayed.

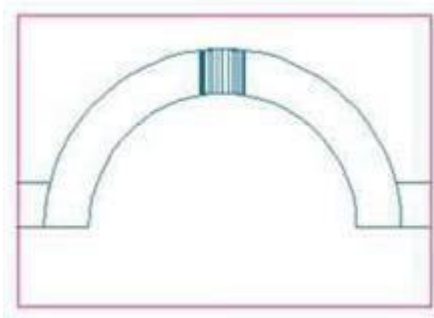
Override Layer Visibility: If this option is on the viewport ignores global layer visibility and uses its own set of visible layers. By default this property is on. But when an DWG/DXF is imported this property is set to "off".

Background: If **Transparent** is not used, you can set a color for the viewport background.



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Scale: Sets the ratio of the viewing scale for the viewport. To set the scale, check the **Fixed** box and then select or enter the scale. The viewport boundaries do not change, so if the scale increases the view, it may extend past the boundaries.



Note: If dimensions are included in the viewport, the dimension is associative with the viewport. This means that the dimension is scaled simultaneously with the scaling of the viewport. To make the dimension non-associative, select both the viewport and the dimension and choose **Drop link** from the local menu.

Rendering: If you want the view inside a viewport to be rendered, check the desired render type on the left side of the **Properties** window.

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Note: If **Hidden Line**, **Draft**, or **Quality** mode is specified, you can explode the viewport. An exploded viewport becomes a Group of graphics or Picture object, which you can verify in the Selection InfoPalette.

Rendering for print only: The rendered view appears only when printed, not on the screen.

Lock Camera: Using Space Change through a viewport allows you to maneuver the view into model space by panning and turning. The Lock Camera setting causes the view to automatically return to the camera location and direction that were in place before using space change.

Lock Scale: Using Space Change through a viewport allows you to maneuver the view into model space through zooming. The Lock Scale setting causes the view to automatically return to the zoom level and view scale that were in place before using space change.

Visible Box: Displays or hides the viewport boundary line.

Tip: When a viewport's boundary is invisible, it can be hard to select the viewport. You can also use F6 to scroll through the viewports while selecting.

Layer Set: See [Layer Sets](#)

Viewport is On: This option allows you to show or hide the content of the viewport. If the option is deselected the content of the viewport will be hidden.

Paper Space ViewPort Explode:

(Available in Platinum only)

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2D Objects contained in Paperspace Viewports can now be exploded.

To Explode:

Create your Viewport with View of 2D Object(s)

Make sure the display mode is set to Hidden Line and that Foreground wireframe option is set to show non-renderable objects

Select the 2D Object(s) in the PaperSpace Viewport, and from the Modify menu, click 'Explode'

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17.5. Woodworking

(Available in Platinum)



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

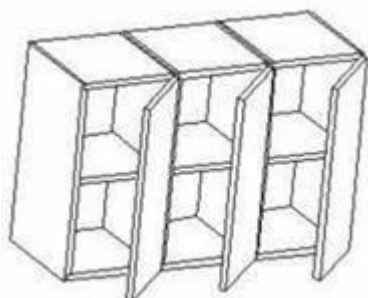
Default UI Menu: Tools/WoodWorking/Create Layout

Ribbon UI Menu:



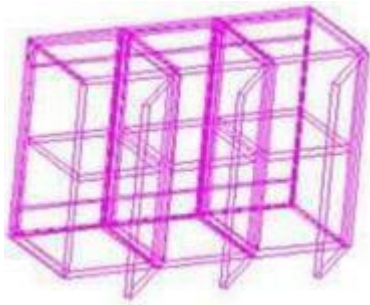
The **Layout** tool can be used for furniture design, and glass or metal cutting. The results show the optimal way to cut rectangular sheets or planks into rectangular parts. Go to **Tools | WoodWorking | Create Layout**

Start with the object or objects you want to lay out. This example uses a parametric symbol from the "Cabinets" folder of the Library palette. (The object does not have to be a symbol or part.)



Activate **Layout**, and click two points of a selection window around the objects. You can skip selecting objects if you are going to manually specify the planks.

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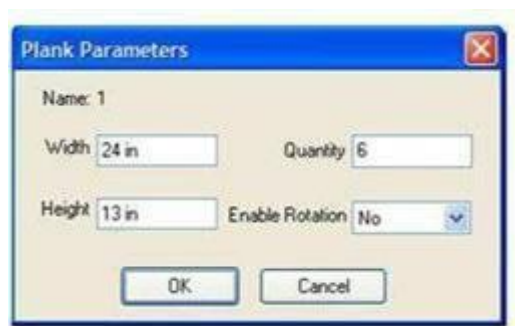
Select **Finish** from the local menu or Inspector Bar.

In the **Layout** window, check all of the pieces you want to include in the layout.

In the table you can:

Double-click on any plank to open the **Plank Parameters** dialog. Where you can specify the properties of each plank individually, Including the quantity.

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Click **Add Plank**. Which will open the Plank Parameters dialog and allow you to manually create and specify planks.

Click **Swap Width and Height**. Which will swap those values for the selected plank.

Select or de-select **Allow Plank Rotation**. This will set the rotation parameter for all of the planks. If plank rotation is allowed the layout of the planks will be optimized to fit the sheet. If it is disallowed you can control how the planks will layout relative to the sheets grain. At the bottom of the window you can specify:

Size of Board: defines the board or sheets from which the pieces will be cut.

Kerf: Defines the width of the cutting blade.

Edge Trim: defines trimming cuts in case the boards have ragged or damaged edges.

Click **Next** .

In the **Format** window, set the following options:



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Draw Planks' Names: If checked, block attributes containing the plank name will be assigned to each piece.

Draw Planks' Sizes: If checked, block attributes containing the dimensions of the planks will be assigned to each piece.

Autosize: Automatically chooses the size of the text.

Layout: Check **Forced VerticalStripes** if you want through cuts, check **Forced Vertical** if you want through cuts along with using **Join Identical Planks**, if you want otherwise, check **Optimal**.

Algorithm options: Check **Join Identical Planks*** if you have more than six identical pieces. And set the maximum number of parts from which pieces will be cut.

Check **Add laid out planks table** to insert a table with all of the plank parameters after you inset the layout.

Click **Create Layout** to begin calculating the layout. (You can interrupt the process by clicking **Stop**.)

8. When the calculation process is complete, click **Finish**.



9. Click in the drawing to bring in the board layout.

If you checked **Add Name or Add Sizes**, you will see the blocks indicated on each piece. Identical pieces have the same block letter.

If you checked **Add laid out planks table**, click in the drawing to place the table with the layout data,

18. Rendering

TurboCAD (Platinum, Professional and Deluxe Only)

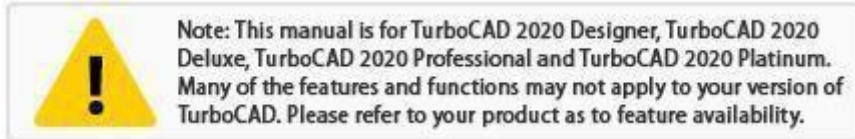
Default UI Menu: Options

Ribbon UI Menu:



One of the most important tools for controlling the view of your model is Render mode. In render mode, all 3D objects are displayed as shaded, or without hidden lines. Higher level rendering also enables you to view materials and textures, showing a realistic view of what your model will actually look like. And in order to create a realistic rendered view, light effects must be added. To further enhance your render, you can assign materials and luminance qualities to objects. You can also create a render scene environment, which affects the background and foreground of your model. Editing of Materials, Luminances, Environments and Render Styles of components is handled in the Render Manager.

18.1. Creating a Rendered View



Default UI Menu: View/Camera/Properties

Ribbon UI Menu:

By default, these tools are also available on the local (right-click) menu. When creating the first render of a model, the easiest way is to select one of the render modes and accept the default lights.

Tip: You can modify the lights by selecting **View / Lights**. For more details, see [Lights](#)

There are several ways to modify and enhance your rendering:

- Adjust render properties (see [Render \(Camera\) Properties](#))

- Modify or add lights (see [Lights](#))

- Define object materials (see [Materials](#))

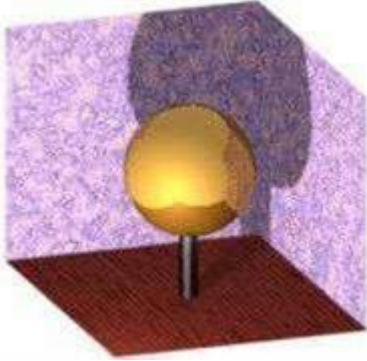
- Create a render scene environment (see [Environments](#))

You can use all 3D tools (as well as most 3D editing tools) in both wireframe and render modes. To render only specific objects, select those objects before selecting the render mode. To return to Wireframe or another render mode, click the relevant icon or use the **View / Camera** menu.

Quality Rendering



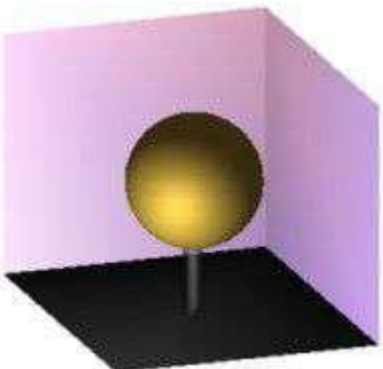
Creates the most photorealistic image with respect to shading, color, reflection, texture, and other qualities. For complex models, this type of rendering can be slow.



Draft Rendering



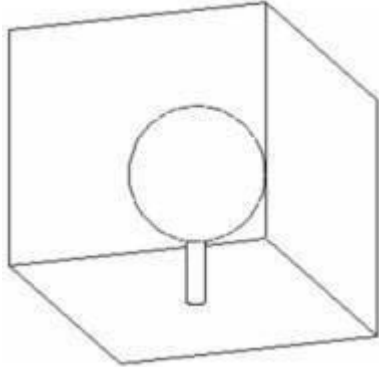
Quickly creates a rendered view. The parameters are more limited than with **Quality** rendering, and the view will not be as detailed. The OpenGL and Lightworks rendering types are available for this mode.



Hidden Line



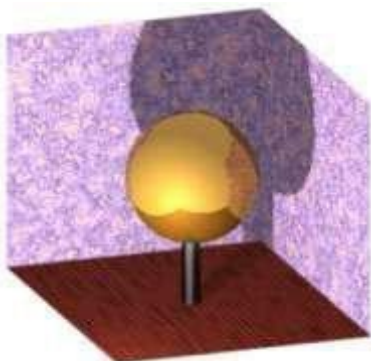
Eliminates the hidden lines from the view.



Advanced Render- RenderStyles



Render Styles are used to give you maximum control over the rendering process. A fair variety of preset values are provided including both raytracing and sketch types of rendering



Note: There are detailed help files available from the Editors for Luminances, Environments, Materials, and Render Styles. These detailed help files describe the function of all settings, and the effect of value changes to parameters in these dialogs

Status Bar Memory Display

Once you initiate any **Rendering** mode for a selected object; the TurboCAD status bar displays the respective memory usage along with the processing time.

For example, if you have just initialized **Quality Rendering**, the status bar located at the lower left corner of the screen displays the value of the memory being used during the process (as shown above) and the total memory used after completion (as shown below). The memory is calculated in Megabytes.



Demo Video:

<https://youtu.be/BgPRwp79O-c>

Creating and Editing Objects in Render Mode

All 3D geometry and most 3D editing tools can be used while in render mode. You can edit objects using **Select Edit** and **Edit Tool**, perform 3D Boolean commands (with the exception of the Boolean operation **3D Slice**) and manipulate the workplane. Tools for creating and modifying 2D geometry cannot be used in render mode. This includes text and tools for inserting and manipulating pictures.

Tip: If you need access to 2D profiles for creating profile objects, you can see them in render mode if you check **Foreground Wireframe / Nonrenderable objects** in the **Camera Properties (View / Camera / Properties)** for the various render modes.

Drag and drop maneuvers are also not available in render mode. Therefore, you cannot drag objects in or out of the Blocks or Symbols Palettes (though the **Format** menu commands are available). You also cannot drag objects into other drawings.

Lightworks Settings

The Lightworks settings under **Options/Lightworks** will effect any render or saved raster file created with render styles. They are on by default. **Transparency Shadow:** Specifies whether a render created with a render style will support shadows cast by transparent objects. This toggles the Shadow Transparency setting for every Luminance which has

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Shadow Transparency turned on.

18.2. Environments

Environments



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Environments

Ribbon UI Menu:

You can specify render options that affect the appearance of the background and foreground, as well as overall tone. These effects can be seen only when using **Quality** render.



Quality rendered view with no Environment

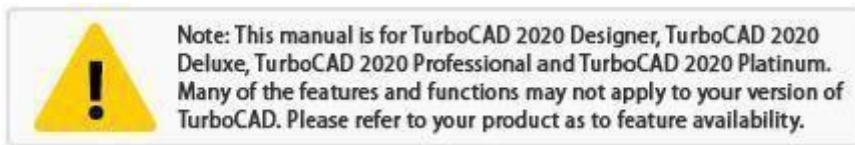
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Render Scene Environment: Background = Clouds, Foreground = Snow

Several environments and categories are provided by default.

You can create a new environment via the **Environments Editor** (see [Managing Environments in the Render Manager](#)).

Loading and Saving Environments



Default UI Menu: Options/Environments/Load

Environments, Options/Environments/Save Environments

Ribbon UI Menu:

Use the categories on the right side of the Save window to specify which environments you want to save. Environment data will be written to a .dat file, located in the *RenderSceneEnv folder. Any new environment you define will be automatically saved in the TurboCAD file as well. To load an environment .dat file, select *Load Environments from the Options / Environments menu, or from the local menu of the palette.

Managing Environments in the Render Manager

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Environments/Edit LightWorks Environments

Ribbon UI Menu:

Editing Environments can also be accessed from the **Render Scene Environment** page of the **Drawing Setup**. You can also access it from the Environments Palette; select the item in the palette, then right-click and select **Edit Environments**. Or double-right-click on the item, or click **Edit Environments**.

Right clicking on tree elements provides a variety of functions. Right clicking on **Environments** allows you to:

- Add Category

- Rebuild Thumbnails for all Environments

- Restore Default Environments

Right clicking on a **Category** allows you to:

- Add SubCategory

- Delete Category

- New Environment

- Rebuild Thumbnails for all Category

If an environment has been copied you will also be allowed to:

- Paste Environment

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Right clicking on a **Environment** allows you to:

Copy Environment Here - adds a copy of the selected environment to the bottom of the current category.

Delete Environment

Rename Environment

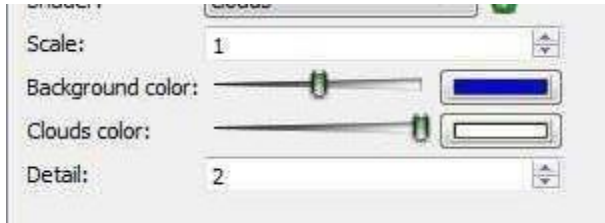
Copy Environment - copies the environment so it can be pasted elsewhere.

Rebuild Thumbnail for Environment

Each environment consists of four categories: Foreground (fog, light scattering, etc.), Background (clouds, planes, images, etc.), Tone (Brighten Up or Scale), and Global Environment (Cube, Panorama, etc.). These categories are found on the left side of the **Render Manager** and each has its own relevant parameters.

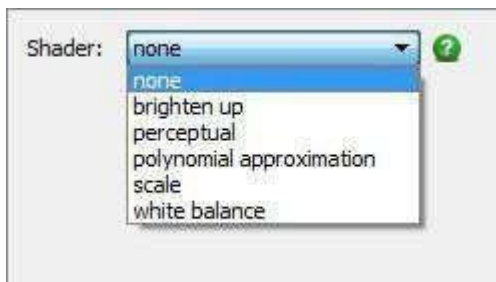
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In the example above, the environment **Ground Fog** has two sets of parameters: **Ground Fog** (fog parameters) and **Graduated** (top and bottom colors). Other environments have different parameters. If you switch to the background environment **Clouds**, you can edit parameters for **Clouds** (color and scale of clouds).



For any environment you can add set of

parameters to the existing defaults. Highlight any **None** page and select a set of parameters from the dropdown list. In this example, you can add snow to a cloudy background.



In the **Preview** section, you can change the shape and orientation of the preview object and desk (the surface beneath the object). Under **Options**, you can toggle the display of the preview object or desk. If **Auto Update** is checked, the preview will update each time a change is made (otherwise you must click the **Preview** button).

The toolbar on the right side of the editor allows you to specify the elements of the

preview.



You can navigate within the preview window using the center mouse button.

Using the Environments Palette

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Tools/Palettes/Environments

Ribbon UI Menu:

Provides easy and fast access to environments

Categories



Set the Environment category.

Find Environment

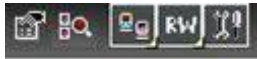
To switch the environment display from thumbnails to list format, click **View Thumbnails** and select a different view.



TurboCAD has LightWorks rendering engine available. The Render Engine Filter allows you to specify which components will be displayed in the palette. ALL sets the palette to show all elements from all installed rendering engines. LW will show only LightWorks elements.

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You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



Right-click on the Environment palette will open a local menu that provides a wide array of options.

Set Environment: Sets the currently selected Environment to the currently selected objects.

Set "None": Sets the Environment to none.

Show Options: Toggle the display of the options on/off.

Show Toolbar: Toggle the display of the toolbar on/off.

View: Allows you to toggle on/off Thumbnails, List, Details

Arrange Icons: Allows you to specify how the icons will be arranged: No arrange, By

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Name, By Name Inversely, By Category, By Category Inversely

Refresh: Refreshs the thumbnails to reflect changes.

Update Thumbnails: regenerates all of the thumbnails in the palette.

Load Environments... : Load Environments from file. [Loading and Saving Environments](#)

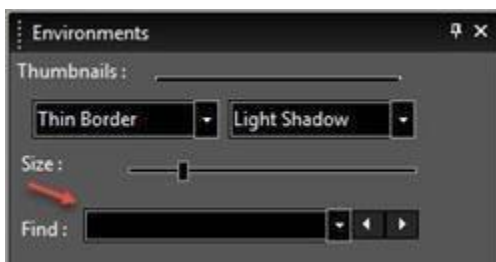
Save Environments... : Saves the current Environments [Loading and Saving Environments](#)

Remove Unassigned Environments: (This option only appears if the category is set to a Drawing Shader Manager) unused environments will be removed from the palette.

Edit LightWorks Materials...: Opens the Render manager. The Render manager is opened to the current material if one is selected. If no material is selected the Render manager opens to the current category.

Search Option:

You can also search a environment in the palette. Type a word in "find" option to search, the first suitable item is highlighted. To browse through the environments, buttons next to the input field highlight and can be used to show the previous or next suitable item. Successful search options are added to the list and saved between sessions.



18.3. Lights

Lights



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Creating realistic rendered views requires realistic light effects. There are several types of lights you can create and position, whose effects can be viewed while rendering. The first time you render a model, you can choose to have a set of default lights created. You can use these, modify them, and/or add your own lights.

Note: You can also create and manipulate lights, and group lights into sets, via the Design Director. See [Design Director-Lights](#)

Controlling Lights

Default UI Menu: View/Lights

Ribbon UI Menu:



Light tab: Shows all of the currently defined lights. An initial list of default lights can be created when you first enter draft or quality render mode. The  symbol indicates whether the light is turned on. The  symbol indicates whether the light indicator will be displayed.

Edit: Click to open the **Properties** window for the selected light. See [Editing Lights](#)

Delete: Deletes the selected light.

New Light: Click to add a new light. For details on the parameters, see [Editing Lights](#)

Show All: Displays all light indicators. **Hide All:** Hides all light indicators.

Restore Defaults: Returns to the initial default light set. New lights will be deleted, and modified lights will be reset to their original settings.

Light Indicators

Each light has an indicator which can be displayed. Indicators appear as 3D icons that indicate the defined light color and brightness. The visibility of indicators can be set in the **Light Properties** window. In the example below, Light 1 is a **Directional**, Light 2 is a **Spot**, Light 3 is **Headlight**, and Light 4 is **Point**.

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When a light is on, the indicator will be dark. When the light is off, the indicator will be light. Light indicators can be selected like any other 3D object. Repositioning and rotating a light indicator will reset the light properties. Light properties can be viewed and edited by selecting the light indicator and opening the **Properties** window.

Editing Lights

For every light in your model, you can edit its properties. You can access the **Properties** window by double-clicking the light indicator (see [Object Properties](#)) or by selecting **Lights** and clicking **Edit** for the desired light. Three pages of a light's properties are specific to lights: **Light**, **OpenGL Specific**, and **LightWorks Specific**.

Light Properties

Parameters of the light quality and location.

Light Type: Type and color of the light, and whether it is on or off.

Power: The intensity of the light, in watts. The higher the setting, the brighter the light. This parameter is relevant only for **Radiosity** rendering. The absolute values (0 through of the color components handled through the **Color** controls determine the absolute value of the light power. For example, if only the Red color is enabled and its

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color value is N ($N=R+G+B$), then the actual value of the light power is determined by the following expression ($P*N/255$) where P is the power value set in the **Power** field.

Tip: Setting wattage values too high can cause objects to have a washed-out appearance.

Spot: Parameters that apply only to **Spot** lights - sources that emit light constrained to a cone.

Cone: The angle of the beam.

Umbra: The angle of the dark center of the spot light.

Beam: The sharpness of the spotlight. A value of 0 means the light will be evenly dispersed. A higher value means more light will be concentrated toward the center of the beam.

Sky: Parameters that apply only to **Sky** lights.

Light distribution: The level of sky clarity, ranging from clear to overcast. For a clear sky, the brightest part of the sky surrounds the sun. For an overcast sky, the brightest part of the sky is straight up.

Min Lod: Minimum level of detail for area source decomposition. Usually this value is between 0.0 and 1.0, closer to 0.0, but values can be greater than 1.0 as well. This parameter determines the initial sampling for lighting calculation. If **Min Lod** is too low, shadow boundaries may not be reproduced correctly. If it is too high, rendering may take a long time. Practical values are between 0.0 and 0.5. **their** values will be reversed.

Temperature: The ambient temperature, in degrees K. Used to set the sky color.

Error Bounds: the shader will perform adaptive sampling, once its initial sampling is complete. The Error bounds parameter then dictates the accuracy of the solution. An value of 0.1 means that the maximum error associated with the illumination of a particular point of interest is less than 10% of the energy associated with the illumination arriving at the point. Values much smaller than 0.1 are not unusual; the default value (0.5) discourages adaptive sampling.

Noise factor: You can add noise to provide patterns which look naturally chaotic and noisy. A value of 0.0 means no noise; a value of 1.0 means maximum noise.

