CADian 2006 for Autodesk AutoCAD Users

IntelliKorea Limited

www.cadian.com

CADian v/s AutoCAD

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CADian Vs AutoCAD

CADian was designed to interface with AutoCAD as seamlessly as possible, however there are some differences and enhancements in CADian as compared to AutoCAD.
This manual is specially designed to help AutoCAD users to migrate to Cadian.

Enhancement in AutoCAD commands

Line

While using line command on AutoCAD one needs to have a good understanding of Relative and Polar coordinate system, especially Polar coordinate system, where user might face difficulty in calculating the desired angle for drawing a line. CADian provides user an additional option of specifying Angle or Length, for users who are not well conversant with coordinate systems in AutoCAD.

Angle: Prompts user for the Angle, user can either provide angle by clicking or by typing the value of angle in degrees. It further prompts for the length, and then draws a line with specified Angle and length from the last point.

E.g.
: Line
ENTER to use last point/Follow/<Start of line>:
Angle/Length/<End point>: a

Angle of line: 30
Length of line: 5

Angle/Length/Follow/Undo/<End point>: ↵
:

Length: This option is exactly similar to the Angle option; the only difference is that it will first prompt for the Length and then the Angle.

E.g.
: Line

ENTER to use last point/Follow/<Start of line>:
Angle/Length/<End point>: l

Length of line: 5
Angle of line: 30

Angle/Length/Follow/Close/Undo/<End point>: ↵
:

Follow: This option is displayed only after at least one line segment is drawn in the current Line command. Using this option user can draw a line of specific length along the direction specified by the last segment drawn.

: Line
ENTER to use last point/Follow/<Start of line>:
Angle/Length/<End point>: a

Angle of line: 30
Length of line: 5

Angle/Length/Follow/Close/Undo/<End point>: f
Length of line: 3

Angle/Length/Follow/Close/Undo/<End point>: ↵
In the above option a line will be created of length 3 with and angle 30 from the last point, since the angle of the last segment drawn is 30.

Circle

In Command Circle Cadian provides additional functionality of creating Multiple Circles in a single command and converting an existing arc to a circle.

Arc: This option prompts user to select an arc, and converts the same to a circle.

: Circle
2Point/3Point/RadTanTan/Arc/Multiple/<Center of circle>: a
Select arc to convert to circle: *Select the arc you wish to convert to a circle*

Multiple: This option allows user to create multiple circle using a single command. Once user selects multiple options, it will prompt for all the options excepting Multiple again. User can go for any options he prefer, the radius of the resultant circle will be kept constant and user will be prompted to specify the center point to further create a new circle. This procedure will be repeated unless and until user gives a null Enter to come out of the command.

: Circle
2Point/3Point/RadTanTan/Arc/Multiple/<Center of circle>: m

2Point/3Point/RadTanTan/