Position: The location or origin of the light.

Direction: For lights with a specific orientation, these are delta values in X, Y, and Z representing the directional vector.

OpenGL Specific Properties

Parameters relevant for OpenGL rendering. The Open GL rendering engine creates views of lower quality than the LightWorks engine, but at a higher speed.

Ambient: The amount of ambient light - light provided equally from all directions.

Diffuse: The quantity of diffuse light reflected by each surface that is lit by the light.

When diffuse light strikes a surface, it is reflected in every direction equally. This makes the surface appear as if it has a rough or matte finish.

Specular: The quantity of specular light reflected by each surface that is lit by the light.

When specular light strikes a surface, it reflects in only one direction. This makes the surface appear as if it has a smooth, mirror-like finish.

Warning: The Open GL settings are cumulative. When all three light component values are set to the maximum, the light source will generate three times the amount of light of a single maximum setting.

LightWorks Specific Properties

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Parameters relevant for LightWorks rendering. The LightWorks rendering engine creates views of high quality, but view creation can take time.

Fall Off: Relevant for **Point** and **Spot** light sources, determines the way in which the light's intensity is attenuated as a function of the distance from the source. By default this parameter is set to "No attenuation", i.e. light intensity is constant. Other available fall-off types include:

Natural Linear: $1/(d+1)$

Natural Quadratic: $1/(d^2+1)$

Linear: $1/d$

Quadratic: $1/d^2$ Where d = the distance from the light source.

Scattering: Creates the effect of light being scattered by a foggy atmosphere.

Shadow: Controls shadow display. Increasing **Resolution** enables you to smooth the shadow edges. You may get some light and shadow effects by specifying **Quality** and **Softness** parameters.

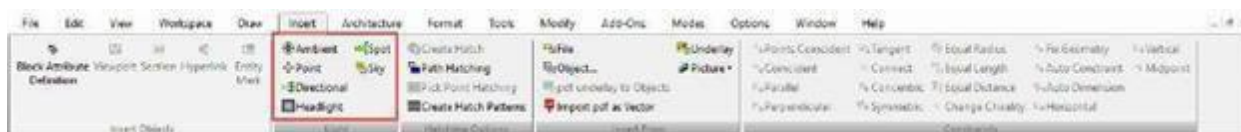
Light Sets

A light set is a group of lights, which can be handy if you want to set identical parameters for multiple lights. Light sets are created and manipulated in the Design Director. See [Design Director-Lights](#)

Creating Lights-Light Types

Default UI Menu: View/Insert Light

Ribbon UI Menu:



You can place the light and define its direction (where applicable) manually on the screen, or enter the coordinates. You can display the **Lights** toolbar by right-clicking in any toolbar area and selecting **Lights**.

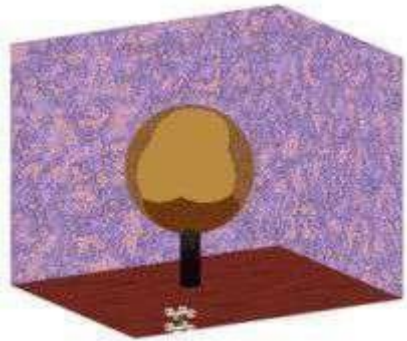


Note: It is important to test changes made in lighting as you proceed. Changing all of the light settings in one step can create unexpected results.

Ambient Light



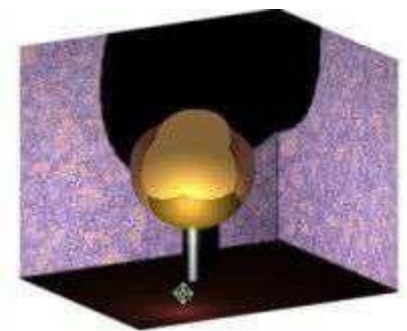
Provides light equally from all directions. Light from an ambient source shines equally on all sides of an object. The **Position** and **Direction** options are turned off because they are irrelevant.



Point Light



Provides light from a selected position. Light from a point source will illuminate each surface of an object depending upon the position of the surface to the light. A surface perpendicular to the light will receive more illumination than a surface that is at an angle. A surface that faces away from the point source will be dark.

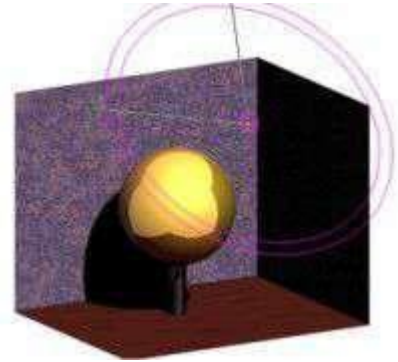


Directional Light



Provides light that has a specific direction. The direction is specified by an origin and a selected point. The length of the directional light is infinite, and does not fade with

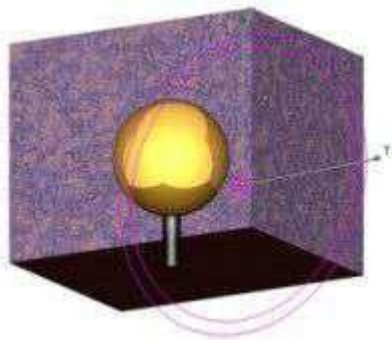
distance.



Headlight



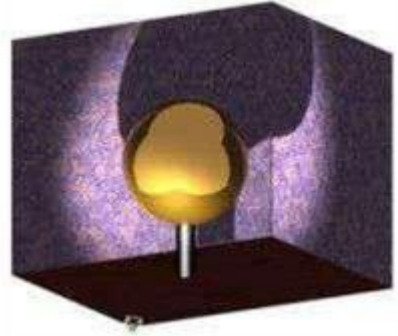
Provides illumination from the camera position. In effect, a headlight is a directional light that uses the camera position as the directional point. As the camera moves, the direction of the light source is updated.



Spot Light



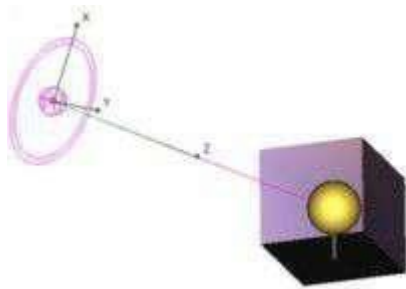
Provides directional light originating from a point and constrained to a cone.



Sky Light



A directional light source that simulates outdoor sunlight.



18.4. Getting Started with TurboLux

TurboLux is a physically based rendering (PBR) solution exploiting state-of-the-art algorithms that simulate the flow of light according to physical equations. Also introduced are a new set of physically based materials with parameters based on the optical properties of metals, glass, matte, glossy, and transparent surfaces. PBR lighting and materials combine together to produce stunning images of photographic quality. Key advantages of using TurboLux include:

- Physically based rendering engines for accurate lighting (four types)
- Support for physical based materials with normal, roughness, and bump channels
- Heterogenous computing utilizing all your CPUs and GPUs
- Caustic support for tracing light paths and sampling methods
- Post process effects for denoising, gamma correction, and tone mapping
- New Gemstone library based on physical properties of gems
- Updated Metal library based on Fresnel properties
- Updated Luminance Editor for creating and editing lighting environments
- More shader options (volumes, bands, mix, cloth, carpaint, glossy coatings)

Key sections in this guide include:

1. *User Interface Components*
2. *Lighting*
3. *Creating and Editing Materials*
4. *Transitioning from Lightworks*

User Interface Components

Accessing the features of TurboLux is accomplished by four key dialog/palettes. These four dialogs include:

1. TurboLux Engine: Rendering engine such as halt time and post processors.
2. TurboLux Scene: Default lighting setups for sun, sky, and HDR maps.
3. Camera: Assign TurboLux rendering for draft or advanced cameras
4. Luminance: Creating predefined lighting groups and environments

Further details of these four key interfaces follows:

TurboLux Engine

All basic settings for TurboLux can be made with this dialog box. Settings include, Platform, Draft Render Settings, Quality Render Settings, Tone Mapping, Gamma Correction, and Logging settings.

More details on each sub-setting can be found in [TurboLux Engine](#) section.

TurboLux Scene

All scene and lighting settings can be changed via TurboLux Scene dialog box available in Drawing Setup.

Light effects for TurboLux can be managed through Drawing Setup/TurboLux Scene/Lighting.

SKY LIGHT:

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ENVIRONMENT MAP:

Details on sub-settings can be found in [TurboLux Scene](#) section.

Camera

TurboLux Render has been added to the Draft Render and Quality Render in Camera Properties menu.

TURBOLUX IN DRAFT RENDER:

TurboLux Rendering Type has been added to the Draft Render section in Camera Properties with RT Path CPU Fixed FPS as a Rendering Mode.

TURBOLUX IN QUALITY RENDER:

TurboLux Rendering Type has been added to the Quality Render in Camera Properties multiple Rendering Modes i.e., RT Path CPU, Path CPU, Tile Path CPU, Bidirectional Path CPU, Bidirectional Path VM CPU, Light CPU.

Luminance

A way to add lights to your drawing is to assign luminance properties to the entire model, or to individual 3D objects. Luminance is one or more lights attached directly to objects, or to the overall WCS or UCS. Luminance differs from lights. Luminance can be accessed through Luminance Palette or Object Properties.

The following TurboLux Luminance Categories have been added to the new version of TurboCAD.

Lighting

Default, quick easy to use lighting in TurboLux Scene

TurboCAD makes it easier to use Lighting in TurboLux Scene quickly with a couple of clicks via Drawing Setup/TurboLux Scene Dialog.

Advanced/custom lighting accessed via Luminance

You will need to describe the luminance editor as it applied to new light types.

Luminance Properties can be accessed through Object Properties/Luminance.

Renderers: Select which render engine you wish to select a material from: All and TurboLux.

Category: Groups of Luminances such as Complex, Environment, Floresc Architec AV, Single, Sky, or the name of an open drawing.

Edit Luminance: The list of the Luminances available in the selected category.

Available In: Shows which of the render engines have definitions for the selected luminance.

Load to render luminance only: If on, the geometry of the luminance object is hidden and only the illumination effects are shown.

Edit... Opens the **Render Manager**.

Editing Luminance

Luminance can be edited via Luminance Palette or Render Manager.

You can select a luminance on the Luminances Palette, right-click and select **Edit TurboLux Luminances**.

Render Manager opens up letting the user edit Luminance.

Right clicking on tree elements provides a variety of functions. Right clicking on Luminances allows you to:

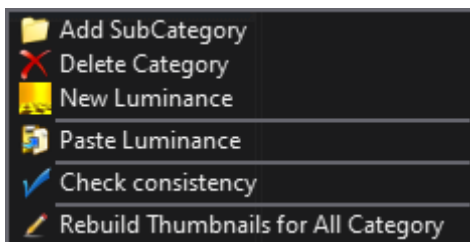
- Add Category
- Rebuild Thumbnails for all Luminances
- Restore Default Luminances

Right clicking on a **Category** allows you to:

- Add SubCategory
- Delete Category
- New Luminance
- Rebuild Thumbnails for all Category

If a luminance has been copied you will also be allowed to:

- Paste Luminance



Right clicking on a Luminance allows you to:

- Copy Luminance Here – adds a copy of the selected luminance to the bottom of the current category.
- Delete Luminance
- Rename Luminance
- Copy Luminance – Copies the luminance so it can be pasted elsewhere.
- Add Light – Adds a light to the selected luminance.
- Rebuild Thumbnail for Luminance

Complex luminances can contain up to five light types; each type is represented by a page on the **Luminance Editor**. For each page, select the desired light type, and edit its parameters as needed. The available parameters depend on the selected light type. In the **Preview** section, you can change the shape and orientation of the preview object and desk (the surface beneath the object). Under **Options**, you can toggle the display of the preview object or desk. If **Auto Update** is checked, the preview will update each time a change is made (otherwise you must click the **Preview** button).

You can navigate within the preview window using the center mouse button.

Light Types

Following are the TurboLux Light Types that can be edited via Render Manager.

- o Sun
- o Sky2
- o Infinite
- o Point
- o Mappoint
- o Sphere
- o Mapsphere
- o Spot
- o Projection
- o Constant Infinite
- o Sharp Distant
- o Distant
- o Laser

Creating and Editing Materials

Drag and Drop Materials

Several material categories with procedural shaders are a part of TurboLux materials libraries.

Materials can be accessed through Render Manger or Materials Palette.

More details on the material types can be found [here](#).

Creating & Editing Materials

Materials can be created or edited via Materials Palette or Render Manager.

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Expanding a folder will show sub-folders and/or names for Materials or Luminance, etc. The list on the left will show tiny thumbnails next to the name (if available) and larger thumbnails in the right-hand pane. If no thumbnails are available, a simple box with a warning sign is used to represent the item.

Clicking on a Material name (or Luminance, etc.) will expand and show the list of shaders used. The right-hand window will split showing the thumbnail in the top pane and the shaders in the lower window. You can double-click the shader to open its properties or click on the shader name in the left-hand list. With the ability to resize the window, scrolling within some property lists can become a thing of the past. The Status Bar at the bottom will display information about the selected item. OK will save your changes and Cancel will exit the editor without saving. As of this writing, the Apply button will only update the thumbnail in the palette, but not the drawing or items in the drawing as before.

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Add Sub Category: Name and add a sub-category to the selected category. It will create a Carpaint "Default" material (this can be renamed). A category has to have a least one material in it.

Delete Category: This deletes the selected category and its objects, plus any sub-categories. There is no Undo.

New Material: Name and create a new material. Initially, this will be a matte white material.

Rebuild Thumbnails for All Category: This rebuilds thumbnails only for materials in the category, not materials in sub-categories (if present), using the last used Preview settings.

Copy Material Here: Place a copy of the selected material in the current category. You will be prompted to create a new name.

Delete Material: This deletes the selected material.

Rename Material: Give the selected material a new name.

Copy Materials: Places a copy of the material on the clipboard.

Use the **Paste Material** command to place it into any other Materials category, except the one it was copied from.

Rebuild Thumbnails for Material: This rebuilds the thumbnail for the selected material using the last used Preview settings.

Advanced Options:

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Displays the settings used when creating a thumbnail preview. Each "settings" window has various options available depending on the setting chosen. Model, scene, and render engine option settings are also available here.

Transitioning from LightWorks

System Requirements

Lightworks was a CPU only rendering engine. TurboLux combines your CPU and GPU maximizing your available hardware. However, to use your GPU, you will need a NVIDIA graphics card that supports OpenCL 1.2 or later or CUDA 10.0 or later. It is recommended you install the latest driver updates before enabling the OpenCL option.

Lighting

As a physically based rendering engine, there is no such concept of ambient light as in Lightworks or OpenGL. Instead, you must always provide a light source. The most common light source to add is a high-definition range image (HDRI). HDR images are often identified into Exterior, Interior, and Studio categories. You can find collections of free HDR images available on the internet.

In addition to using HDR images to replace ambient light, other light sources include material emission (area lights), sun & sky environments, or create custom Luminances.

The View

Lights do not activate for TurboLux. Use the Edit:TurboLux Luminance dialog to add infinite, point, sphere, spot, projection, distant, and laser light sources.

Materials

The material parameters are quite different between Lightworks and TurboLux. TurboLux material parameters are based on physical properties. For example, the metal shader is driven by Fresnel based values and the glass shader is driven by index of refraction and dispersion values. Additionally, many of these parameters have multiple formats. For example, roughness can be defined by a single floating point or a 2D image texture map. Each shader allows you to specify the formats of the parameters.

Post Process Effects: TurboLux has several important options available for post processing an image. These include:

Denoiser: Reduces firefly effect of progress-based sampling.

Tone Mapping: Balances the overall lighting of a scene

Gamma Correction: Increase/decrease scene contrast

Modifications to these effects can impact the final quality and light intensity of your rendered image.

18.5. TurboLux Wiki

(Available in Platinum, Professional, and Deluxe)

TurboLux is a physically based and unbiased rendering engine. Based on state of the art algorithms, TurboLux simulates the flow of light according to physical equations, thus producing realistic images of photographic quality.

Render Configuration

Engines

The choice of engine decides how lights are traced in the scene. TurboLux offers the following engines (integrators):

- **Path** (CPU/OpenCL): Unidirectional pathtracer that casts rays from the camera. Samples the whole film progressively. Supports all AOVs.
- **Tiled Path** (CPU/OpenCL): Unidirectional pathtracer that is almost the same as **Path**, but uses a special sampler which iterates over the image in tiles (this leads to a slightly lower RAM usage). The OpenCL version adapts the number of tiles that are rendered at once to the performance of the compute devices (GPUs/CPU), so the tile size does not matter a lot (if rendering performance is bad due to small tiles, the number of rendered tiles is increased automatically). When the last tile of a pass is reached, it is split among the compute devices. Supports all AOVs.
- **Bidir** (CPU only): Bidirectional pathtracer that casts rays from both camera and light sources. Samples the whole film progressively. Supports only a subset of AOVs: RGB, RGBA, ALPHA, DEPTH and SAMPLECOUNT. It is recommended to combine this engine with the **Metropolis** sampler.

Light Strategy

The light strategy controls how much processing power is spent on each light in the scene. All lights in the scene can be sampled with the same probability (*uniform* light strategy), but if some light sources are much brighter than others, the image will be more noisy because a lot of processing power is spent on weak lights that are not contributing much to the lighting. The solution to this problem are the *power* and *log power* light strategies. They dedicate more processing power to lights that are brighter.

The following light strategies are available:

- Uniform
- Power
- Log Power (default)

The sampling probability of a light can also be influenced with the importance setting of the light source:

Importance: How much processing power to spend on this light source compared to other light sources. Used to scale the light importance computed by the light strategy. For instance, if you set a *uniform* light strategy, a light with a user importance of 2.0 will be sampled 2 times more often than one with 1.0. If you use a *power* light strategy, the user importance will be multiplied by the light power.

Materials

Basic

MATTE

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Matte represents a dull, diffuse surface and is very quick and efficient to render. Matte does not have any specular highlights so it works well for small background objects and test renders. It can represent dull surfaces such as matte plastic, clay, concrete, unpolished stone, and drywall.

MATTE TRANSLUCENT

Matte Translucent allows for transmission of light through the material. It is well suited to plastic lamp covers, where glossy translucent would simply add calculation time, as well as dull polymers, some dull organic materials (such as cheese or banana peels) and some kinds of textiles.

GLOSSY

The Glossy material represents a diffuse surface with a varnish coat. It is well suited to most plastics, lacquered wood, glazed ceramics (porcelain/tile), and wet surfaces. With the "multibounce" option enabled, the material will have a soft appearance, mimicking the scattering by fuzz or hair, making it well suited to rendering many types of cloth.

GLOSSY TRANSLUCENT

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Glossy Translucent represents a diffuse surface with both a transmission component and a varnish coat. This material is well suited to most organic materials, including skin, internal organs, leaves, eyes, milk, ketchup, alien slime, etc. It can also work well for soap, and certain plastics.

Metals

METAL

The metal material represents a generic metal surface. It supports any color as base as well as loading of presets such as aluminum and gold.

Color/Fresnel Texture

This defines the base color of the metal. Metal presets can also be loaded by using the fresnel texture. Fresnel NK files with measured data can be used as well in either the Sopra or Luxpop formats.

MIRROR

Glass

GLASS

The glass material represents a smooth dielectric substance. It works well for materials such glass, crystals, clear liquid, transparent plastic, and gemstones.

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ROUGH GLASS

The Rough Glass material represents a rough dielectric surface. It works well for frosted glass and some kinds of ice.

Others

CAR PAINT

Car Paint is an advanced glossy material, simulating a diffuse surface with multiple reflective coatings. It works well for car paint, obviously, but is also well suited to any sort of "high-tech"-looking painted surface, such as painted aircraft skin, starships, futuristic weapons, etc.

Presets

Presets are available for some quick paint jobs. These are derived from the paper that the car paint material itself was based on.

VELVET

The Velvet material represents a fuzzy, velvety surface. It works well for velvet and other types of cloth. Its appearance can vary wildly with lighting conditions, so you may find it easier to use in a mix with another material such as glossy or cloth.

CLOTH

The Cloth material simulates a woven fabric. The model's parameters are too many to expose individually, but presets are available for typical fabrics such as denim, polyester cloth, wool and silk. The material is based on the PhD thesis ["The Appearance of Woven Cloth"](#) by Piti Irawan. LuxCoreRender's implementation is an adaptation of the code in Wenzel Jakob's [Mitsuba renderer](#), which was again based on code by Irawan.

Important note: the object you are assigning this material to will have to be UV unwrapped for this material to work properly.

Preset Name

Six presets with different stitch patterns are available. The ones below are rendered with the same diffuse and specular colors, but due to the different stitch patterns and specularity of the preset the look of the cloth varies. More advanced cloth materials can be created by using textures instead of solid colors on the diffuse and specular channels, and by mixing with other materials such as velvet or glossy translucent.

Six presets with different stitch patterns are available. The ones below are rendered with the same diffuse and specular colors, but due to the different stitch patterns and specularity of the preset the look of the cloth varies. More advanced cloth materials can be created by using textures instead of solid colors on the diffuse and specular channels, and by mixing with other materials such as velvet or glossy translucent.

MIX

The Mix material is a special material that is used to combine two other materials. Note that it is possible to use one mix material as a component of another mix material. This allows for complex shader combinations.

NULL(TRANSPARENT)

GLOSSY COATING

Textures

Image Textures

Image textures are two dimensional images that are projected on 3D objects. These textures can be used in a variety of material channels, for example color and bump. Typically, the UV mapping of images on 3D objects is handled by the 3D application.

FILE FORMATS

The following formats can be used as textures:

- BMP (uncompressed)
- EXR
- JPG
- PNG
- HDR (Analyze 7.5)
- RAW (consisting of a very simple header (in ascii), then the image data)
- ASC (Ascii)
- INR (Inrimage)
- PPM/PGM (Portable Pixmap)
- PAN (Pandore-5)
- DLM (Matlab ASCII)

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In most cases, PNG files work best for texturing in LuxCoreRender. TIF and PSD (Photoshop) images may work on some platforms.

IMAGE TEXTURE PARAMETERS

When using an image as a texture in any material channel, a couple of parameters are available:

- Gamma - The gamma correction value for the texture. 1.0 will not affect the texture, which is what you want when you need to use the actual RGB values encoded into the texture, such as for a bump map or specular map. If you want a color image texture to appear as the image did on screen, you will want to set this to the screen gamma it was built using, which is probably 2.2.
- Gain - A scaling factor that will be applied to the image.

Procedural Textures

Procedural textures are defined mathematically, which means that the texture value (or color) at any given point is calculated by the program instead of being defined by an existing bitmap image. By combining various textures, very complex materials can be created. At the same time, using procedural textures can be very time effective since UV mapping is not required. Many of the procedural textures are fully three dimensional, which means that regardless of the shape of an object, they will always fit perfectly at edges.

The downside of having LuxCoreRender calculate the texture's values is that your 3D application may not be able to show previews for all textures. More complex procedural textures can also add to the render time.

uxCoreRender's procedural textures are very flexible and covers a wide range of different looks. Not only that, but they can also be combined by using the mix/add/subtract/multiply helper textures for mathematic operations. Most textures generate a noise based on noise size and noise depth. Noise size adjusts the scale of the texture and noise depth adjusts the amount of details. Increasing depth will give finer details but are also slower to render.

Spectrum Textures

The spectrum textures are a set of color textures that define an actual light spectrum rather than an RGB color. They are mainly used for absorption and emission, but can be used on any color field if desired.

Volumes

LuxCoreRender includes a flexible and powerful volume system that controls how light behaves when it moves through objects or the space in-between objects. With it you can create visible sun rays in a dusty room or orange juice or anything in between. Three types of volumes are available to simulate a wide range of volumetric effects.

Volume Types

1. CLEAR VOLUME

Clear is a simple volume that features refraction and absorption. It is primarily meant to be used with clear materials, such as colored glass. It can also be used to add some extra realism to translucent materials with little performance cost. It has two properties, an index of refraction(IOR) and an absorption color.

Absorption Color

This is the main color of the volume. It determines how light is lost(absorbed) as it passes through the volume, making the color darker and more saturated the farther it travels through the volume. Your exporter will have a *Color at Depth* setting which will make the volume match the absorption color at the depth you specify.

Index of Refraction

The refractive properties are defined by the IOR value. Your exporter will have some preset ready for the most common volumes. It's important to note that the only material that will use this IOR setting is glass. The other transmissive materials(null, matte translucent and glossy translucent) do not support refraction.

2. HOMOGENEOUS VOLUME

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The homogeneous volume represents a volume with an even distribution of microscopic particles. It can be used for subsurface scattering (SSS) such as milk or orange juice. It can also be used for atmospheric scattering such as a dusty room with visible sun rays.

Please note that atmospheric scattering is very light. Set Scattering Color to full white and try setting Scattering Scale to 0.001. To simulate it efficiently, you should keep multiscattering disabled in your exporter. This will greatly reduce the noise in the scene. Multiscattering should be used with heavy scattering volumes, such as milk or orange juice. Homogeneous uses the same index of refraction and absorption parameters as the clear volume. In addition to those, it has two extra parameters, scattering color and scattering asymmetry.

Scattering Color

The scattering color determines the color and density of the particles. Higher values are denser. This control can also affect the color of your volume, but that will be determined primarily by the absorption color.

While the scattering color has red, green, and blue values, it is not limited to this range. You can specify values higher than 1 (pure white). In fact, for heavy interior volumes you will often need to use values in the 30-1000 range. To help deal with this wide range of values, your exporter will present you with a set of RGB values and a "scale factor" that the RGB values will be multiplied by.

Scattering asymmetry

The asymmetry value determines the scattering direction. A positive value is primarily forward scattering (light is scattered in the same direction as the ray was traveling) and a negative value is backwards (back towards where the ray came from). Clear particles have a more forward scattering and opaque particles a more backwards scattering. The default value 0 scatters light evenly in all directions.

3. HETEROGENEOUS VOLUME

The heterogeneous volume is the most powerful (and also the slowest) of the three volume types. It has the same functionality as the homogeneous volume, but it also includes ray-marching support. This means it can deal with volumes with varying internal properties, such as clouds, smoke, and ground-hugging fog. It has the same properties as the homogeneous volume, with one extra setting, step size.

The heterogeneous volume can be used as the exterior volume as well, although this comes at a substantial performance cost (even when not using multiscatter) and is not recommended in most cases.

The clear and homogeneous volumes evaluate volume properties only once as it enters the volume. They assume these properties hold constant all the way to other side of the volume. For volumes that are continuous, this is a helpful optimization. However, if the volume is not continuous (such as a cloud), this causes blurred details and the edges of the volume container to become visible. In order to handle changing details, heterogeneous subdivides the volume transit path into several sub-paths (a process known as ray-marching) and evaluates the properties again at each sub-point.

Step Size

Step size defines the spacing between the ray-marching sub paths, in meters. Smaller steps will show greater volume detail, but are slower to render. Setting too large of steps may result in a blocky or noisy appearance of the volume.

Volume Priority

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Volumes have a *priority* setting, which is an integer number. In regions where volumes overlap, the volume with highest priority is chosen and replaces all other volumes.

Lighting

1. Area Lights and Mesh Lights

The most common way of lighting in LuxCoreRender is by using area lights. With this light you are able to simulate a wide range of lighting. It supports common settings such as power and color but also allows to control the spread angle, as well as the shape of the emitted light using IES files.

Another common way is by using mesh lights. This type of light supports the same features as area lights but also features textured lighting. As mesh lights are objects, you can assign an entire object to emit light, such as the tungsten filament in a light bulb or halogen bulb. In general, it's advised to use mesh lights with as few faces as possible to keep render times down.

Objects only emit light in the direction that the normals of the object's faces are pointing.

2. Environment Lighting

Environment lighting encloses the scene entirely, lighting it from all angles. It is normally used for environment mapping, although it can also be used to add a solid color environment. The hemi lamp is used for this type of lighting and it accepts latlong environment maps.

Environment lighting is usually done with HDR(high dynamic range) images but it is possible to use low dynamic range images (like JPG or PNG) as well, but in this case you may want to add some additional lighting to your scene to avoid getting a poorly contrast render. This is not necessary with HDR environment maps, which can be used as the only source of light to create a realistic lighting. Note that the hemi lamp's location in the scene does not matter, only its rotation.

The External Resources page has some links to sites that offer free and quality HDR's. Also a simple web search for "hdri maps" or "hdr maps" will give a lot of results. You can also create your own hdri maps with LuxCoreRender by using the environment/panorama camera.

3. Sun and Sky

The sun and sky light creates a lighting setup that simulates the light of the sun and atmosphere, based on the direction of a sun light in the scene and a parameter named *turbidity* which defines the clearness of the sky. Both the sun angle and the sky clearness influence the color of the light.

Note that the sun lamp is very bright (as it is in real life) so you may have to turn the Gain way down if you have other lights in the scene. The other lights may look like they are not even emitting, but they are actually overpowered by the sun. It is possible to use this light with sun only, sky only, or the default sun+sky. If you want softer shadows try adjusting the relative sun size parameter. Note that the sun lamp's location in the scene does not matter, only its rotation.

4. Point Lights

Point lights are infinitely small light sources that emit light in all directions. In most cases, it's recommended to use only mesh/area lights and environmental lighting for your scene, rather than point or spot lights. This is because the point and spot lights have no real world equivalent as the light source is an infinitely small point. As a result, they tend to produce unnaturally hard lighting and sharp shadows.

5. Spot Lights

Spot lights are infinitely small lights that emit light in a cone shape. Apart from color and location, the width of the cone can be set (as the number of degrees of the center line it will extend), and there is a setting to feather the intensity towards the edge.

As with the point lights, it's recommended to use only mesh/area lights and environmental lighting. This is because the point and spot lights have no real world equivalent as the light source is an infinitely small point.

Projector

Both point lights and spot lights can distribute their light and color according to an image. With spot lights, this results in an effect like something being projected from a diapositive projector or beamer.

6. IES data

With the help of IES files you can get the light distribution pattern of a real light fixture. This will keep render times down as you don't need a reflector or glass material to replicate the pattern. The renders below were lit with the same area light, only with different IES files. Area lights, spot and point lights supports IES files.

18.6. Luminance



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Luminance

Ribbon UI Menu:

Another way to add lights to your drawing is to assign luminance properties to the entire mode, or to individual 3D objects. Luminance is one or more lights attached directly to objects, or to the overall WCS or UCS. Luminance differs from lights (see [Lights](#) in that the range of luminance effects is much larger than those available for drawing lights. An object's luminance can be seen either in **Draft** or **Quality** rendering, as long as an appropriate Lightworks rendering mode is used. Several luminances and categories are provided by default. You can create a new luminance via the **Luminances Editor** (see [Managing Luminances in the Render Manager](#)).

Loading and Saving Luminances



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

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Default UI Menu: Options/Luminance

Ribbon UI Menu:

Use the categories on the right side of the Save window to specify which luminances you want to save. Luminance data will be written to a .dat file, located in the Luminance folder.

Any new luminance you define will be automatically saved in the TurboCAD file as well. To load a luminance .dat file, select Load Luminances from the Options / Luminance menu, or from the local menu of the palette.

Luminance Properties

Renderers: Select which render engine you wish to select a material from: RedSDK, LightWorks, or both.

Category: Groups of Luminances such as complex, single, sky, or the name of an open drawing. See [Model Shaders and Applications Shaders](#)

Material: This field depends on the category. For example, if the **Category** is sky, etc.

Edit Lumninance: The list of the Luminances available in the selected category.

Available In: Shows which of the render engines have definitions for the selected luminance.

Load to render luminance only: If on, the geometry of the luminance object is hidden and only the illumination effects are shown.

Edit... Opens the **Render Manager**.

Managing Luminances in the Render Manager



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Or you can select a luminance on the Luminances Palette, right-click and select **Edit Luminance**.

Right clicking on tree elements provides a variety of functions. Right clicking on **Luminances** allows you to:

- Add Category
- Rebuild Thumbnails for all Luminances
- Restore Default Luminances

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Right clicking on a **Category** allows you to:

Add SubCategory

Delete Category

New Luminance

Rebuild Thumbnails for all Category



If an luminance has been copied you will also be allowed to:

Paste Luminance



Right clicking on a **Luminance** allows you to:

Copy Luminance Here - adds a copy of the selected luminance to the bottom of the current category.

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Delete Luminance

Rename Luminance

Copy Luminance - Copies the luminance so it can be pasted elsewhere.

Add Light - Adds a light to the selected luminance.

Rebuild Thumbnail for Luminance



Complex luminances can contain up to five light types; each type is represented by a page on the **Luminance Editor**. For each page, select the desired light type, and edit its parameters as needed. The available parameters depend on the selected light type. In the **Preview** section, you can change the shape and orientation of the preview object and desk (the surface beneath the object). Under **Options**, you can toggle the display of the preview object or desk. If **Auto Update** is checked, the preview will update each time a change is made (otherwise you must click the **Preview** button).

The toolbar on the right side of the editor allows you to specify the elements of the preview.

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You can navigate within the preview window using the center mouse button.

Now you can set "Direction by Geo-Location" in sky and sun shaders

Using the Luminances Palette

Default UI Menu: Tools/Palettes/Luminances

Ribbon UI Menu:

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Provides easy and fast access to object luminances.

Categories

Active Drawing(Drw1) ▼

Specify the luminance category.

Find Luminance.



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To edit a luminance, select the item in the palette, then right-click and select **Edit Luminance**. Or double-right-click on the item, or click **Edit Luminance**.



To switch the luminance display from thumbnails to list format, click **View Thumbnails** and select a different view.



TurboCAD has LightWorks rendering engine available. The Render Engine Filter allows you to specify which components will be displayed in the palette. ALL sets the palette to show all elements from all installed rendering engines. LW will show only LightWorks elements.

You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



Right-click on the Luminance palette will open a local menu that provides a wide array

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of options.

Set Luminance : Sets the currently selected Luminance to the currently selected objects.

Set "None": Sets the Luminance of the currently selected object to none.

Show Options: Toggle the display of the options on/off.

Show Toolbar: Toggle the display of the toolbar on/off.

View: Allows you to toggle on/off Thumbnails, List, Details

Arrange Icons: Allows you to specify how the icons will be arranged: No arrange, By Name, By Name Inversely, By Category, By Category Inversely **Refresh:** Refreshes the thumbnails to reflect changes.

Update Thumbnails: regenerates all of the thumbnails in the palette.

Load Luminances... : Load Luminances from file. [Loading and Saving Luminances](#)

Save Luminances... : Saves the current Luminances [Loading and Saving Luminances](#)

Remove Unassigned Luminances: (This option only appears if the category is set to a Drawing Shader Manager) unused luminances will be removed from the palette.

Edit LightWorks Luminances...: Opens the Render manager. The Render manager is opened to the current Luminance if one is selected. If no Luminance is selected the Render manager opens to the current category.

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Search Option:

You can also search a luminance in the palette. Type a word in "find" option to search, the first suitable item is highlighted. To browse through the luminances, buttons next to the input field highlight and can be used to show the previous or next suitable item. Successful search options are added to the list and saved between sessions.



18.7. Materials

Default UI Menu: Options/Materials

Ribbon UI Menu:

By applying materials to a 3D object, you can obtain a more realistic view of the model during a render. An object's materials can be seen in **Quality** render mode, and in certain types of **Draft** rendering. Several materials and categories are provided by default. You can create a new material via the **Render Manager** (E.g. [Editing Lightworks Material](#)).

Differences between Lightworks and RedSDK materials

Representation, storage structures and all other elements are completely identical for Lightworks and RedSDK Luminances, Environments and Render Styles. At the same time the two sets of materials have a number of differences, which are described in more detail below:

All information about Lightworks materials is stored in a special text file. Before the rendering process initiate a special code appends the Lightworks material definition into this text file. All information about the RedSDK materials is stored in *.RED files, and only a few special master properties are available for user to change. Therefore, RedSDK material stored in TurboCAD, are held as small text file, which contains the file path to the corresponding RED file. Without *.RED file materials will not work in RedSDK.

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In the case of Lightworks if a shader has a texture, the text representation of this shader contains only the file path to the image file. Only immediately before the rendering process is the image file loaded. In the case of RedSDK the RED file contains not only path to the texture, but the embedded image file itself. the path to the image file may not be valid, but the loaded texture inside RED file is always valid.

RED files can be stored by the following paths:

- Path to propin.dat (Luminance.dat, sceneenv.dat or RenderStyles.dat) / RedSDK

- Path to Default.dat/RedSDK (for materials, luminances, environments or render styles)

- Near active drawing file

In all these cases a relative path to the RED file will be used. A user can use any other custom RED file, however, In this case, an absolute path will be used.

When a user deletes a material, the text representation of the material also is deleted. In the case of the RedSDK material the *.RED file also is deleted except in cases when the RED file was custom or default (i.e. installed together with TurboCAD). In the case of a custom file you cannot delete the *.RED file because you did not create it.

A user can create a new RedSDK material in two main ways.

- Create from a custom RED file.

- Create it from a special copy of the default RED file. Therefore, when the first new material is created in the Render Manager a new *.RED file is created as a copy of the default RED file.

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In the case of Lightworks, a text representation of the material is included in the *.tcw file. That is why it is possible to send just a *.tcw file to another user and that user will be able to reproduce a render.

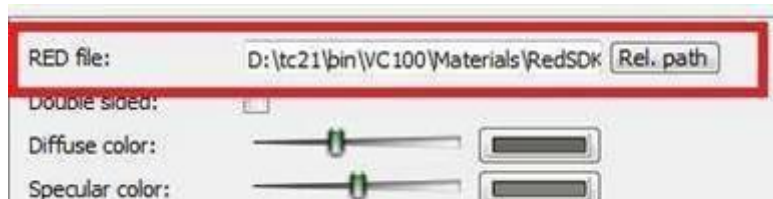
As mentioned, in the case of RedSDK for the *.RED file is needed for rendering. That is why it is necessary to use the e-pack functionality in the case of exchanging files that depend upon RedSDK.

There is a RedSDK material specific UI control which displays the path to the *.RED.

This control is used for these cases:

When the RedSDK material contains relative path to the **.RED file**. Pressing the Abs. path* button user can see the absolute path which is used for loading of the RED file.

By pressing the **Rel. path** button the user can return to the state where the relative path is displayed.



Using the Materials Palette

Default UI Menu: Tools/Palettes/Materials

Ribbon UI Menu:

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Provides easy and fast access to object materials.

Categories

Active Drawing(Drw1) ▼

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All materials that have been defined are displayed here, and can be assigned to objects and edited.

Find Material of Selection.



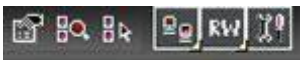
To edit a material, select the item in the palette, then right-click and select **Edit Material**. Or double-right-click on the item, or click **Edit Material**.



You can use **Select by material** to select the object by applied material. Select the material in the materials palette and click **Select by material** the object having the material applied will be selected. Make sure that Active Drawing is selected in the categories.



To switch the luminance display from thumbnails to list format, click **View Thumbnails** and select a different view.



TurboCAD has Lightworks rendering engine available. The Render Engine Filter allows you to specify which components will be displayed in the palette. ALL sets the palette to show all elements from all installed rendering engines. LW will show only LightWorks

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elements.

You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



Right-click on the Material palette will open a local menu that provides a wide array of options.

Set Material: Sets the currently selected material to the currently selected objects.

Set "None": Sets the material of the currently selected object to none.

Select by Material: Select all objects that have the material currently selected in the palette.

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Find material of Selection: In the Palette locate the material of the currently selected object.

Show Options: Toggle the display of the options on/off.

Show Toolbar: Toggle the display of the toolbar on/off.

View: Allows you to toggle on/off Thumbnails, List, Details

Arrange Icons: Allows you to specify how the icons will be arranged: No arrange, By Name, By Name Inversely, By Category, By Category Inversely **Refresh:** Refreshes the thumbnails to reflect changes.

Update Thumbnails: regenerates all of the thumbnails in the palette.

Save Materials: Saves the current materials. See [Loading and Saving Materials](#)

Remove Unassigned materials: (This option only appears if the category is set to a Drawing Shader Manager) unused materials will be removed from the palette.

Edit LightWorks Materials...: Opens the Render manager. The Render manager is opened to the current material if one is selected. If no material is selected the Render manager opens to the current category.

Search Option:

You can also search a material in the palette. Type a word in "find" option to search, the first suitable item is highlighted. To browse through the materials, buttons next to the input field highlight and can be used to show the previous or next suitable item. Successful search options are added to the list and saved between sessions.

Dragging and Dropping Materials

Another way to apply materials to object is to select a material in the Material palette, then drag-and-drop it onto the desired object.

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Drag on Facet

Selecting the **Drag on facet** setting in the local menu of the **Material** palette will allow you to drag a material to a single facet of an object.

Lightworks Material Properties



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You can access material properties via the **Material** page of an object's **Properties** window (see [Object Properties](#)), or by using the Materials Palette.

Renderers: Select which render engine you wish to select a material from: RedSDK, Lightworks, or both.

Category: Groups of materials such as glass, metals, stone, and wood, or the name of

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an open drawing. See [Model Shaders and Applications Shaders](#)

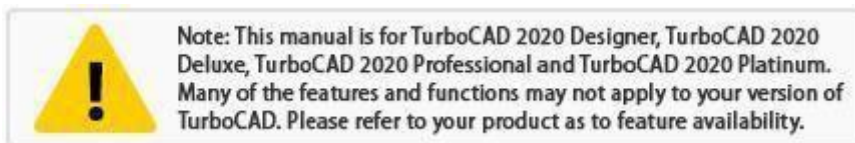
Material: This field depends on the category. For example, if the **Category** is Metals, the materials include brass, iron, aluminum, etc.

Material: The list of the materials available in the selected category.

Available In: Shows which of the render engines have definitions for the selected material.

Edit... Opens the **Render Manager**. See [Materials](#)

Loading and Saving Materials



Default UI Menu: Options/Materials/Load Materials, Options/Materials/Save Materials

Ribbon UI Menu:

Use the categories on the right side of the **Save** window to specify which materials you want to save. Material data will be written to a **.mat** file, located in the ***Materials** folder. Any new material you define will be automatically saved in the TurboCAD file as well. To load a materials **.mat** file, select ***Load Materials** from the **Options / Materials** menu, or from the local menu of the palette.

Managing Materials in the Render Manager

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Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Or you can select a luminance on the Luminances Palette, right-click and select **Edit Material**.

Right clicking on tree elements provides a variety of functions. Right clicking on **Materials** allows you to:

- Add Category

- Rebuild Thumbnails for all Materials

- Restore Default Materials

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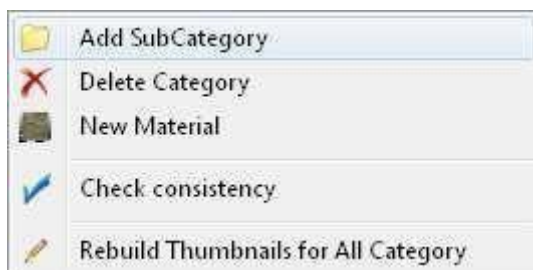
Right clicking on a **Category** allows you to:

Add SubCategory

Delete Category

New Materials

Rebuild Thumbnails for all Category



If an luminance has been copied you will also be allowed to:

Paste Materials

Right clicking on a **Luminance** allows you to:

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Copy Materials Here - adds a copy of the selected luminance to the bottom of the current category.

Delete Materials

Rename Materials

Copy Materials - Copies the luminance so it can be pasted elsewhere.

Add Light - Adds a light to the selected Materials.

Rebuild Thumbnail for Materials

The toolbar on the Left side of the editor allows you to specify the elements of the preview.

You can navigate within the preview window using the center mouse button.

18.8. Model Shaders and Applications Shaders

TurboCAD divides its shaders into two database categories: Model Shaders and Applications Shaders.

DEFINITION

In TurboCAD a shader is a component which is use to define how elements in the drawing are to be rendered. These elements inculde: Materials, Luminances, Environments and Render Styles.

Application Shaders manager: This is the shaders database which contains all shaders sets, which are connected with TurboCAD as a whole. **Model Shaders manager:** This is the shaders database which contains the shaders sets, which are connected with the specific drawings.

Initially, in a new drawing, there are no elements in the Applications Shader database. Elements are added as they are applied or created. Once they are in a drawing they wil be listed in that drawings shaders. Thes can be seen in the drop-down combo boxes will have the following format:

Under Categories:

drawing	Active Drawing	// Materials in the active
	Drawing name1	// "In model" Shaders manager
	Drawing name2	
previous TC		
	All TC database	// Equal to "All Categories" in
	Brick	
	Circuit	
		
	Wood floor	

This means that when Active Drawing is selected (or any other drawing) that only those materials in that drawing will be listed.

As it is possible to see from the described format, in the first place is always the Shaders manager for the active drawing. After that follow the others "In model" shaders managers (for all opened drawings). Next comes the special non-clickable separator. After that follows the "All TC database", which is equal to "All Categories" in previous TC versions. Finally, at end, the list of all categories of the **Application Shaders manager** is present.

N.B. Here it is important to note that shaders manager for the active drawing is always present in the combo box in comparison with other drawings shaders managers, which are in the combo box only if they have at least one shaders set.

Saving the shaders sets thumbnails in the TCW file

The Active Drawing allows you to store all render attributes in a TCW file and be independent from the application's databases. However, there is a problem: where to store the preview images of the shaders sets (which was created in the Render Manager)?

To resolve this issue there is an option 'Save Render Manager preview images 'in the TCW 'Save As'\Setup' dialog (Fig. 5). This option is on by default.

This file was saved with preview images

The increase in size of the TCW file size depends on the chosen size of the preview window in the Render Manager. In the example above, the preview window size is 240x240 pixels. The file size without preview images is equal to 510 KB, with preview images - 580 KB. The small size of images is achieved due to pre-compressing of the images to JPEG format.

Copying shaders sets to the Application Shaders Manager

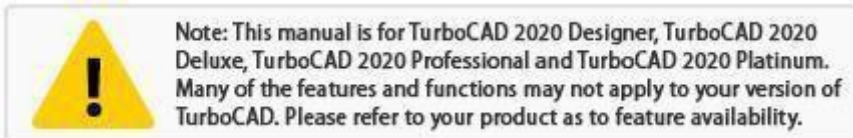
Sometimes it will be desirable to copy shaders from a drawing into the Application Shaders Manager. To resolve this issue there is an option on the 'Preferences' page.

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By default this option is off. When this option is active, when a file is opened all shaders sets will not only be added to the Model Shaders manager but also will be copied to the Application Shaders Manager.

It is not advisable to keep this option always active because if it is the Application Shaders Manager will gradually turn into a "trashbin". So the best way to use this option is to turn it on before opening the desired file and then turn it off again.

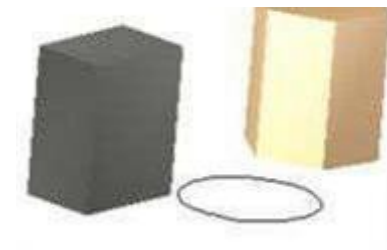
18.9. Render (Camera) Properties



The properties for all render modes contain the following parameters:

Foreground Wireframe: Controls wireframe display in render mode. These parameters can be helpful when objects are hidden by other objects, or when not all objects are rendered.

Nonrenderable Objects: If checked, objects that cannot be rendered are displayed as wireframes during rendering. Nonrenderable objects include most 2D objects, though some 2D objects can be rendered, such as images.



If not checked, these objects will not be seen.



Renderable Objects: Displays the contour lines of renderable objects.

Renderable objects include 3D objects and certain 2D objects such as images.

Wire frame Properties

Parameters for wireframe display.

Rendering type: The graphics engine that will be used to create the render. The differences between the options relate to the different libraries used for object representation.

OpenGL: All contour lines are displayed.

LightWorks: All contour lines are displayed, but color is not displayed.

Native Wireframe: The standard wireframe mode. For fast drawing, only main contour lines are displayed for ACIS objects.

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Rendering mode: The available modes depend on the rendering type:

Wireframe: All contour lines are displayed.

Wire OpenGL: One-color wireframe rendering.

Visualize: Visualize wireframe mode.

Wire Material Color OpenGL: Adds color highlighting in accordance with LightWorks algorithms.

Native Wireframe: The standard wireframe mode. For fast drawing, only the main contour lines are displayed for ACIS objects.

Hidden Line Properties

Parameters for **Hidden Line** rendering

Rendering type: The graphics engine that will be used to create the render.

Hidden Line: The standard display mode - transparent rendering.

LightWorks: A for of Non-transparent rendering Hidden lines are completely hidden.

Rendering mode: The following modes are available for both rendering types:

Hidden Line: Displays only the contour. Display of intersections and hidden lines depends on the options on the lower part of window. Colors are displayed.

Hidden OpenGL: Non-transparent wireframe render, suppresses hidden lines. Colors are not displayed.

Hidden Material Color OpenGL: Non-transparent wireframe rendering, with suppressed hidden lines. Materials and lighting are also rendered.

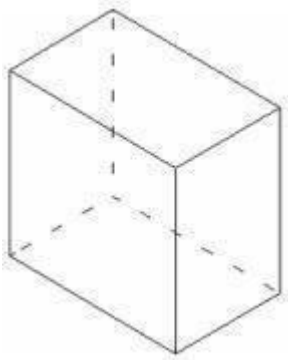
Visualize: Hidden Line as dashed line.

Always clip object behind the camera: Creates cross-sections by placing a camera

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inside the object. This means that the object will be "cut" by a plane perpendicular to the camera view direction.

Draw hidden line as dashed: Displays all hidden lines as dashed.



Set custom hidden line: Enables you to specify a custom line for hidden lines. Click

Edit to open the Pen Properties window (see [Properties Window](#))

Draw Intersections: Displays the lines of object intersections.

Draft Render Properties

Parameters for **Draft** rendering

Rendering Type: The graphics engine that will be used to create the render.

OpenGL: Allows quick rendering in order to gauge the shape and placement of objects.

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Because of its speed, OpenGL is best suited for walk-throughs to quickly review the structure of the drawing, but not to show all details. For example, an object's pen color is used rather than its defined material.

LightWorks: Used to create higher quality renders, but can take time.

Rendering Mode: The available modes depend on the rendering type:

Flat: LightWorks' fastest method, produces images quickly but with low image quality. Each facet is painted uniformly with an average color.

Gouraud: a method used in graphics to produce continuous shading of surfaces.

Phong: interpolates surface normals across rasterized polygons.

Flat OpenGL: LightWorks OpenGL's fastest but lowest-quality method, does not support smoothing.

Smooth OpenGL: The most commonly-used LightWorks OpenGL method, and the default for **Draft** rendering. Speed and quality are optimized.

Visualize: "Suppress Hidden Line" and "Draw hidden line as dashed" support.

Gouraud Shaded - Smooth shaded display. Colors interpolated between vertices.

Flat Shaded - Faceted display. One color per face.

Quality Render Properties

Parameters for **Quality** rendering

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Rendering Type: LightWorks is available

Rendering Mode:

Preview: lowest quality.

Full: creates rendered images faster than **Raytrace** options, but of lower quality.

Raytrace Preview: Similar to Raytrace but of lower quality.

Raytrace Full: creates very high quality renderings at the cost of speed. **Radiosity:** Generates light distribution throughout the scene; the resolution is independent of any particular viewpoint. The radiosity algorithm is intended for solving the lighting problem, not the visibility problem. Once the light distribution has been generated, images can be rendered using standard visibility and shading algorithms. Radiosity is light-dependant and requires careful preparation to create the highest quality imaging.

Extended Parameters: Available if **Radiosity** is used.

Render Mode: Sets the render method **Radiosity** will use to draw the image after making calculations.

Iterations: The level of refinement in the **Radiosity** calculations. The higher the number, the finer the results. A typical value is between 20 and 30.

Quality: The integrated parameter for improving the quality of an object's shadow displaying. Each quality value corresponds to the default set of radiosity parameters. Two of them, **Max Area** and **Min Area** can be adjusted manually to smooth the object shadow.

Render Styles Properties

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Parameters for **Render Styles** rendering

Category: Allows your to select the category or the render style. The following Categories are provided by default: Depth of field, Final Gather (with Ambient Occlusion and Weather subcategories), Ray Traced, and Sketch.

Render Style: Allows you to select the render style from the selected category.

Edit Render Style: Opens the Render Style editor for modifying existing render styles and creating new render styles. The editor will open to the currently selected style.

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18.10. Render Manager

Default UI Menu: Tools/Render Manager

Ribbon UI Menu:



The Render Manager is opened whenever you click an option to Edit: Materials, Environments, Render Styles, or Luminances.

Expanding a folder will show sub-folders and/or names for Materials or Luminance, etc. The list on the left will show tiny thumbnails next to the name (if available) and larger thumbnails in the right-hand pane. If no thumb's are available, a simple box with a warning sign is used to represent the item.

Clicking on a Material name (or Luminance, etc.) will expand and show the list of shaders used. The right-hand window will split showing the thumbnail in the top pane and the shaders in the lower window. You can double-click the shader to open its properties or click on the shader name in the left-hand list. With the ability to resize the window, scrolling within some property lists can become a thing of the past. The Status Bar at the bottom will display information about the selected item. OK will save your

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changes and Cancel will exit the editor without saving. As of this writing, the Apply button will only update the thumbnail in the palette, but not the drawing or items in the drawing as before.

Tree Context Menus

Each menu is the same in Materials, Luminance, etc., but Materials is used below just as an example.

- **Add SubCategory:** Name and add a sub-category to the selected category. It will create a matte white "Default" material (this can be renamed). A category has to have a least one material in it.
- **Delete Category:** This deletes the selected category and its objects, plus any sub-categories. There is no Undo.
- **New Material:** Name and create a new material. Initially, this will be a matte white material.
- **Rebuild Thumbnails for All Category:** This rebuilds thumbnails only for materials in the category, not materials in sub-categories (if present), using the last used

Preview settings.

- **Copy Material Here:** Place a copy of the selected material in the current category. You will be prompted to create a new name.
- **Delete Material:** This deletes the selected material.
- **Rename Material:** Give the selected material a new name.

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Copy Materials: Places a copy of the material on the clipboard. Use the **Paste Material** command to place it into any other Materials category, except the one it was copied from.

Rebuild Thumbnails for Material: This rebuilds the thumbnail for the selected material using the last used Preview settings.

Toolbars

Standard toolbar:

- **Back** and **Forward** arrows allow you to retrace your steps through the editor.
- **Up:** will move you up to the next Category level.
- **View Tree:** will show/hide the left-hand pane.
- **Views:** gives you size choices on how you want to view the Category thumbnails.
- **Sort:** There are 2 **Sort** options available; Name: Ascending and Name: Descending. The Sort option only affects the lists in the right hand pane. It will not sort the list on the left.

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Preview-Render toolbar:

- **Options:** Opens the Advanced Options dialog.
- **Units:** Specify the units to be used in the thumbnail model.
- **Refresh:** Use this to re-render the thumbnail, after making changes to settings (if **Auto Update** is turned Off in Advanced Options).
- **Stop:** Stops the rendering process of the thumbnail.

Models toolbar:

- **Cube:** creates a thumbnail containing a cube with rounded edges.
- **Sphere:** creates a thumbnail containing a sphere.
- **Cylinder:** creates a thumbnail containing a cylindrical object.
- **Presentation:** creates a thumbnail with the Presentation icon.
- **Selected:** creates a thumbnail using a selected object in the drawing.
- **Compound Selection:** creates a thumbnail using several selected objects in the drawing.
- **No Model:** no model is used in the thumbnail. You might use this when creating a Environment thumbnail.

Scenes toolbar:

- **Floor:** inserts a limited size grid patterned floor into the scene
- **Room:** creates a thumbnail inside an enclosed area containing a grid pattern.
- **Studio:** creates a thumbnail inside a Ray Cube environment.
- **Studio with enhanced lights:** creates a thumbnail inside a Ray Cube environment with improved lighting.
- **No scene:** creates a thumbnail without any scenery present; just a Graduated background.

Views toolbar:

- **Default:** a Front view with perspective (25°), looking down at about 30°.
- **Dimetric:** a south-east dimetric type of view, but the viewers position is lower than normal.
- **Active View:** uses the camera settings used by the current view in the drawing.
- **Custom:** Enter coordinates for your viewing position (From), to where you are

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looking (To). In most cases, set the UP Vector (Up) value to: 0,0,1. For a "Plan/Top" type view, use: 0,1,0 for the UP Vector. You may need to click on the green Refresh icon after closing the Camera Properties dialog (seen below).

- **Top:** creates a Top view.
- **Front:** creates a Front view.
- **Back:** creates a Back view.
- **Right:** creates a Right view.
- **Left:** creates a Left view.

You can rotate, pan and zoom within the thumbnail window.

- **Rotation:** center mouse button / hold down the wheel.
- **Pan:** Shift + center mouse button
- **Zoom:** roll the mouse wheel.

Right-clicking on a thumbnail will give you two options: "Open Containing Folder" or "Properties". "Open Containing Folder" will open Windows Explorer in the folder containing the thumbnail. You can delete the image file, which will remove the thumbnail from the editor. "Properties" opens the standard Windows Properties dialog for the thumbnail.

Advanced Options

Displays the settings used when creating a thumbnail preview. Each "settings" window has various options available depending on the setting chosen.

Preview options:

- **Window Size:** options display the current size of the preview pane. These sizes will be used when creating a saved thumbnail. Currently, the numbers can't be edited in the dialog. Resizing the upper right pane will change the numbers.
- **Auto Update:** is on by default. As changes are made to settings, the thumbnail will update; just changing a number will trigger an update. When this is turned off, use the **Refresh** button on the **Render** toolbar to update the thumbnail.

Model options:

- **Size:** Defines the size basic models (Box, Sphere, etc.) used in thumbnails. The default size is 39.370079 inches or 1 meter.
- **Material:** Specify the material settings for the basic models. Not available in "Material Settings".
- **Double-sided:** Controls the visibility for the front and back sides.

Scene options:

- **Desk Material:** A pre-defined Grid is the default for the desk or floor material or choose a different material from the list. Available in all settings.
- **Luminance:** A pre-defined setting for Materials and Render Styles and a different pre-defined setting for Environments. Not available in "Luminance Settings".
- **Environment:** A pre-defined "Ray Cube Gradient" setting is used or choose from the list. Not available in "Environment Settings".
- **Units:** "By Drawing" is the default or choose from the list.

Render engine options: Choose a style from the list. Not available in "Render Style Settings".

- **Advanced Render — Render Style:** On by Default
- **Quality Render — Quality Render Mode:** If activated, use "Raytrace Full" or select from the list.

Editor Window

Displays the various options for the selected shader. Clicking on the green question mark icon will open this Help file. Some fields will have an cycle icon. This allows you to choose the type of field you want to use; spinner (absolute values) or slider (percentage

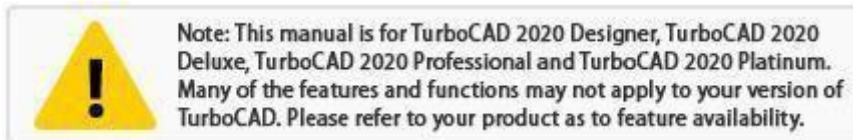
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values).

The **Is Custom** column allows you to checkmark only those fields you may want to make frequent changes to.

When you go to edit the material, it will open at the "Custom" shader setting and only the checked fields in the various categories will be displayed together on one page. No need to jump back-n-forth between shader pages.

18.11. Render Styles



Advanced series of settings allowing you to create stylistic renders.

Default UI Menu: Options/Rendering Styles

Ribbon UI Menu:

Using the Advanced Render Styles Palette

Default UI Menu: Tools/Palettes/Advanced Rendering Styles

Ribbon UI Menu:

If no rendering style is selected the Render manager opens to the current category. Provides easy and fast access to render styles.

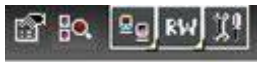
Categories

Active Drawing(Drw1) ▼

Specify the Render Style category.

Find Rendering Style

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To switch the render style display from thumbnails to list format, click **View Thumbnails** and select a different view.



TurboCAD has Lightworks rendering engines available. The Render Engine Filter allows you to specify which components will be displayed in the palette. ALL sets the palette to show all elements from all installed rendering engines. LW will show only LightWorks elements.

You can hide or show the palette fields and toolbars by clicking **Show / Hide Options**.



Right-click on the Advanced Render Styles palette will open a local menu that provides a wide array of options.

Set Render Style: Sets the currently selected Render style to the currently selected objects.

Set "None": Sets the render Style of the currently selected object to none.

Show Options: Toggle the display of the options on/off.

Show Toolbar: Toggle the display of the toolbar on/off.

View: Allows you to toggle on/off Thumbnails, List, Details

Arrange Icons: Allows you to specify how the icons will be arranged: No arrange, By Name, By Name Inversely, By Category, By Category Inversely **Refresh:** Refreshes the thumbnails to reflect changes.

Update Thumbnails: regenerates all of the thumbnails in the palette.

Load Rendering Styles... : Load render Styles from file. [Loading and Saving Render Styles](#)

Save Rendering Styles... : Saves the current render Styles [Loading and Saving Render Styles](#)

Remove Unassigned Rendering Styles: (This option only appears if the category is set to a Drawing Shader Manager) unused Render Styles will be removed from the palette.

Edit LightWorks Rendering styles... : Opens the Render manager. The Render manager is opened to the current material if one is selected. If no rendering style is selected the Render manager opens to the current category.

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You can also search a render style in the palette. Type a word in "find" option to search, the first suitable item is highlighted. To browse through the styles, buttons next to the input field highlight and can be used to show the previous or next suitable item.

Successful search options are added to the list and saved between sessions.



Set of post-effects has been added :

Sharpen,Glow,
Blur,
Depth Of Field.

New color manipulation tools have been added :

Brightness / contrast controls,Hue Saturation Lightness controls,
Hue Saturation Lightness controls,
Input / output levels controls.

Interactive Color adjustments, Filters and Sketch rendering

Color adjustment

Filters

Sketch rendering

Color adjustment

The 'Colors' section was added to the palette 'Advanced Render Styles'. This section allows to edit the set of 'Color adjustment' parameters of the selected render style.

These parameters comply with the same parameters in the Render Manager:

Filters

The 'Filters' section was added to the palette 'Advanced Render Styles'. This section allows to edit the 'Filters' parameters of the selected render style.

These parameters comply with the same parameters in the Render Manager:

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Sketch rendering:

The 'Sketch' section was added to the palette 'Advanced Render Styles'. This section allows to edit the 'Sketch' parameters of the selected render style.

These parameters comply with the same parameters in the Render Manager:

Limitations:

Interactive post-processing is not available in the preview render in Render Manager. Note that if you change the tone mapping parameters of the rendering style in the Render Manager and press Ok, all windows with this style will render.

Interactive Tone Adjustments

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Where a Render Style with Tone is used it is possible to dynamically adjust the tone settings after the render has been completed.

The active Render Style must be selected in the palette to adjust the settings.

To make sure that you have the active Render Style selected:

Right Click in the Palette in the Thumbnail area.

Click on Find Rendering Style.

The option for the RedSDK tone settings should now be available.

Adjusting the Gamma

Adjusting the White color balance

Adjusting the Curve

Adjusting the Tone Mapping

Adjusting the Burn factor

Adjusting the Exposure

Adjusting the Key

Loading and Saving Render Styles



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Default UI Menu: Options/Rendering Styles/Load Rendering Styles, Options/Rendering Styles/Save Rendering Styles

Ribbon UI Menu:

If you want to save Render Style properties for future use, select Save Render Styles, or right-click in the Render Styles Palette and select Save Render Styles. Use the

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categories on the right side of the Save window to specify which render styles you want to save. Render style data will be written to a .dat file, located in the Render Styles folder. Any new render style you define will be automatically saved in the TurboCAD file as well. To load a render style .dat file, select *Load Render Styles from the Render Styles menu, or from the local menu of the palette.

Managing Render Styles in the Render Manager



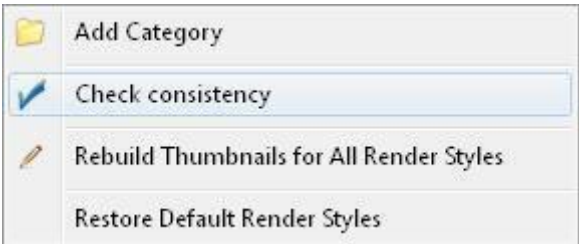
Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

Or you can select a Render Styles on the Advanced Render Styles palette, right-click and select **Edit Render Styles**.

Right clicking on tree elements provides a variety of functions. Right clicking on **Render Styles** allows you to:

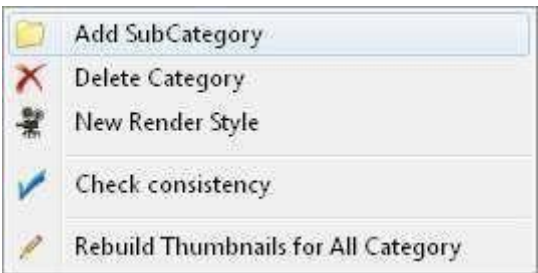
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- Add Category
- Rebuild Thumbnails for all Render Styles
- Restore Default Render Styles



Right clicking on a **Category** allows you to:

- Add SubCategory
- Delete Category
- New Render Style
- Rebuild Thumbnails for all Category



If a render style has been copied you will also be allowed to:

- Paste Render Styles

Right clicking on a **Render Style** allows you to:

Copy Render Style Here - adds a copy of the selected render style to the bottom of the current category.

Delete Render Style

Rename Render Styles

Copy Render Styles - Copies the Render Style so it can be pasted elsewhere.

Rebuild Thumbnail for Render Styles

The toolbar on the Left side of the editor allows you to specify the elements of the preview.



You can navigate within the preview window using the center mouse button.

Tone Mapping Control

The Tone Mapping control uses a curve based input control. The control consists of two parts: "Graph" part and "Text" part. The "Text" part presents the "reference" points, which are illustrated in the "Graph" part by rectangles. The curve is constructed on the basis of the "reference" points using the Lagrange polynomials (with condition that x, y should be inside $[0, 1]$). The curve must contain at least two points to be valid $(0, 0)$ and $(1, 1)$.

The curve can be edited as through the "Graph" part and through the "Text" part in the following way:

Adding the reference point. Can be done through the "Text" part by adding the new pair (x_i, y_i) or by using the mouse, clicking on the "Graph" to set the point. It should be far enough from already created reference points (if it is too close to an existing point that point will be selected and ready for the dragging). If created by mouse the new point will be automatically selected and ready for the dragging.

Changing the reference point. Can be done through the "Text" part by changing some pair (x_i, y_i) or by mouse by clicking and dragging.

Removing the reference point. Can be done through the "Text" part by removing the appropriate pair (x_i, y_i) or through the "Graph" part by using keyboard (pressing "Delete" button) when the mouse cursor is inside the rectangle of the appropriate point. The point must **not** be selected.

UV Mapping

Default UI Menu: Tools/UV Mapping Tools

Ribbon UI Menu:

UV mapping is a method of projecting (wrapping) an image onto a 3D object.

U and V indicate the axes of the image they are used in lieu of X Y Z to avoid confusion, since X Y, and Z are used to designate the axes of the 3D object. Color from the image is mapped to the polygons that compose the surface to the 3D object. In essence the process designates the where each area of the image is to appear on the 3D object. When a 3D object is rendered a mesh of that object is passed to the rendering engine. For surfaces, and Smeshes, the surface mesh itself is passed directly to the rendering engine. For solids, a mesh version of the solid is generated then passed to the rendering engine.

Multiple images can be used where each image is mapped to discrete areas of the mesh.

UV Mapping requires the following primary steps:

- Creating and inserting the image to be mapped

- Creating a UV Material

- Associating the image with the 3D object

- Unwrapping the mesh

- Refining the image

Associating the Image with the 3D Object

To associate an image with a 3D object:

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Select UV Mapping Tools

Click on the Image

Note: You can only select the UV Mapping Tools if there is both an image and a 3D object in the drawing.

1. Click on the 3D Object

The object and the facet you clicked on will become highlighted. At this point the image and the object are associated but not yet mapped.

Using Multiple Images

You can map multiple images to a 3D object. To do so follow these steps:

- Create a unique material for each image you wish to use.

- Insert each image you wish to use into the drawing.

- Use the Facet Editor to assign the correct material to each facet as needed.

 - Select the Facet Editor.

 - Select the facet(s) you wish to modify.

 - Right click and select Properties from the local menu.

 - Navigate to the Materials page and assign the desired material. Then click OK.

- Map the correct image to each facet(s) as needed.

Creating and Inserting the Image

You can use any graphic tool to create your image, including TurboCAD. Image can be JPGs, PNGs, BMPs, or GIFs.

The higher the resolution of the image the more refined the material will be on your model. You should consider an image of 1000x1000 as the bare minimum. You can insert a picture using either From File or From Image List.

Creating the UV Material

Use the Render Manager to create a new category; we suggest that you use "UV" as a name.

Create a new material.

Select the Pattern section of the material properties.

Set the Shader to Wrapped Image.

Specify the location of the file you will be using as your image.

Set the Softness to 0. This will prevent fuzziness in your image.

Select the Wrapping section of the material properties.

Set the Shader to UV.

Click OK to close the Render manager.

Select the 3D object and assign the new material you created.

Proportional Editing

Proportional editing acts like a magnet to smoothly deform the UV part by modifying vertices within a given range.

To use the tool:

Activate UV mapping tools

Select the Image

Select the TC surface the selected TC surface should already have unwrapped segments.

Enter the **Edit 2D Parts** mode

Select a single 2D Part

In the local menu choose **Start Proportional Editing**.

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Specify the radius of editing area and lock the starting position of editing area.
The radius of editing area is defined in the UVmap coordinate system.

The radius of editing area can be changed in three ways:

A. Pressing the "+" and "-" buttons. B. Choosing the local menu items B. Manually specify the exact radius in the Edit bar

To lock the starting position of the editing area position the mouse and then click the left mouse button or press ENTER.

After fixing the editing area a user changes the points of 2D part by dragging the mouse.

The manner in which points are moved is defined in the UV Properties in the Transformation Equation.

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Constant transformation = No Falloff, all point move a once.

Linear transformation = Linear falloff

Quadratic transformation = Smooth falloff

Cubic transformation = Exponential falloff

Sharp transformation = Steep falloff

All the segments transformations are recorded in the Undo buffer, but Undo or Redo operations are possible only after leaving UVMapping.

Refining the Image

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The primary process for improving or creating your finish image is by refining the image. There are two primary components:

- Export 2D Part Objects

- Move 2D parts as needed

Note: You can also review your image in your Windows default image editor or viewer by using **Open the Selected Image**.

The process is to:

- Export 2D Part Objects

- Edit the result in your favorite image editor

- Update the Selected Image

- The repeat steps 2 and 3 (or 1, 2, 3 if necessary) until the image is as you need.

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Original Image

Refined Image

Render Result

Export 2D Part Objects

You can access the Export 2D Part objects via the UV Mapping tools local menu. At least one facet must be selected in order for this option to appear.

When exporting only the facets that are selected will be used/embedded in the export.

Export to

Selected Image - Export result to the selected images file.

New Drawing - Open a new CAD drawing and embed result

Current Drawing - Embed the result into the current CAD drawing.

Image –

Add – Destination = Destination + Source (If the amount is greater than 255, the result is 255)

And – Destination = Destination & Source (Bitwise AND two images)

Xor – Destination = Destination xor Source (Bitwise XOR two images)

Or – Destination = Destination or Source (Bitwise OR two images)

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Copy – Destination = Source

Sub – Destination = Destination - Source

Blend – Destination = Source + degradation

Screen – Destination = Destination + Source + degradation (The result image will have a lighter tonality)

Avg – Destination = (Destination+ Source)/2

Bg. Color - Set the background color for the resulting image.

Contours - Export the contours to the result.

Solid Fill - Export the solid fills to the result.

Surfaces - export the surfaces to the result.

UV Rectangle - (not available for Selected Image)

Update the Selected Image

You can Update the Selected Image used for any UV Mapping.

Selected the UV Mapping Tools.

Select the Image.

Right click and select Update the Selected Image from the local menu.

Unwrapping the Mesh

Once you have associated an image to a 3D object you will need to unwrap the facets of the 3D object into 2D parts that can be arranged on the image. This will allow you to 'map' which part of the image is to appear on each facet.

The following applies equally to unwrapping ACIS Solids and Surfaces.

You can individually select facets on the 3D object, or select all of the facets.

To select an individual facet, click on the blue node of that facet. To select multiple individual facets, hold down the shift key and then click on each facet's blue node.

To select all of the facets drag a selection rectangle around the entire 3D object.

At this point you will want to begin arranging the 2D parts. There are two complementary methods to do this: Packing and Editing.

Packing 2D Parts

Packing is a algorithmic method of arrange a group object within a defined space. For UV Mapping, packing arranges all of the 2D parts within the area defined by the image. To pack the 2D parts:

Select the desired facets.

Right click and select Packing 2D Part object(s) .

3. The 2D parts will automatically be laid out on the associated image.

You can un-pack the parts by clicking Packing 2D Part object(s)... again.

If you have un-packed you can re-pack by again clicking Packing 2D Part object(s) ...
The second time you pack the arrangement will be adjusted. To get back to the original packing arrangement. un-pack and repack once more.

When you are done packing right click and select Finish packing 2DPart object(s) ..

To de-select facet click away from the 3D object and image.

Editing 2D Parts

You can move, rotate, and scale parts to arrange them better for your purposes.

Select a facet or facets

Right click and select Edit 2D part object(s)...

The selected part(s) will be highlighted to indicate that it can be edited.

Click on the part(s) to begin editing.

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5. You can then edit the part by standard methods:

Move

Scale

Rotate

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You can finish any one alteration by right clicking and selecting Apply transform to 2D Part object(s).

7. To complete editing right click and select Finish edit 2D Part object(s).

Inversed Normals

Occasionally the part and the image will map with an inversed normal. This means the section of the image appears backward on the facet. To remedy this You need to reverse the layout of the 2D part. Select the part then use the middle blue node on the edge and drag in past the opposite edge. You can now reposition the newly edges, as long as you do not recross their positions. The edge you select depends upon how the reversal appears, in other words is it reversed along the X or Y axis. You cannot simply scale -1 along the X or Y axis via the Edit bar. This will only flip the part but the reversed association will remain. Do not flip the 2D part via rotation. This will only flip the part but the reversed association will remain.

Unwrapping Surfaces

In addition to the basic UV Mapping functions there are some additional functions for mapping surfaces.

Since it is generally undesirable to export every single facet of a surface as an individual 2D part the Surface functions are designed to gather groups of facets to create Segments of the 3D object. The resulting segment can then be converted into editable 2D parts.

The functions and their purpose are as follows:

Quick Selection: Selects a contiguous series of facets based on the setting in the UV Mapping Options.

Create Segment by Selection: Creates a segment based on the currently selected facets.

Auto Segmentation: Automatically composites a set of segments for all facets of the selected 3D object.

Explode Segements: Explode the currently selected facets back into facets.

Unwrap Segments: Unwrap the selected segments into 2D Parts.

Select segment by edges:

To activate the tool:

Activate UV mapping tools

Select the Image

Select a TC surface

Select "Create segment by edges..."

After that, select a closed edge loop. For this there are two possibilities:

Manual - Select the edge loop manually by clicking one edge after the other. All selected edges should be connected. This means that you can only select those edges which have common vertices with those already selected. Another limitation for each vertex there can only be two edges from the selected edge loop this prevents self-intersections.

Mixed manual and automatic. If you have already have selected a few edges, but not closed, you can try to close edge loop automatically by selecting **Close current selected edge loop...** The program will try to find the shortest path between the ends of the currently selected edges and connect them.

After creating of the closed loop, you can create a segment by seleting **Create segment by selection...**

For the each closed edge loop it is possible to create two segments; One from internal facets, the other from the external facets. The algorithm will automatically choose one of these domains. If the algorithm choose the opposite domain from the one you wanted select **Invert created segment...** which will invert the selection to the alternate facets.

It is important to note, that **Invert created segment...** will only function immediately after using **Create segment by selection...**

UV Mapping Options

You can access the UV Mapping Options by:

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Selecting the UV Mapping Tools

Right click and select UV Mapping Options.

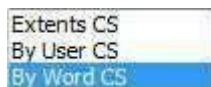
Note: You can only select the UV Mapping Tools if there is both an image and a 3D object in the drawing.

Additional Manipulation Options

Coordinate System Extents by - Sets the coordinate system (workplane) used by the selector shell. Because these settings work in tandem with other settings, their performance can become complex



Edit Bar CS - Sets which coordinate system will be used in the Edit Bar.



Display-2D Part

Show Surface - Surfaces with their gridlines will be shown

Max grid lines (5...2000) - defines the parameters of ACIS Refinement that are used to derive 2D polygonal meshes for surfaces. The greater the parameter value, the more precise the ACIS refinement will be, which also means a greater impact on time and memory.

Fill Contours - Show a fill for 2D parts.

Transparency (0...100) - set the transparency level for fills.

Item based on selected face - Set the color 2D parts associated to selected faces on the 3D object.

Selectable Item - Set the color for 2D parts which can be selected but are not.

Item based on unselected face - Set the color 2D parts associated to un-selected faces on the 3D object.

Item based on unselected face with data - Set the color 2D parts associated to un-selected faces on the 3D object which have UV Mapping already associated.

You can use the standard color pickers to assign the colors.

Display-General

The purpose of the General page is to control drawing of various objects in rendered views.

Skip 2D Part draw to render views - stops rendering of 2D Part objects.

Skip Marks draw to render views - stops rendering of marks.

Export-Components

Contours - the contours will be exported.

Solid Fill - the solid fills will be exported.

Surfaces - the surfaces will be exported.

UV Rectangle - the UV Rectangle will be exported.

Export-Colors

Color for exported components

Contours - set the color for exported contours.

Surfaces - set the color of exported surfaces.

Background - set the color of the exported background.

Export-General

Export to - How will the image be exported:

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New Drawing - In a new CAD Drawing

Current Drawing - Into the Current CAD Drawing

Selected Image - Out to the Selected Image's file.

Image Operation – This sets the method by which the Destination (Image File) is combined with the Source (exported 2D Objects). The color is combined for each RGB color channel per pixel.

Add – $\text{Destination} = \text{Destination} + \text{Source}$ (If the amount is greater than 255, the result is 255)

And – $\text{Destination} = \text{Destination} \& \text{Source}$ (Bitwise AND two images)

Xor – $\text{Destination} = \text{Destination} \text{ xor } \text{Source}$ (Bitwise XOR two images)

Or – $\text{Destination} = \text{Destination} \text{ or } \text{Source}$ (Bitwise OR two images)

Copy – $\text{Destination} = \text{Source}$

Sub – $\text{Destination} = \text{Destination} - \text{Source}$

Blend – $\text{Destination} = \text{Source} + \text{degradation}$

Screen – $\text{Destination} = \text{Destination} + \text{Source} + \text{degradation}$ (The result image will have a lighter tonality)

Avg – $\text{Destination} = (\text{Destination} + \text{Source}) / 2$

Image Map-General

Wait for completion of the program - the program will get control only after the other program finishes its work.

Use Default Image Program - set that image editing will use a default defined application.

Command - any string command that can be entered, for example, Run: dialog.

Working Directory - the default working directory for the image files.

Packing-General

Specify the parameters for the packing algorithm

Iteration count (5...100) - the number of iterations used by the packing algorithm. The greater the number, the better (packing is tighter), but the more time is spent on calculations. **Border Size Factor (0...0.3)** - the factor that affects the field size of parts. The smaller it is, the smaller the field. **Item Distance Factor (0...0.3)** - the factor that affects the distance between packed objects. The smaller it is, the smaller the distance. **Show Preview Rectangles** - if this option is on, then an outline rectangle is drawn around the packed objects. **Show Preview Rectangles** - Show the boundary rectangles around parts.

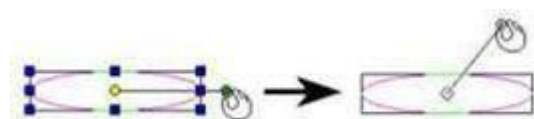
Selector-2D Parts

Shell Shape: Select **None** to make the selector shell (bounding rectangle) invisible.

Show Rotation Bars: If unchecked, the rotation bar is not shown and is not accessible.

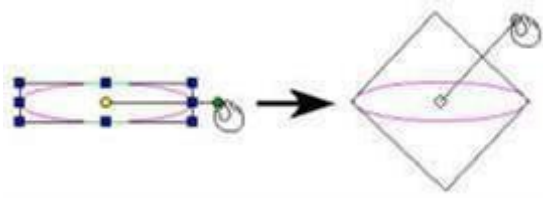
Link Selector Shell with Rotation Bar: If checked, when you press Ctrl and move the rotation handle, the selector shape will change accordingly.

Link Selector Shell not checked



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Link Selector Shell checked



Show Object while Dragging: Displays the selected objects dynamically as they are transformed. If not checked, only the selection shell is visible.

Wireframe - In wireframe mode.

Edit Bar Visibility: Controls the 2D transformation fields that appear on the Edit bars.

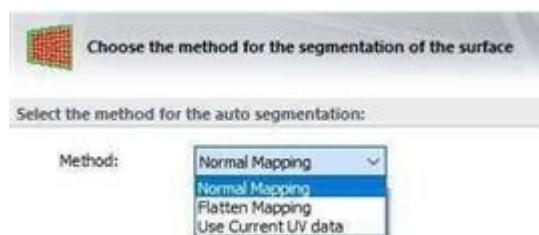
Local Menu Items visibility:

Default Reference Point, Edit Reference Point - Show the options in the Local menu.

Default Rotation Bars - Show the Rotation bars.

TC Surface Segmentation-General

Select the method for the auto segmentation:



Method – sets the method of mapping.

Normal Mapping - method of mapping applies planar maps based on directional procedures. It is the simplest method, but can result in greater texture distortion.

Flatten Mapping - method of mapping applies planar maps to contiguous polygons that are within a specified angle threshold. It prevents overlap of mapping clusters, but can still cause texture distortion. The Flatten Mapping dialog lets you control how clusters are defined and mapped.

TC Surface Segmentation-Properties

Normal Mapping

Top/Bottom mapping - Mapping originates from top and bottom.

Back/Front Mapping - Mapping originates from front and back.

Left Right Mapping - Mapping originates from left and right (relative to the a World CS)

Box Mapping - Mapping originates from cubically.

Flatten Mapping

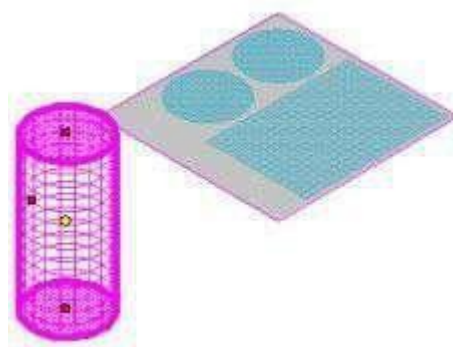
Face Angle Threshold (1.0...89.0) - The maximum angle that can exist between faces in a 2D part. As Flatten Mapping collects faces to be mapped, this parameter is used to determine which polygons it are combined. The higher the number is the larger the parts that result will be.

Unwrapping-Solid Body

Normalize the UV extents by geometric extents - 2D parts are a visualization of ACIS facets. Different types of facets have different scale in ACIS. For example, periodic facets (sphere, cone) are limited in size from $-\pi$ (3.14) to $+\pi$ (3.14). If this option is off, 2D parts have exactly the dimensions that are given to them by ACIS. If it is on, then the 2D part is changed in size in accordance with their geometric dimensions.

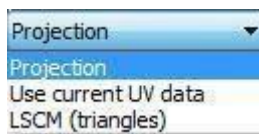
Off

On



Unwrapping-TC Surface

Specify method for the UV parametrization



Projection - In this case for the UV parametrization of each facet the coordinates are projected on a certain plane

Use current UV data - In this case it is used the texture coordinates which are already set. This method can be useful for the models which come from other programs, where the texture coordinates were set (on the picture below it is possible to see the example for the model from SketchUP).

LSCM (Triangles) - Least Squares Conformal Maps is a more complex algorithm, which allows us to receive better unwrapping results. There is one important restriction for the current implementation of this algorithm, namely all facets must be triangles.

For example, this segment has facets which are practically orthogonal to each other

The result is as follows with the Projection method.

However, with LSCM the following results will occur.

Specify the projection direction for UV parametrization

Note: This option is valid only for the Projection method.

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Automatic - In this case the direction of the projection is chosen automatically by the program.

X-axis - In this case as a projection direction the X - axis of the current workplane is used.

Y-axis - In this case as a projection direction the Y - axis of the current workplane is used.

Z-axis - In this case as a projection direction the Z - axis of the current workplane is used.

Specify the parameters direction for UV parametrization

Fix self-intersections in 2D parts - This option affects the presence of self-intersecting facets in 2D parts. This can lead to artifacts in rendering. When this option is enabled, during unwrapping presence of self-intersections is checked, and an attempt of resegmentation is made with the purpose to remove self -intersections. This procedure may take a few iterations and considerable time but it is recommended for the complex bodies.

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When Unchecked (option is disabled)

When Checked (option is enabled)

Remove holes in 2D parts - When this option is enabled, an attempt to merge the large and small 2D parts will be made. It will require to reduce the number of small 2D parts.

Note: This option is valid only for the Projection method.

UV Material Wizard

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The Material wizard automates the process of creating a UV compatible material.

To use the Material wizard:

Go to Tools.

Select UV Mapping Tools. (This will only be available if there is atleast one 3D object in the model)

Optionally: Click on an image to select it. And/Or Click on a 3D object to select it.

3. Right click and select UV Material Wizard .

4. Set the parameters in the Wizard dialog, and then press OK.

PARAMETERS

LightWorks - If selected a Lightworks material is created.

Category - Set the category in which the material will reside.

Material - Select or type in the name for the material.

Image From File - If you pre-selected an image this will be unchecked, and the file and path field will be filled.

File and Path field - If unfilled use the adjacent File Open button start the File Open dialog, and then locate and select an image. **File Open button** - Opens the File Open dialog.

Scale:

U - Set the U scale for the material.

V - Set the V scale for the material.

Shaders:

Color - Set whether the image will be used to specify the color or pattern of the material.

Reflectance - Set whether the image will be used to specify the color intensity or Reflectance pattern of the material.

Transparency - Set whether the image will be used to specify the color intensity or Transparency pattern of the material.

Bump - Set whether the image will be used to specify the color intensity or Bump pattern of the material.

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Open render Manager on Exit - I checked the Render Manager will open when you press OK.

Apply to the Selected body - If checked and if you pre-selected a 3D object the new material will be assigned to that object.

18.12. Visualize

(Available in Platinum, Professional, and Deluxe)

TurboCAD now supports Visualize rendering.

Visualize Preferences Settings can be found in Program Setup/Visualize Preferences dialog.
Other Visualize Settings are found in Drawing Setup/Visualize..

Relevant Device Mode Selection:

Automatic device type change on weak or virtual video cards or when working on a remote computer.

If the user's computer does not support the selected device type, then the appropriate device mode is automatically selected in the following order: OpenGL ES2 -> OpenGL -> GDI.
If OpenGL ES2 is not supported, then OpenGL is used. If OpenGL is not supported, then GDI is used.

The corresponding warning is displayed.

If the computer does not support the selected device type with hardware acceleration, then corresponding warning is displayed.

Camera Properties:

Visualize Render mode has been added to the camera properties of Hidden Line and Draft modes.

Further Visualize settings can be accessed via Drawing Setup/Visualize.

Following are the examples of Visualize rendering in different modes:

19. Customized Programming

You can customize your TurboCAD application by recording and editing macros, and using the SDK.



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

19.1. Macro Recorder

(Available in Platinum)

Default UI Menu: Tools/Palettes/MacroRecorder

Ribbon UI Menu:

Menu: View / MacroRecorder Palette

Creates and plays scripts - scenarios of creating objects and manipulating their properties. You can also record object transformation - move, copy, scale, and rotate. Scripts can be saved for future play.

Recording and Playing Scripts

To record a script, click **Start Record** in the MacroRecorder palette.



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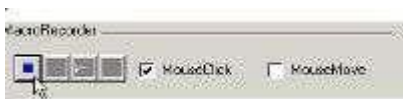
Script Options:

Mouse Click: The script will include each mouse click, including work in the Inspector Bar, use of the local menu, etc.). A script created with mouse clicks can be used only in TurboCAD. A script that was created without mouse clicks can be used in any TurboCAD add-on, if the add-on is created based on the TurboCAD Object Model.

Note: A script that uses mouse clicks cannot close windows. For example, if a viewport creation is recorded, when this script is played, the **Named View** window must be closed manually.

Mouse Move: Available when recording mouse clicks, records all mouse moves as well. For example, if you record the creation of a circle using mouse moves, you are recording exactly how the cursor moves while sizing the circle. Without mouse moves, you will only record the end result of the circle. This option increases the size of the script, and results in slower playback. Mouse move recording is recommended for scripts in which exact mouse positioning is important, such as work in the **Edit Tool**.

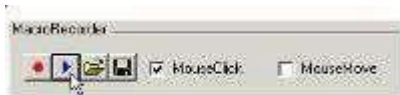
To finish the script, click **Stop Record**.



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Warning: Keep in mind that the script will only contain the actions performed after starting the script and before ending it. For example, you can record creating a line, stop the recorder and then change the line color. When you play the script, the line will have a different color because the script does not include the color change.

The script text will be displayed in the **MacroRecorder** palette. You can edit the script directly in the palette by placing the cursor on the desired line and changing text. You can play the script back by clicking **Play Script**.



If you clear all items from the screen, or open a new file, playing the script will create new objects on the screen. To save the script, click **Save Script**. Scripts are saved as **.tcr (TurboCAD Recorder) files**.



You can play a saved script by clicking ***Open Script**.



Select the desired script, and it is loaded into the palette. Click **Play Script** to play it.

Sample Scripts

Several sample scripts are provided for you. Click **Open Script**.

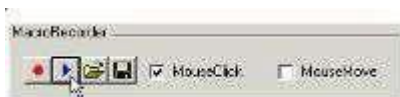
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Set **Files of type** to VBS and browse to TurboCAD's installation folder. Browse to SDK \ Samples \ VBS \ WshScriptPack. This folder contains a few scripts you can run.

Select the script you want to run and click **Open**. The script opens the Macro Recorder palette. (You can also open these files in a text editor.)

Click **Play Script**.



Some scripts give you messages related to what is found in your open files. The last script draws several 3D objects.

Script Limitations

Currently, some TurboCAD objects, such as images, cannot be used in scripts. Scripts record object creation only, and not all TurboCAD settings can be reproduced. For example, workplane position and views cannot be set in a script. Therefore, set the desired positions of the workplane and views before playing the script.

19.2. Parametric Part Script Editing

(Available in Platinum)

Default UI Menu: Tools/Palettes/Parametric Part Script Editor

Ribbon UI Menu:

Parametric parts (PPM) are defined using a text description (script). The script defines the structure, editable properties, and output that result in a parametrically editable part. The script must be saved with a *.PPM extension. The name of the file determines the name of the part.

Examining a Script

A simple example of a parametric part is a rectangle where the width, height and rotation angle are defined through parameters. The script of such part might look as follows:

```
// Here is a description of simple rectangle.  
H = Parameter("Height", 5, LINEAR, Interval(0, 100)); L =  
Parameter("Length", 10, LINEAR, Interval(0, 200)); Angle =  
Parameter("Angle", 0, ANGULAR, Interval(0, 360));
```

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```
Rect1 = Rectangle(H, L);  
Rect = RotateZ(Rect1, Angle);  
Output(Rect);
```

Let's examine each line of this example:

LINE 1

// Here is a description of simple rectangle.

The '/' indicates a comment. Comments do not affect the behavior of a part. All text following '/' to the end of the line are contained by the comment. **LINE 2** H =

```
Parameter("Height", 5, LINEAR, Interval(0, 100));
```

The second line specifies the definition of the 'H' parameter. Let's break out each element of the line to define its function: **H** This is the identifier (name) of the parameter in the part description = The equals sign associates the identifier with its definition

Parameter This is a function. 'Parameter' defines that H is a Parameter (Specifies the start of the Parameter function's properties **"Height"** The name of the parameter that will appear in the Properties dialog , Indicates the end of one property and the beginning of the next **5** Assigns the default value for H , Separates properties **LINEAR** Specifies that H is a linear value , Separates properties **Interval(0, 100)** Specifies the allowable values for H as an interval from 0 to 100) Specifies the end of the Parameter function's properties ; End of definition for H

LINES 3 – 4

```
L = Parameter("Length", 10, LINEAR, Interval(0, 200));
```

```
Angle = Parameter("Angle", 0, ANGULAR, Interval(0, 360));
```

The next two lines in the example are similar to the previous one. They define the characteristics of L and Angle parameters in a similar layout. Note that the 'Angle' parameter uses an ANGULAR interval rather than LINEAR.

LINE 5

```
Rect1 = Rectangle(H, L);
```

This line uses the Rectangle function to define a rectangle called 'Rect1'. It uses the previously defined H and L parameters to specify its properties, height and length. The center of this rectangle will be at the world origin (x=0,y=0,z=0) in the drawing. More on the rectangle tool will be covered later.

LINE 6

```
Rect = RotateZ(Rect1, Angle);
```

This line defines a new rectangle called 'Rect' which is a rotated version of 'Rect1',

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using the Angle parameter to define the rotation.

LINE 7

Output(Rect);

The last line specifies that the output of the script will be the rotated rectangle called 'Rect'. This is what the be drawn as the part.

Script syntax

The description of a parametric part consists of the entire contents of a text file, except comments, tabs, and other control characters, which are ignored. Comments are specified either using "/" characters that mean that all subsequent characters up to the end of the line are comments, or using the pair "/" **and** "/" that denote beginning and end of the comment, respectively.

A text description is a set of two types of operators:

<Identifier>

and

<Expression>;

Identifiers

The <Identifier> defines the symbolic name of an object. It is a set of Roman letters and Arabic numerals, which must start with a letter. For example valid names would be:

PART2a

MyPart

A134

Object identifiers may not be the same as names of functions or such names as PI, or LINEAR. These are reserved words that are used to designate the constants of the

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scripting language. The list of all reserved names is provided in the reserved word list which appears at the end of this chapter.

Expressions

Expressions define the associated identifier. Expression syntax matches the expression syntax in the majority of programming languages. They may define numeric value, arithmetic operations, the dependence of the defined object on other objects and function calls. **The structure of a function call is:**

<Function name> (<list of parameters>),

Examples of correct expression syntax:

(D --1/4) * k;

Polyline(Point(0, 0.25 - 1/8), Point(0, D), Arc1(L-C, - m, m), Point(0,0));

A=B+0.5;

B=7;

Arithmetic Operations

Arithmetic operations may use the standard arithmetical operators '+' (addition), '--' (subtraction), '*' (multiplication), '/' (Division) and parenthesis '('and ')', to determine the sequence of performing arithmetic operations. Object identifiers and numbers serve as operands.

Script Semantics

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A script contains full description of a parametric part. The collection of script operators determines which actions need to be performed to create the resultant object(s). Correct understanding of a script, requires having a clear understanding of how its operators are interpreted. Identifiers that are used in a <Expression> must be defined. In other words it must have been used as:

<Identifier> = <Expression>;

The list of resultant objects is defined in the Output(..) operator. The Output(..) operator contains a list of which objects are to be displayed in the resulting part. This operator must be present in the script. Each object in the list of arguments for Output(..) must be defined. In other words it must have been used as:

<Identifier> = <Expression>;

This operator must be present in the script. At least one object must be listed in the Output operator, but you need not output every object used in the script. The Output operator determines the method that will be used to create an object with this name.

A correct script describing a parametric part should conform to the following rules:

A script may have more than one Output(..) operator, but any <Identifier> should be contained in only one Output(..) operator__.

For each object used in the Output(..) operator there should be one, and only one, instance of any <Identifier>.

For each object used in an <Expression> there should be one, and only one, instance of any <Identifier>.

Each identifier should be used only once as <Identifier>

Each identifier should be at least once in an <Expression> operator or in an Output(..) operator__.

Circular calculation and interdependent referencing are not allowed. The script must not contain interdependent where "Item One" is defined by "Item Two", and "Item Two" is defined by "Item One".

The following situations:

A = B + 0.5; B = sin(A);

or

A = C+5; B = D+42; C = (3(2+A));* D = A/2;

are **not** allowed. The first case A and B define each other directly. In the second case A is defined by B through C, and B is defined by A through D. This also means that an identifier is not allowed to depend on itself. For example, you cannot use an operator of this form: $H = H * 1.05$;

NOTE: The sequence of script operators is not important (except certain special cases that will be described later); because operators are sorted before the script is run.

Basic Functions

Probably the most significant advantage of this method of creating parametric parts is the compact size and clarity of the text description of parametric parts in script form. The set of basic functions used in such a description, determines the level of clarity and simplicity of scripts for a particular class of parametric parts. **Note:** It is intended that the set of basic functions will expand from version to version.

Description of Parameters

It is important to understand the structure used to within a Parameter function.

Format: `<id> = Parameter(<name>, <default value>, <type>[, <condition1>][, <condition2>].);`

NOTE: The '<>' markers are used to designate elements in the expression, and the '[']' markers are used to indicate elements which are optional.

<name>	The name displayed in the user interface;
<default	The default value of the parameter;

value>

Defines the parameter type. The following example values are possible: **LINEAR**, means that the parameter is a linear value in the selected linear units of measure. **ANGULAR**, means that the parameter is an angular value in the selected angular units of measure. (only degrees are available at this time) **TEXT**, a text string; **FONT**, a font name; **COLOR**, an RGB color value; **MATERIAL**, a material name; **CHECKBOX**, a logical value, either ON or Off

<type>

These are optional. They define possible restrictions imposed on parameters. Restrictions can be listed in arbitrary order and may take on the following forms: **Set(<value>,...)** – a list of permissible values of the parameter **Interval(<minvalue>, <maxvalue>)** – sets the minimum and maximum values of the parameter; **LessThan(<value>)** – indicates that parameter value should be less than the specified value

LessOrEqual(<value>) – indicates that parameter value should not be greater than the specified value **GreaterThan(<value>)** – indicates that parameter value should be greater than the specified value

<condition>

GreaterOrEqual(<value>) – indicates that parameter value should not be smaller than the specified value **Set(FolderList)**--- particular case of Set operator, when a list of permissible values are defined by operator FolderList. Restrictions should not contradict each other. For example, you cannot combine GreaterThan(5) with LessThan(2). When specifying parameter restrictions, it is not allowed to use identifiers or expressions that directly or indirectly depend on other parameters, as arguments of the above-mentioned functions. Only constants or constant expressions can be used, for example: LessOrEqual(PI/2).

Example of Parameter Description:

Alpha = Parameter("Rotation Angle", 45, ANGULAR, Interval(-90, 90)); // This creates a parameter used to define a rotation angle. The name is 'Rotation Angle', the default is 45, the value type is ANGULAR, and the Interval is from '-90' to '90'

Functions for Creating 2D Entities

The following functions are used to create 2D graphic entities

Circle

The Circle function is used to create circles

Format: Circle(<radius>[, <cx>, <cy>]);

<radius>	Defines the circle's radius
<cx>, <cy>	Defines optional arguments that set the (x, y) coordinates of the circle center. By default, cx = 0, cy = 0

Example:

```
N = Circle(D/2, 0, y0);
```

A more extensive example:

```
//circle.ppm-two circles r1 = Parameter("Radius1", 2.5, LINEAR, Interval(0.0, 10.0)); r2  
Parameter("Radius2", 1.25, LINEAR, Interval(0.0, 10.0)); xc = Parameter("CenterX", 3,  
LINEAR, Interval(-100, 100)); yc = Parameter("CenterY", 3, LINEAR, Interval(-100,  
100)); c1 = Circle(r1); // circle centered on the origin c2 = Circle(r2, xc, yc); // circle offset  
from the origin Output(c1, c2);
```

Rectangle

The Rectangle function is used to create rectangles.

Format: Rectangle(<width>, <height>[, <cx>, <cy>]);

<width>	Defines the rectangle width
<height>	Defines the rectangle height
<cx>, <cy>	Defines optional arguments that set the (x, y) coordinates of the rectangle center. By default, cx = 0, cy = 0

Example:

```
rect = Rectangle(W, H, W/2. H/2); // Left bottom corner is in (0,0) point
```

Polyline

The Polyline function is used to create polylines consisting of straight line segments and arc segments.

Format: Polyline(<list of arguments>);

<list of arguments>	Defines the list of arguments, delimited with commas. Arguments define individual segments of a polyline

A line segment is defined by 2 Points. An arc segment is defined with a Fillet function or with an Arc0 or Arc1 function and two Points on the ends of the arc. For polylines that contain only straight line segments, the <list of arguments> consists of only 2D points, defined using Point(x,y) function.

Format: Point(<cx>,<cy>)

<cx>	Defines the x coordinates of the point
<cy>	Defines the y coordinates of the point

For example, a rectangle can be defined in the following way:

```
rect = Polyline( // no end-of-line is used semicolon here Point(0,0), // since this function  
in on multiple lines Point(W, 0), Point(W,H), Point(0, H), Point(0,0) );
```

It should be noted that when a polyline's, first and last points are coincident, is called a

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closed polyline. This type of polyline bounds a certain area, and can be used for creating 3D objects. Polylines with arc segments are defined by adding auxiliary functions Arc0 and Arc1 to the list of arguments. Arc0 builds the circular arc clockwise, while Arc1 builds the circular arc counterclockwise.

Format: Arc0(<cx>,<cy>), Arc1(<cx>,<cy>),

<cx>	Defines the x coordinates of the arc center
<cy>	Defines the y coordinates of the arc center

The start and end point of an arc are defined by the preceding and the following arguments. Arc0 and Arc1 cannot be the first or last argument in the list of arguments. For a polyline that contains only one arc segment, the <list of arguments> consists of 2 Points defined with Point(x,y) function and an arc defined with either the Arc0 or Arc1 function.

Example of arc0 and arc1 in a polyline:

```
//Polyarc.ppm-polyline with arcs YSize=5; XSize=6; R= 1; Path= Polyline(Point(0, R),
start at top of rounded lower left corner. Point(0, YSize-R), // go to bottom of rounded
top left corner. Arc1(0, YSize, R), // make this corner a "cutout" Point(R, YSize), // left
side of top edge Point(XSize-R, YSize), Arc0(XSize-R, YSize-R, R), // make this corner
a "fillet" Point(XSize, YSize-R), Point(XSize, R), Arc0(XSize-R, R, R), // another fillet
Point(XSize-R, 0), Point(R, 0), Arc1(0, 0, R), // another cutout Point(0, R));
Output(Path);
```

Another method of creating an arc in a polyline is to use the auxiliary function Fillet, which "smooths" two linear segments that start and end in the preceding point, by adding an arc with the specified radius into the corner. This ensures smoothness at the junction points.

Format: Fillet(<radius>);

<radius>	Defines the radius of the fillet

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Example of fillets in a polyline:

polyfillet.ppm - polyline with fillets H = 5; L = 10; FR = 1; p2 = Polyline(// Rectangle with rounded corners Point(0,0), // lower left corner Point(L,0), // lower right corner Fillet(FR), // places fillet at bottom right Point(L,H), // upper right corner Fillet(FR), // places fillet at top right Point(0,H), // upper left corner Fillet(FR), // places fillet at top left Point(0,0), // closes rectangle Fillet(FR) // fillets start/end corner. Since this is a closed shape, // no following Point function is needed.); Output(p2);

Fillet and Arcs can be used together in the same Polyline function.

Example of Arcs and Fillet in a polyline:

Poly1 = Polyline(//Rectangle with rounded corners Point(0,0), Point(W - r, 0), Arc1(W - r, r), Point(W, r), Point(W, H - r), Arc1(W - r, H - r), Point(W - r, H), Point(0, H), Fillet(r), Point(0,0), Fillet(r));

Functions for Creating 3D Entities from 2D Entities

You can use 2D entities as the basis for creating 3D objects.

Thickness

The Thickness function creates a 3D entity based on the 2D entity by adding thickness. It also allows you to change the thickness property of the 3D object.

Format: Thickness(<Object>, <value>);

<Object>	Defines the initial graphic object
<value>	Defines new value of Thickness

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Example of Thickness:

```
RectA = Rectangle(2, 5); RectThick = Thickness(RectA, 3);
```

Example of Thickness Used to Create a Box Function:

```
Input(x0,y0,z0,x1,y1,z1) R = Rectangle(x1-x0, y1-y0, (x0+x1)/2, (y0+y1)/2); T =  
Thickness(R, z1-z0); Output(Move(T, 0, 0, z0));
```

Another Example of Thickness:

```
//thickrect.ppm - draws a 2D rectangle and adds thickness L = Parameter("Length", 4,  
LINEAR, Interval(0.1, 20)); W = Parameter("Width", 3, LINEAR, Interval(0.1, 20)); H =  
Parameter("Height", 1.5, LINEAR, Interval(0.1, 20)); Rect = Rectangle(L, W); Box =  
Thickness(Rect, H); Output(Box);
```

An Example of Thickness with a Circle:

```
thickcircle.ppm - draws a circle and adds thickness Cylind=Thickness(Circle(1,2,2),2);  
Output(Cylind);
```

An Example of Changing Thickness:

```
thickcircle2.ppm - draws a cylinder and changes thickness  
Cylind=Thickness(Circle(1,2,2),2); Cyl2 = Thickness(Cylind, 4); // changes the thickness  
of the first cylinder Output(Cyl2);
```

Sweep

The Sweep function creates a 3D object by extruding a specified profile along a path, defined by a 2D polyline or circle. The profile is defined by a closed 2D polyline or circle.

Format: Sweep(<profile>, <path>[,<rotation angle>]);

<profile> This defines the profile using a 2D polyline

<path> This defines the path, along which the profile is "dragged"; the path is defined by a 2D polyline **Note:** The plane of the path and the plane of the profile must not be parallel.

<rotation This optional argument, defines the rotation angle of the profile relative the Z axis; by default, the

angle>	argument is equal to zero

Example of Sweep:

```
Poly1 = Polyline( Point(0,0), Point(1, 0), Point(1,2), Point(0, 2), Point(0,0)); PolyProfile  
RotateX(Poly1, 90); // the Rotate function will be explained later PolyPath = Polyline(  
Point(0,0), Point(10, 0), Point(10,10), Point(0, 10), Point(0,0) ); PolySweep =  
Sweep(PolyProfile, PolyPath); Output(PolySweep);
```

Another Example of Sweep

```
//sweep1.ppm R = 2; D = 5; C1 = RotateX(Circle(R, D/2+R, 0),90); // profile C2 =  
Circle(D/2,0,0); //path Torus = Sweep(C1,C2); Output(C1,C2,Torus); //C1 and C2  
shown for reference
```

An Extended Example of Sweep

```
//sweep2.ppm - another sweep example L = Parameter("Length", 5, LINEAR,  
Interval(0.005, 1000)); W = Parameter("Width", 3, LINEAR, Interval(0.005, 1000)); H =  
Parameter("Height", 1, LINEAR, Interval(0.1, 3)); FR = Parameter("FilletRadius", 0.3,  
LINEAR, Interval(0.001, 100)); p = Polyline(Point(0,0), Point(0,H), Point(-FR,H), Point(-  
FR,0), Point(0,0)); p1a = RotateX(p,90,0,0); p1 = Move(p1a, 0, W/2, 0); p2 = Polyline(  
Point(0,0), Point(0,W), Fillet(FR), Point(L,W), Fillet(FR), Point(L,0), Fillet(FR),  
Point(0,0), Fillet(FR)); s = Sweep(p1, p2); Output(s);
```

Functions for Creating 3D Entities Directly

3D object may also be created directly without reference to a 2D entity.

Sphere

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The Sphere function is used to create a 3D sphere.

Format: Sphere(<radius>[,<cx1>,<cy1>,<cz1>]);

<radius>	This value specifies the radius of the sphere
<cx1>,<cy1>,<cz1>	These are optional argument used to specify the x, y, z location of the sphere's center point. By default the values for these argument is zero

Sphere Example:

```
SR1 = Sphere(10,1,3,5.5);
```

Another Sphere example:

```
//sphere.ppm - simple sphere example  
R = Parameter("Radius", 2.5, LINEAR,  
Interval(0.01, 20)); cx = Parameter("CenterX", 0, LINEAR, Interval(-100, 100)); cy =  
Parameter("CenterY", 0, LINEAR, Interval(-100, 100)); cz = Parameter("CenterZ", 0,  
LINEAR, Interval(-100, 100)); S = Sphere(R, cx, cy, cz); Output(S);
```

Cone

The Cone function is used to create a 3D cone.

Format: Cone(<Height>,<baseradius>[,<topradius>]);

<Height>	This value specifies the height of the cone
<baseradius>	This value specifies the radius for the base of the cone
<topradius>	This optional argument specifies a radius for the top of the cone, creating a truncated cone. By default the value for this argument is zero

Cone Example:

```
CN1 = Cone(10,5,2);
```

Another Cone Example

```
//cone1.ppm - a simple cone  
R = Parameter("BaseRadius", 0.5, LINEAR, Interval(0.01, 10));  
H = Parameter("Height", 3, LINEAR, Interval(0.05, 20));  
Cone1 = Cone(H, R, 0);  
Output(Cone1);
```

Example of a Truncated Cone:

```
//cone2.ppm - a truncated cone  
R1 = Parameter("BaseRadius", 0.5, LINEAR, Interval(0.01, 10));  
R2 = Parameter("TopRadius", 0.1, LINEAR, Interval(0, 10));  
H = Parameter("Height", 3, LINEAR, Interval(0.05, 20));  
Cone2 = Cone(H, R1, R2);  
Output(Cone2);
```

Functions for Transforming Geometric Objects

This class of functions is used for moving, and rotating geometric objects. These transformations are related to the transformation of the coordinate system. As always functions create transformed objects, while original objects do not change.

Move

The Move function is used to move (shift) graphic objects.

Format: Move(<Object>, <dx>, <dy>, <dz>[,count]);

<Object> Defines the original graphic object

<dx>,
<dy>, <dz> Defines value of movement along x, y and z axes, respectively

<count> Defines the number of created objects, where each subsequent object is created by moving the

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preceding object; this argument is optional, with the default value of 1

Example of Move:

```
PolyProfile = Move(Poly1, 1, 3);
```

Another Example:

```
//move.ppm- illustrates the Move function RB=Parameter("BaseRadius", 2, LINEAR, Interval(0.1, 10)); RT = Parameter("TopRadius", 0.5, LINEAR, Interval(0, 10)); H = Parameter("Height", 4, LINEAR, Interval(0.1, 20)); con1 = Cone(H, RB, RT); cx = Parameter("CenterX", 5, LINEAR, Interval(-10, 10)); cy = Parameter("CenterY", 0, LINEAR, Interval(-10, 10)); cz = Parameter("CenterZ", 0, LINEAR, Interval(-10, 10)); count=Parameter("Copies", 2, LINEAR, Interval(1, 10)); con2=Move(con1, cx, cy, cz, count);// create count copies, offsetting each by cx, cy, cz Output(con1, con2);
```

Rotate

The RotateX, RotateY, RotateZ functions are used to rotate graphic objects around the X, Y and Z axes, respectively.

Format: RotateX(<Object>, <rotation angle>[, <cy>, <cz>[, <count>]]);
RotateY(<Object>, <rotation angle>[, <cx>, <cz>[, <count>]]); RotateZ(<Object>, <rotation angle>[, <cx>, <cy>[, <count>]]);

<Object>	Defines the original graphic object
<Rotation angle>	Defines the angle of rotation
<cx>, <cy>, <cz>	Sets an offset for the rotation axis relative to the X, Y and Z axes (in accordance with function names). These arguments are optional; however, only all three arguments can be omitted at once. Default value for each of <cx>, <cy>, <cz> is zero
<count>	Defines the number of created objects, where each subsequent object is created by transforming the preceding object; this argument is optional, with the default value of 1

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Example of Rotate:

```
PolyProfile = RotateX(Poly1, 90);
```

Another Example of Rotate:

```
//rotate.ppm - demonstrates the rotate functions c1 = Circle(2, 10, 0); // create a circle c2  
RotateX(c1, -90, 0, 0); // rotate the circle to lie in the XZ plane c3 = Move(c2, 0, -0.05,  
0); // move it back half the thickness c4 = Thickness(c3, 0.1); c5 = RotateZ(c4, 30, 0, 0,  
11); //duplicate the circle by rotating about the Z axis c6 = Circle(2, 0, 10); c7 =  
Move(c6, 0, 0, -0.05); c8 = Thickness(c7, 0.1); c9 = RotateX(c8, -30, 0, 0, 11); c10 =  
Circle(2, 0, 0); c11 = RotateZ(c10, -90, 0, 0); c12 = Move(c11, 10, 0, -0.05); c13 =  
Thickness(c12, 0.1); c14 = RotateY(c13, 30, 0, 0, 11); Output(c4, c5, c8, c9, c13, c14);
```

Functions for Loading External Symbols as Elements

You can load non-parametric external symbols from external files to be a part of a parametric part. The files must be importable (supported) by the CAD system, such as *.TCW, *.DWG, *.SKP

StaticSymbol

The StaticSymbol function loads non-parametric symbols from external files. When the external symbol's filename is specified with no path information, the symbol is automatically assumed to reside in a sub-folder named Macro that is located in the ppm file's home folder.

Format: StaticSymbol(<FileName>[,BlockName]);

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<FileName> Defines the file name with extension. If the extension is not specified, native file format will be used

This is an optional argument, which indicates that only the block with the given name should be used as the symbol for loading, and the rest of the contents should be ignored; if the argument is not defined, the active drawing will be loaded as a symbol

Example of StaticSymbol:

//staticsym1.ppm - loads an external file from the Macro sub-folder S =
StaticSymbol("ExternalSymbol.tcw"); Output(S); //static symbol from
ExternalSymbol.tcw file is inserted on the drawing

Set(FolderList(...))

To create a list of files in a folder, Set(FolderList(...)) is typically used as the Parameter restriction.

Format: <id> = FolderList(<path> <mask> = "*.ppm");

<path>	Defines the path to the folder from which the list of files will be created
<mask>	Defines the mask of file names and extensions

Example of Set(FolderList(..)):

staticsym2.ppm - loads an external symbol from a different folder than Macro
DrawingName = Parameter("Drawing", "Drawing1", Set(FolderList("..\\..\\Drawings",
"*.tcw"))); //quantity of "..\\.." (before folder Drawings) is equal to quantity //of steps over
folder tree starting from the Macro sub-folder. S0 =
StaticSymbol("..\\..\\Drawings\\DrawingName.tcw"); //here a static symbol is loaded
from a file with a tcw-extension, // and a filename picked from the FolderList obtained
via the DrawingName parameter. Output(S0);
When specifying a relative path, you must remember that the path is always assumed to

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start, not at the folder that contains the ppm file, but in a folder below that named "Macro". In the example above, assume for the moment that staticsym2.ppm is located in:

C:\Users\Me\Documents\MyCAD\PPM Documentation Samples

The path used in the FolderList path and the StaticSymbol path must then implicitly begin at

C:\Users\Me\Documents\MyCAD\PPM Documentation Samples\Macro

The external symbol is being loaded from:

C:\Users\Me\Documents\MyCAD\Drawings

That means the script must navigate up three directories to the MyCAD folder, then back down one level to the Drawings folder, so the correct relative path is:

..\..\..\Drawings

Another example, which loads a specific .tcw file from the Drawings folder:

//staticsym3.ppm - loads a specific file from a different folder S =

StaticSymbol("../..\..\Drawings\3DSliceTest.tcw"); //only loads the specific file

3DSliceTEst.tcw. //Remember that the relative path is still rooted in the Macro subfolder. Output(S);

A parametric part (a file with a *.ppm extension) can be loaded by calling the name of the parametric file as if it were a function, whose arguments are the parameters of the part to be loaded, in the order in which they are described in the file. Refer to "Creating Custom Functions" below for more details on this process.

Functions for 3D Boolean Operations

Functions of this class are used to perform Boolean operations on 3D geometric objects.

BooleanUnion

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The BooleanUnion function creates an object by adding the specified objects together.

Format: BooleanUnion(<Object>, <Object>,....);

<Object> Defines an object to be used in the Boolean operation. There must be at least two objects

Example of BooleanUnion:

```
S1 = Sphere(5); S2 = Sphere(5,5,5); S3 = Sphere(5,5,-5); S4 = Sphere(5,-5,5); S5 = Sphere(5,-5,-5); S6 = BooleanUnion(S1,S2,S3,S4,S5); Output(S6);
```

Another Example:

```
R = Parameter("Radius", 8, LINEAR, Interval(0.001, 1000)); s = Sphere(R); c = Circle(R/3); c1 = Thickness(c, R*2); c2 = Move(c1, 0, 0, R); //Cylinder s1 = BooleanUnion(s, c2); //Sphere with cylinder Output(s1);
```

BooleanSubtraction

The BooleanSubtract function creates an object by subtracting the secondary objects from the primary object.

Format: BooleanSubtract(<PrimaryObject>, <SecondaryObject>, ...);

<PrimaryObject>	Defines an object to be used in the Boolean operation. There is only one primary object
<SecondaryObject>	Defines a secondary object to be subtracted from the primary object There must be at least one or more secondary objects

Example of BooleanSubtract:

```
S1 = Sphere(5); S2 = Sphere(5,5,5); S3 = Sphere(5,5,-5); S4 = Sphere(5,-5,5); S5 = Sphere(5,-5,-5); S6 = BooleanSubtract(S1,S2,S3,S4,S5); Output(S6);
```

Another Example of BooleanSubtract:

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```
R=Parameter("Radius", 8, LINEAR, Interval(0.001, 1000)); s = Sphere(R); c =  
Circle(R/3); c1 = Thickness(c, R*2); c2 = Move(c1, 0, 0, -R); //Cylinder s1 =  
BooleanSubtract(s, c2); //Sphere with hole Output(s1);
```

BooleanIntersect

The BooleanIntersect function creates an object derived from the intersection of the primary and secondary objects.

Format: BooleanIntersect(<Object>, <Object>)

<Object> Defines an object to be used in the Boolean operation. There must only two objects

Example of BooleanIntersect:

```
S1 = Sphere(5); S2 = Sphere(5,5,5); S3 = Sphere(5,5,-5); S4 = Sphere(5,-5,5); S5 =  
Sphere(5,-5,-5); S6 = BooleanIntersect(S1,S2); Output(S6);
```

Functions for Modifying 3D Objects

Several functions are available to modify the geometry of 3D objects.

Fillet Edges

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The Fillet Edges function allows rounding one or multiple edges of 3D object.

Format: G3Fillet(<Object>,<Edges>, <Radii>);

<Object>	Defines the 3D object whose edges are to be rounded
<Edges>	Defines the edge or multiple edges, which are to be filleted. Each edge is defined by Point(xc,yc,zc) or Array of Points. Point(xc,yc,zc) is the middle point of an edge to be filleted (for example in the TurboCAD "Fillet Edges" operation, this point is marked with a blue square). Array of Points defines a set of edges to be filleted.
<Radiuses>	Defines the Fillet radiuses. Fillet radiuses are set by Array function. For a single edge the Array contains pair of values, for multiple edges - multiple pairs of values.

Fillet Edges Example:

```
Array(Point(x1,y1,z1), Point(x2,y2,z2), Point(x3,y3,z3)); //defines 3 edges for filleting  
//Point(x1,y1,z1), Point(x2,y2,z2), Point(x3,y3,z3); - 3 middle points on 3 edges to be  
filleted
```

Another Example:

```
Array(r1, r2)-- //array of radius values for rounding the selected edge. It defines  
rounding //radiuses for 2 ends of the selected edge. //r1 - start radius of fillet //r2 - end  
radius of fillet.
```

Example of Filleting 1 Edge:

```
G3Fillet(PartA,Point(xc,yc,zc), Array(r1, r2)); //where Point(xc,yc,zc) - middle of the  
edge.
```

Another Example:

```
Door=G3Fillet(Door0, Point(0, -1, (Height-FHeight-4-3/4)/2), Array(1, 1)); For example  
(fillet of 1 edge of the box): x = Parameter("size", 5, LINEAR, GreaterThan(0)); r1 =  
Parameter("r1", 1, LINEAR, GreaterThan(0)); b0 = Box(0, 0, 0, x, x, x); b1 = G3Fillet(b0,  
Point(x/2, 0, 0), Array(r1, r1*2)); Output(b1);
```

Example of Filleting 4 Edges of a Box:

```
L = Parameter("Length", 5, LINEAR); W = Parameter("Width", 3, LINEAR); H =  
Parameter("Height", 1, LINEAR); R = Parameter("Radius", 0.5); g0 = Box(0,0,0,L,W,H);  
g1 = G3Fillet(g0, Array(Point(L/2, 0, 0), Point(0, W/2, 0), Point(L/2, W, 0), Point(L, W/2,  
0)), Array(R, R, R, R, R, R, R, R)); Output(g1);
```

Chamfer Edges

The Chamfer Edges function allows chamfering any edge or multiple edges of 3D object.

Format: G3Chamfer(<Object>, <Edges>, <Offsets>);

<Object>	Defines the 3D object whose edges are to be chamfered
<Edges>	Defines the edge or multiple edges, which are to be filleted. Each edge is defined by Point(xc,yc,zc) or Array of Points. Point(xc,yc,zc) is the middle point of an edge to be filleted (for example in the TurboCAD "ChamferEdges" operation, this point is marked with a blue square). Array of Points defines a set of edges to be chamfered.
<Radiuses>	Defines the Chamfer distances. These are set by Array function. For a single edge the Array contains a pair of distance values, for multiple edges - multiple pairs of distance values.

A Chamfer Example:

Array(d1, d2)-- //array of 2 offset values at the ends of an edge.

Another Example:

Door= G3Chamfer(Door0, Point(0, -1, (Height-FHeight-4-3/4)/2), Array(1, 1)); //Here Door0 -is the object whose edge is to be chamfered. //Point(0, -1, (Height-FHeight-4-3/4)/2) - indicates this edge. //Array(1, 1) sets 2 chamfer distances

Another Example:

x = Parameter("size", 5, LINEAR, GreaterThan(0)); r1 = Parameter("r1", 1, LINEAR, GreaterThan(0)); b0 = Box(0, 0, 0, x, x, x); b2 = G3Chamfer(b0, Point(x/2, x, x), Array(r1, r1+r1)); Output(b2);

G3Offset

The G3Offset function extends a solid face inward or outward.

Format: G3Offset(<Object>, <Face>, <Offsets>);

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<Object>	Defines the 3D object whose edges are to be extended
<Face>	Defines the face, which is to be extended. The Face is defined by a Point(x,y,z) belonging to this face
<Offsets>	Defines the offset distance. A positive value will offset the face outward, and a negative value will offset inward

Offset Example:

G3Offset(PartA, Point(xf, yf, zf), dist); Where: PartA – is the 3D object whose faces are to be offset Point(xf, yf, zf) – is a point for selecting the face to be offset dist – is the value of face offset

Another Example:

```
x = Parameter("size", 5, LINEAR, GreaterThan(0)); r1 = Parameter("r1", 1, LINEAR, GreaterThan(0)); b0 = Box(0, 0, 0, x, x, x); b3 = G3Offset(b0, Point(x,x/2,x/2), r1/2); Output(b3);
```

G3Shell

The G3Shell function allows shelling the shape of solid object, leaving the selected face open. It creates a shell of a specified thickness from a single solid object. The new faces are created by offsetting existing faces inside or outside.

Format: G3Shell(<Object>, <Face>, <Thickness>);

<Object>	Defines the 3D object whose edges are to be shelled
<Face>	Defines the face that should remain open. It is defined by the Point(xc,yc,zc) function which describes a point belonging to this face
<Thickness>	Defines the shell thickness. A positive value creates an outward shell, and a negative value creates an inward shell

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Shell Example:

G3Shell(PartA, Point(xf, yf, zf), thickn); Where: Part3 – selects the object which is to be shelled Point(xf, yf, zf) – is the point on the face, which should remain open thickn – is the shell thickness

Another Example:

L = Parameter("Length", 5, LINEAR); W = Parameter("Width", 3, LINEAR); H = Parameter("Height", 1, LINEAR); T = Parameter("Thickness", 0.2, LINEAR); g0 = Box(0,0,0,L,W,H); g1 = G3Shell(g0, Point(L/2, W/2, H), T); Output(g1); //After inserting a shelled object in the drawing, shell thickness can be edited in the Selection Info palette (as well as Length, Width and Height parameters)

G3Bend

The G3Bend function is used for bending 3D objects.

Format: G3Bend(<Object>, <Line>, <Angle>, <Radius>, <Depth>);

<Object>	Defines the 3D object which is to be bent
<Line>	Defines a line about which the solid object will be bent. It is defined by 2 Points: Point(x1, y1, z1), Point(x2, y2, z2). The line must lie on the solid face selected for bending.
<Angle>	Defines the bending angle. The angle is measured from the plane of the bent face.
<Radius>	Defines the bending radius
<Depth>	Defines the Neutral Depth to set the distance into the depth of material along which there will be no tension or compression

Bend Example:

G3Bend(Part3, Point(x1, y1, z1), Point(x2, y2, z2), Angle, R, 0);

Another Example:

P1=Thickness(Rectangle(10,20),3); B0 = G3Bend(P1, Point(3, 3, 0), Point(3,8,0), 90, 2, 0); Output(B0);

Setting and Changing Object Properties

The SetProperties function is used to set the properties of objects.

Format: SetProperties(<Object>, <PropertyName> = PropertyValue, <PropertyName> = PropertyValue, .);

<Object>	Defines the object to be used as the base for the new object with set properties
<PropertyName>	Defines the name of the property to be set. The name should be surrounded with quotation marks
<PropertyValue>	Defines the value to be assigned to the property

Example of SetProperties:

```
BlueRect=Rectangle(10,5); RedRect = SetProperties(BlueRect, "PenColor" = 0xff,  
"PenWidth" = 0.2); Output(RedRect);
```

Another Example:

```
Side2M = SetProperties(Side2, "Material" = "Wood\Pine", "PenColor" = 0xff);
```

Another Example:

```
PL1 = SetProperties(PL0, "Brush" = "SOLID");
```

Another Example: SetPlastic = ("Material" = "Plastics\Plain white"); BoxMaterial = SetProperties(MyBox,SetPlastic);

In the Parametric Part manager there is a special tool to choose the required value for such properties as Material, Pen Color and Brush Style. To activate it, right-click on the property name. This will open the Local Menu either for Material table or PenColor table or BrushStyle table. The appropriate table will appear where the desired value can be chosen.

Nesting Functions

Functions can be nested within a single expression to optimize scripting efficiency.

For Example: BF = BooleanSubtract(B1,Move(RotateZ(RotateY(Box(-5,-5,-5,5,5,5),45),45),-1,-1,-1));

Example Used in a Small Script:

B1 = Box(0,0,0,10,10,10); BF = BooleanSubtract(B1,Move(RotateZ(RotateY(Box(-5,-5,-5,5,5,5),45),45),-1,-1,-1)); Output(BF);

Functions for Creating Text

Text

The Text function defines the text string itself and its characteristics, including fonts, style, effects, etc. Acceptable font values are dependent upon those installed on your machine.

Format: Text(<Text object>, <Text Font>, <Text Style>);

<Text object>	Defines the text string. Text string can be specified either directly here (with quotation marks) or via an identifier of text object
<Text Font>	Defines the text font
<Text Style>	Defines the text style

Example:

bsb = Text("BS(b)", Tfont, Tstyle);

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TextFont

The TextFont function sets the text font, size, and the angle of text line location.

Format: TextFont(<mode>, <Height>, <Angle>,);

<mode>	Defines the mode of the text: Standard (when mode=0) or Scalable (when mode=1 or any other value different from 0)
<Height>	Defines the text font size
<Angle>	Defines the angle of text line
	Defines the text font

Example:

```
Tfont = TextFont(0,2, 45, "Arial");
```

Where: **0** – means that text is Standard **2** – text height **45** – text line is located at 45 degrees **Arial** – font

TextStyle

The TextStyle function sets the text style including justification, text effects and styles.

Format: TextStyle(<list of characteristics>);

<list of
characteristics>

Defines the text characteristics separated with commas. The following values of characteristics are allowed: For Justification: LEFT, CENTER, RIGHT, TOP, MIDDLE, BASELINE, BOTTOM
For Text Effects: BOX, UNDERLINE, STRIKETHROUGH, ALLCAPS For Style: BOLD, ITALIC

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Example:

```
Tstyle = TextStyle(LEFT, TOP, UNDERLINE);
```

Another Example:

```
//Standard text of Times New Roman font with 5in of font size, //with Left,Top  
justification, with TextBox effect, Bold, Italic, at 45 degrees of Angle ht=5; font_name =  
"TimesNewRoman"; Tfont=TextFont(0, ht, 45, font_name); Tstyle=TextStyle(LEFT,  
TOP, BOX,BOLD, ITALIC); bsb = Text("BS(b)", Tfont, Tstyle); Output(bsb);
```

Auxiliary Functions

Extents

The ExtentsX1, ExtentsX2, ExtentsY1, ExtentsY2, ExtentsZ1 and ExtentsZ2 functions are used to calculate the extents of graphic objects. **Format:**

```
ExtentsX1(<Object>); ExtentsX2(<Object>); ExtentsY1(<Object>);  
ExtentsY2(<Object>); ExtentsZ1(<Object>); ExtentsZ2(<Object>);
```

<Object>	Defines the object to be used

The presence of X, Y or Z characters in the function name determines axis along which the extents will be calculated. 1 or 2 index--indicates whether minimum or maximum value should be calculated.

Example of Extents:

```
xmin = ExtentsX1(PartA); xmax = ExtentsX2(PartA); ymin = ExtentsY1(PartA); ymax =  
ExtentsY2(PartA); zmin = ExtentsZ1(PartA); zmax = ExtentsZ2(PartA); P1 = Box(xmin,  
ymin, zmin, xmax, ymax, zmax);
```

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Another Example of Extents:

```
A0=Thickness(Rectangle(H-3/4,D), 3/4); A1=RotateY(A0,90); xmin = ExtentsX1(A1);  
xmax = ExtentsX2(A1); ymin = ExtentsY1(A1); ymax = ExtentsY2(A1); zmin =  
ExtentsZ1(A1); zmax = ExtentsZ2(A1); P1 = Box(xmin, ymin, zmin, xmax, ymin+3,  
zmin+4);
```

ParameterPoint

The ParameterPoint function defines a parametric point with number and coordinates.

Format: ParameterPoint (<N>,<xc>,<yc>,<zc>);

<N>	Defines the number of the parametric point
<xc>,<yc>,<zc>	Defines the coordinates of parametric point

Example of ParameterPoint:

```
P0 = ParameterPoint(0, l, -l, 0); P1 = ParameterPoint(1, 0, 0, 0);
```

PointX, PointY, PointZ functions

The PointX, PointY, PointZ are used to calculate the coordinates of parametrical point. The PointX function calculates X-coordinate of parametrical point. The PointY function calculates Y-coordinate of parametrical point. The PointZ function calculates Z-coordinate of parametrical point.

Format: PointX (<point>); PointY(<point>);. PointZ(<point>);

<point>	Defines the parametrical point from which the X or Y or Z coordinate will be extracted

Examples of Point:

```
x0 = PointX(P0); // x0=1 for P0 = ParameterPoint(0, 1, -1, 0); y1 = PointY(P1); //y1=0 for  
P1 = ParameterPoint(1, 0, 0, 0); z1 = PointZ(P1); //z1=0 for P1 = ParameterPoint(1, 0,  
0);
```

Special functions and operators

IF

The IF function allows various actions to be performed depending upon whether the specified condition is fulfilled or not fulfilled. It plays the role of a conditional operator, and can be used to create branches in the logic of building a parametric part.

Format: IF(<Condition>, <ExprOnTRUE>, <ExprOnFALSE>);

<Condition>	Defines the condition under test using the following comparison operations: == (equal) < (less than) > (greater than) <= (not greater than) >= (not less than)
<ExprOnTRUE>	Defines the value of the IF function when the value of <Condition> is TRUE;
<ExprOnFALSE>	Defines the value of the IF function when the value of <Condition> is FALSE;

IF Example:

```
A = IF(L >= H, Rectangle(L, H), Rectangle(H, L)); //Regardless of the specified size of L  
and H, the created rectangle A will be positioned //horizontally (the longer side will be  
along the X axis). /* In this example "Rectangle(L, H)" is the TRUE result and  
"Rectangle(H, L" is the FALSE result. */
```

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Another Example:

```
Tstyle = IF(dir > 0, TextStyle(MIDDLE, RIGHT), TextStyle(MIDDLE, LEFT));  
//Regardless of the specified size of dir, Text Style will be specified with Right or Left  
justification.
```

UNITS

The UNITS function defines the units that will be used in the script. It defines the System, Space Units and Scale of dimensions used while creating objects. This function allows loading parts correctly in drawings with different specified units.

Format: Units(<N>[<units of dimension>]);

<N>	Defines object scale
<units of dimension>	Defines the units in the English or Metric systems

Units (1[in]) – this means that the main units of measurement are inches. It is possible to use other units for some particular values even when the entire drawing is created with the default unit. In order to use millimeters for particular values while inches are default units, you can explicitly declare the desired unit for these values.

For Example:

Units(1[in]); // means that default unit of drawing is inches
Units(1[mm]); // means that default unit of drawing is millimeters

For example, you can use value M=5[mm]; and Units(1[in]) in the same script. It means that only M value is measured in mm while all others are measured in inches.

Moreover, this function allows for scaling the created objects down (when N<1) or up (when N>1).

For Example:

Units(2[in]); // created object is scaled up 2 times compared with the case of Units(1[in]);

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`_Units(0.5[in]);`// created object is scaled ½ as large as compared with the case of
`Units(1[in];_`

RefPoint

The RefPoint function sets the location of the Reference Point for the parametric part. When the Reference Point is one of the output values of a script, it is inserted in the drawing along with the part. This enables precise insertion of the parametric object into the drawing.

Format: RefPoint(<Point>);

<Point>	Defines the (x,y,z) coordinates for location of Reference Point

For Example:

`xArrow = PointX(P0); yArrow = PointY(P0); rf = RefPoint(xArrow, yArrow, 0); //->`

RefPoint is placed on the point (xArrow,yArrow, 0) Output(rf);

Input and Output

The Input and Output functions are used for inputting initial values or objects into the script and outputting result objects from the script.

Format: Input(<list of variable identifiers, separated with commas>); Output(<list of variable identifiers, separated with commas>);

<list of variable identifiers, separated with	Defines the list of variables or objects for input or a list of results

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commas>	for output

For Example:

```
Input(H, W, D, A, Dis); Output(SideA_L,Bottom_B,Back_I, Face1, FalseD1,  
E1,E2,E3,E4, N1, T1, Door, FF, SideA_R);
```

Example of the Output with Conditional Output:

```
Sw = Parameter("Switch", 1, CHECKBOX); P1 = Thickness(Rectangle(5,5), 3); S1=  
Thickness(Circle(2.5),4); Output(IF(Sw,P1,S1)); //Here is either cylinder or box inserted  
on the drawing //depending on checkbox Sw value
```

min and max

The min and max functions are used for choosing the minimum or maximum values within a set of values.

Format: min(<set of values>); max(<set of values>);

<set of values>	Defines the set of numerical values, identifiers of variables or Array of variables

For Example:

```
r=min(2,5,1,7,9);//r=1 R=max(2,5,1,7,9);//R=9
```

For Example:

```
A=2;B=5;C=1;D=7;E=9;A1=2;B1=5;C1=1;D1=7;E1=9;r=min(A,B,C,D,E);//r=1  
R=max(A1,B1,C1,D1,E1);//R=9
```

Example of using Array of Values:

```
A=2; B=5; C=1; D=7; E=9; r=min(Array(A,B,C,D,E));//r=1
```

Note: A Group of objects cannot be used as argument of these functions, because a Group is a collection of graphic objects, rather than a collection of numbers.

Mod

The Mod function is used for finding the remainder of the integer division. For example, Mod(5,4) is 1, because $5/4 = 1$, with a remainder of 1. Mod(7,4) is 3, because $7/4 = 1$, with a remainder of 3. Mod(7,3) = 1, because $7/3 = 2$, remainder 1. Note: The Mod function is often used to determine if a number is odd or even, because Mod(AnyOddNumber, 2) = 1, while Mod(AnyEvenNumber, 2) = 0. **Format:** Mod(<value1, value2>);

<value1 > Defines the expression or identifier that represents the dividend

<value2> Defines the expression or identifier that represents the divisor

For Example:

A = 7; B = 4; C = Rectangle(A, Mod(A,B)); Output(C);

Div

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The Div function is used to perform division.

Format: Div(<value1>,<value2>);

<value1>	Defines the dividend
<value2>	Defines the divisor

For Example:

A=7; B=3; result1 = A/B; result2 = Div(A, B); rect = Rectangle(result1, result2)

Output(rect);

Additional Math Functions

sqrt

Calculates the square root of a specified number $P = \text{sqrt}(b)$;

asin

Calculates the arcsine. Returns the angle in radians $P = \text{asin}(0.5)$;

acos

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Calculates the arcsine. Returns the angle in radians $P = \arcsin(0.5)$;

Array

The Array function defines an array of values, or an array of Points, by directly listing the elements of the array. In other words the Array function collects geometric objects or values into an Array object.

Format: Array(<list of objects>)

<list of objects>	list of numerical values or geometric objects An <object> can be represented by either a value, or the <identifier> of a value, or by a Point(x,y,z) function.

For Example:

Array(Point(L/2, 0, 0), Point(0, W/2, 0), Point(L/2, W, 0), Point(L, W/2, 0)) // It is the array of points defining the edges for G3Fillet. Array(R, R, R, R, R, R, R, R) //It is the array of radius values for filleting the array of edges.

Another Example: Can

```
txt=Parameter("text", "Simple text example", TEXT); a=Array(TextFont(0,10,"Arial"),
TextStyle(CENTER, MIDDLE, ITALIC)); //Array of 2 items: TextFont and TextStyle) s0 =
Text(txt, a); Output(s0);
```

Group

The Group function collects multiple graphic objects into a group and assigns an identifier name to the result. It allows the script to work with multiple objects as if they were a single object. Also a Group can be the output value of a script. Groups of objects

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can take part in different operations: Move, Rotate, etc.

Format: Group (<list of objects>);

<list of objects>

Defines the list of graphic objects, separated with commas. The <object> may be any graphic objects

For Example:

```
bse = Group(bse_below, bse_above); //group of 2 graphic objects
Br2 = Group(Br0, Br1);
```

For Example:

```
Bx = Group(Move(BxL, -Dis*1.5), Move(BxR, Dis*1.5)); ShelfFBx =
BooleanSubtract(ShelfF, Bx); Output(ShelfFBx, Bx);
```

Special Functions without Parameters

PI

The PI function calculates the value of Pi = 3.14159...

#\$AUX@_Contour

Defines a clipping contour for the resulting object. This is used to clip (remove) sections of walls when the object is inserted into a wall. It uses a referenced contour that is defined elsewhere in the script. The referenced contour must be closed.

For Example:

```
contour = Polyline(P10, Arc0(PointX(C1), PointY(C1)), P11,  
Arc0(PointX(C2), PointY(C2)), P12, Arc0(PointX(C3),  
PointY(C3)), P13,  
P03, P00, P10);  
contourZ = RotateX(contour, 90);  
ClipContour = SetProperties(contourZ, "$AUX@_Contour" = 1, "PenColor"= 0x0000ff);  
TS = RefPoint (W/2, 0, -elevation);  
Output(ClipContour, TS);
```

Creating custom functions

When scripts of the same type are created, which describe a particular class of parametric parts, it can be convenient to have the sequence of repeated actions as a separate specialized function. To achieve this, the repeated actions can be put into a separate <name>.ppm file. In this case, all input variables should be listed in the Input operator:

Format:

```
Input(<list of variable identifiers, separated with commas>);
```

For Example:

```
Input(x0,y0,z0,x1,y1,z1);
```

The Output operator should also be defined.

Format:

```
Output(<list of variable identifiers, separated with commas>);
```

A custom function created in this manner must be placed in a Macro folder, which is always located inside the folder of the calling script. When the custom function is used, the script's file name (without the .ppm extension) is used just as if it was a built-in function.

Format:

```
<file name>(<list of input parameters>)
```

Below is an example of a custom function. The file box.ppm can be found in the PPM

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Documentation Samples/Macro folder:

box.ppm - defines a custom Box function. // The custom function is called in this way:
B = Box(Xmin, Ymin, Zmin, Xmax, Ymax, Zmax); // The function creates a 3D box with
given min/max values Input(x0,y0,z0,x1,y1,z1); R = Rectangle(x1-x0, y1-y0, //
Rectangle with Xmin = x0, Xmax= x1 (x0+x1)/2, (y0+y1)/2); // Ymin = y0, Ymax = y1 T =
Thickness(R, z1-z0); // depth = Zmax - Zmin Output(Move(T, 0, 0, z0)); // move result
along z to Zmin The script below is box blend.ppm, which calls the custom function
box.ppm _//box_blend.ppm uses the custom Box.ppm function in the Macro folder. x =
Parameter("size", 5, LINEAR, GreaterThan(0)); r1 = Parameter("r1", 0.5, LINEAR,
GreaterThan(0)); b0 = Box(0, 0, 0, x, x, x); b1 = G3Fillet(b0, Point(x/2, 0, 0), Array(r1,
r1*2)); Output(b1);

File location is crucial when using parametric scripts as custom functions. In the
example above, if blend_box.ppm lies in the folder D:/Symbols, then it can only find the
box.ppm script if box.ppm is located in the folder D:/Symbols/Macro.

Parametric Parts Reserved Word List

PI	LINEAR	TEXT
ANGULAR	MATERIAL	FONT
COLOR	CHECKBOX	ITALIC
BOLD	UNDERLINE	BOX
ALLCAPS	STRICKETHROUGH	TOP
MIDDLE	BOTTOM	BASELINE
LEFT	CENTER	RIGHT
Call	Array	+
-	*	Div
Mod	/	-
sin	cos	tan
atan	min	max

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**	=	==
!=	<	>
<=	>=	&
Solid	Extrude	
UNIQUE	GraphicId	VertexId
	Vertex	Face
Edge	Source	Bound
Intersect	OperationList	BlendArg
BlendParam	BlendType	BlendRadiusMode
BlendSetback	BlendRadiusBlendSmooth	BlendRadiusParam
BlendOffsetParam	BlendFaceEntity	BlendFaceEdge
BlendFaceVertex	BlendEdgeEdge	BlendEdgeVertex
BlendEdgeVertexMain	BlendEdgeVertexAux	ShellArg
ShellThickness	ShellFace	ShellEdge
FaceEditArg	Transform	ScaleX
ScaleY	ScaleZ	ShearXY
ShearXZ	ShearYZ	RotateX
RotateY	RotateZ	TranslateX
TranslateY	TranslateZ	Path
Profile	LateralFace	LateralEdge
CapFace	CapEdge	JointEdge
Profiles	HighLight	FaceMaterialArg
FaceMaterial	FaceOffsetArg	FaceHoleArg
FaceHole	BendId	BendRadius
BendAngle	BendNeutral	BendFlag
BendPosition	BendFlangeHeight	BendAxialDistance
BendAzimuthAngle	BendEdgeStartPosition	BendEdgeEndPosition
Face2FaceLoftArg	Face2FaceLoft	
AssemblyAxis	Input	Output
Include	Units	StaticSymbol

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FolderList	Macro	Parameters
Parameter	ParameterPoint	PointX
PointY	PointZ	Set
Interval	LessThan	GreaterThan
LessOrEqual	GreaterOrEqual	Circle
Rectangle	Polyline	Point
Arc0	Arc1	Fillet
IF	Move	Thickness
Sweep	Cone	BooleanUnion
BooleanSubtract	BooleanIntersect	G3Fillet
G3Chamfer	G3Shell	G3Offset
G3Slice	G3Bend	ExtentsX1
ExtentsX2	ExtentsY1	ExtentsY2
ExtentsZ1	ExtentsZ2	Text
TextFont	TextStyle	Group
SetProperties	PatternCopy	

Example PPM Symbol



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

The following code will generate a window. To use the resulting window you must either:

Save the script as a .ppm file, and load the folder containing the file into the symbol library, and then drag the symbol onto a wall.

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Run the PPM script. Select the result, and turn it into a block. Finally drag the block from the block palette onto a wall.

3ArchWindow.ppm

```
Units(1[mm]);
H = Parameter("Window Height", 1200, LINEAR, Interval(400, 10000));
W = Parameter("Window Width", 2000, LINEAR, Interval(700, 10000)); Curvature =
Parameter("Curvature", 40, LINEAR, Interval(0, 100)); elevation = Parameter("Window
Eleveation", 700, LINEAR, Interval(0, 10000)); sash = Parameter("Window Sash", 30,
LINEAR, Interval(0, 100));
W1 = W/3.0;
P00 = Point(0,0); P10 = Point(0, H); P11 = Point(W1, H); P12 = Point(2*W1, H); P13 =
Point(3*W1, H); P03 = Point(3*W1, 0);
C1 = Point(0.5*(PointX(P10) + PointX(P11)), 0.5*(PointY(P10) + PointY(P11)) -
Curvature);
C2 = Point(0.5*(PointX(P11) + PointX(P12)), 0.5*(PointY(P11) + PointY(P12)) -
Curvature);
C3 = Point(0.5*(PointX(P12) + PointX(P13)), 0.5*(PointY(P12) + PointY(P13)) -
Curvature);
contour = Polyline(P10, Arc0(PointX(C1), PointY(C1)), P11, Arc0(PointX(C2),
PointY(C2)), P12, Arc0(PointX(C3), PointY(C3)), P13, P03, P00, P10);
contourZ = RotateX(contour, 90); ClipContour = SetProperties(contourZ,
"$AUX@_Contour" = 1, "PenColor"= 0x0000ff); TS = RefPoint (W/2, 0, -elevation);
Output(ClipContour, TS);
PL = Point(sash/2, H); PR = Point(W1 - sash/2, H); ArcWind1 = Polyline(Point(sash/2,
0), Point(sash/2, H), Arc0(PointX(C1), PointY(C1)), Point(W1 - sash/2, H), Point(W1 -
sash/2, 0), Point(sash/2, 0)); ArcWindz1 = RotateX(ArcWind1, 90);
ArcWind2 = Move(ArcWind1, W1, 0, 0); ArcWindz2 = RotateX(ArcWind2, 90);
```

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```
ArcWind3 = Move(ArcWind2, W1, 0, 0); ArcWindz3 = RotateX(ArcWind3, 90);  
ArcWindz1D = Thickness(ArcWindz1, sash); ArcWindz1Surf =  
SetProperties(ArcWindz1D, "Solid" = 0);  
ArcWindz2D = Thickness(ArcWindz2, sash); ArcWindz2Surf =  
SetProperties(ArcWindz2D, "Solid" = 0);  
ArcWindz3D = Thickness(ArcWindz3, sash); ArcWindz3Surf =  
SetProperties(ArcWindz3D, "Solid" = 0);  
contourZ3D = Thickness(contourZ, sash); contourZ3DSurf = SetProperties(contourZ3D,  
"Solid" = 0);  
res1 = BooleanSubtract(contourZ3DSurf, ArcWindz3Surf); res2 =  
BooleanSubtract(res1, ArcWindz2Surf); res3 = BooleanSubtract(res2, ArcWindz1Surf);  
Output(res3);
```

19.3. SDK

(Available in all TurboCAD Variants)

The TurboCAD Software Development Kit enables you to program your own routines within TurboCAD.

If you install TurboCAD using the **Full** installation, an SDK folder is created. This folder contains SDK samples and documentation (online help). Several SDK tools are provided in TurboCAD, located in the **AddOns** menu. These are located in the sections to which they are relevant. Each tool can be found in the index.

19.4. Using the Ruby Console

(Available in Platinum and Professional)

Default UI Menu: Ruby.Net Scripts/Toggle Ruby Console

Ribbon UI Menu:

You can use the Ruby console to run functions, load ruby scripts, or even define new functions.

The top portion of the Ruby console is the **Output Panel**. This is where Ruby scripts will put text output, and where the Ruby engine will notify you of any errors that it has encountered, or provide other notifications. It is possible to copy text from the Output Panel to the clipboard for reuse in the Input Panel or elsewhere.

The bottom panel of the Ruby console is the **Input Panel**. Here you can type in any functions that you want to call, or define new values or even functions.

The **Load...** button allows you to open a ruby script using a standard "Open" dialog box.

The **Save...** button allows you to save the contents of the input panel as a ruby script using a standard "Save As" dialog box.

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The **Evaluate** button tells Ruby to evaluate the text that you have typed into the Input Panel.

The **Close** button closes the Ruby Console. After you close the Ruby console you can re-open it at any time from the **Scripts/Toggle Ruby Console** menu command. The **Multiline** checkbox allows you to turn Multiline input off or on.

When **Multiline** is unchecked (the default), only a single line of input in the input panel is evaluated. This mode is convenient for running already-defined functions or defining and setting variable values on the fly. In this mode, pressing the **Enter** key is the same as pressing the **Evaluate** button.

When **Multiline** is checked, you are allowed to enter as many different lines of text as you want. In this mode, the **Enter** key works to end the current line and move the cursor to a new line, instead of performing the **Evaluate** function.

Multiline mode can be useful if you want to quickly define a simple function right in the Ruby console. After you are done entering lines, be sure to uncheck **Multiline** before pressing **Evaluate**.

Loading a script with the Load... button

To load a ruby script, click on the Load... button in the Ruby Console. This opens a dialog box that lets you browse to a ruby script and load its functions and other definitions into memory.

Note, however, that loading a script in this way will not automatically execute any of the methods in the script - you'll have to do that from the Ruby console as well. For example, if you want to execute a function **draw_stuff** which is contained in the ruby script **ConsoleLoadSample.rb** you would do the following:

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Click on the "Load..." button.

Using the "Open" dialog, browse to the folder containing ConsoleLoadSample.rb. Highlight ConsoleLoadSample.rb in the file list, and click "Open". The Output Panel adds a line "true" to indicate the script was opened successfully, but nothing else visibly happens.

Type "draw_stuff" in the Input Panel, and click **Evaluate** or press **Enter**. Now the function draw_stuff will perform whatever tasks it is supposed to.

Once the script has been loaded, it remains in memory for the duration of the TurboCAD session - you won't have to reload the script each time you want to run the draw_stuff function; simply enter "draw_stuff" in the input panel and press **Enter** to run the function again.

Loading a script using the load command

You can also load a script by using the **load** command in the Input Panel. In this case, you must specify the full path to the script, and use double backslashes as path separators. For example: load("C: MyRubyScripts draw_plus.rb") If the script is loaded successfully, the Output Panel will print a "true" response. If not, the Output Panel will print out one or more error messages. As when you use the **Load...** button, loading a script in this fashion does not automatically run any of the functions in the script.

Defining a function in the Input Panel

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To create a new function definition in the Ruby Console is straightforward. Let us walk you through a simple example:

Open the Ruby Console, if it's not already open.

Place a check mark in the **Enable Multiline** checkbox.

Type the following lines into the input panel, using the Enter key to end each line:

```
def sayit puts "There, I said it!" end
```

Click **Execute**. (Note: Don't press Enter. Use the Execute button.) The Output Panel should echo all the lines of the function, followed by "nil".

Uncheck **Enable Multiline**.

Type "sayit" in the Input Panel, and hit **Enter** or press **Evaluate**. The Output Panel should produce "There, I said it," followed by "nil".

Setting variables in the Input Panel

To set a single variable using the Input Panel, disable Multiline entry, and simply enter the variable's definition, and press Enter or click Evaluate. `a = 5` press **Enter** To set a series of variables, enable Multiline entry and enter the variables, then disable Multiline entry and click Evaluate. `a = 6 b = 4 c = 5 puts a puts a*b puts b+c` click **Executer**.

Clearing the Output Panel

Any time you want to clear the accumulated text in the console's Output Panel, use the `cls` command in the Input Panel: `cls` press **Enter**

More Ruby

Examples of Ruby scripts can be found in the RubyScripts folder within the Programs folder where you installed TurboCAD. Looking at these examples is the best way to familiarize yourself with Ruby in TurboCAD. For more advanced information there are several online sights dedicated to programming in Ruby, and there are many books available.

You will also want to familiarize yourself with the TurboCAD SDK for a better grasp of TurboCAD's functions. Some of the Ruby functions available emulate the functions of Ruby as used in Google SketchUp. Therefore it is advisable to look at the documentation of Ruby scripting in SketchUp as well.

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With this version of TurboCAD we have moved all the sample Ruby scripts to this RubyScripts subfolder of your "User Data" folder, in order to give you better access to them. Previously they were stored in a sub-folder of Program Files; restricted permissions on that folder made adding new scripts difficult for many users. The original goal with Ruby scripting for TurboCAD was to match as closely as possible the SketchUp Ruby function set; ultimately we had to abandon this goal due to

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significant differences in the two programs' feature sets. We have instead made the TurboCAD .NET API available via Ruby, and encourage you to use TC's native .NET API for calling TurboCAD functions. All the sample scripts that begin with "TC" use this API; all scripts beginning with "SU" utilize the older, incomplete Sket chUp-like API.

ABOUT THIS FOLDER

Note that ALL Ruby scripts placed in this folder will be automatically loaded by TurboCAD upon startup. If you don't want a script to be executed as TurboCAD starts, store that script in a different folder; either a subfolder of this one or a folder parallel to this one should work equally well. When you have tested a script to your satisfaction, and you want to make that script's function(s) available in the Ruby console when TurboCAD starts, add a line similar to the following to the end of your script:

```
UI.menu("Console").add_item("Name_on_menu"){function_name}
```

For example, a very simple script might say "Hello world!"; the contents of your ruby file would look something like this:

```
# MyHelloScript.rb def say_hello MessageBox.Show("Hello world!") end
UI.menu("Console").add_item("Hello"){say_hello}
```

When you save the script above to this folder, then close and restart TurboCAD, a new item "Hello" will be added to the droplist of available functions in the Ruby Console's pre-loads button.

REFERENCING XAML-BASED FORMS

TurboCAD's Ruby engine allows you to use WPF to create XAML-based dialog boxes for user input and other interactions with your script. By default, the Ruby engine will look in your default Drawings folder for such XAML files; if the XAML file cannot be found in that location, the Ruby engine will also attempt to find a non-pathed XAML file here. Otherwise, the reference to the XAML file in your script must point to the full path and filename of the XAML file, or use a path relative to this folder or the Drawings folder.

Some Examples:

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`window = UI.LoadXaml("MyXamlForm.xaml")` No path given – Ruby will first look for `MyXamlForm.xaml` in the default Drawings folder, then in this RubyScripts folder.

`window = UI.LoadXaml("./MyXamlForm.xaml")` Self-referencing relative path - Just as in the example above, Ruby will first look for this file in the Drawings folder, then in this RubyScripts folder.

`window = UI.LoadXaml("../XAML/MyXamlForm.xaml")` Relative path to parallel folder - In this example, Ruby goes up to the parent folder, then back down into a folder named "XAML" to look for `MyXamlForm`.

`window = UI.LoadXaml("C:\MyScripts\MyXamlForm.xaml")` Absolute path - Ruby looks for the xaml file in the exact location specified.

Note: IF YOU CHANGE YOUR DEFAULT DRAWINGS LOCATION, THE DEFAULT PATH WHERE RUBY LOOKS FOR XAML FILES WILL CHANGE ACCORDINGLY WHEN TURBOCAD RESTARTS. So if you change the default drawings path in TurboCAD's Options/File Locations dialog, you must update any relative paths to the XAML files in your scripts as well. In such a case, it may be easiest to move your entire RubyScripts folder into the parent folder that contains your new default drawings folder.

20. 3D Print



Note: This manual is for TurboCAD 2020 Designer, TurboCAD 2020 Deluxe, TurboCAD 2020 Professional and TurboCAD 2020 Platinum. Many of the features and functions may not apply to your version of TurboCAD. Please refer to your product as to feature availability.

(Available in Platinum, Professional and Deluxe)

The 3D Print tool allows you to automatically pass an STL file from TurboCAD to Axon. Axon is the 3D Printer control software for BFB (Bits from Bytes) printers.

In order for 3D print to work you must have Axon installed. Axon will also require that you have the Python programming language installed on your computer, and the Axon installer will prompt you for the installation of Python. The Axon software can be downloaded from the Bits from Bytes website <https://bitsfrombytes.com/> Register on the site - then navigate to the resources, then download. You are not required to have a BFB printer to use Axon, but the final build output of the software is only useable with BFB printers.

To use 3D Print, select 3D Print from the Tools menu.

Then select any (single) 3D solid in your drawing.

Axon will automatically start with your part loaded. If axon prompts you that there is an error with your STL file allow Axon to repair the file. Axon assumes that the units used in modeling are millimeters, so if to used inches to draw your file, once in Axon press the

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To Inch button.

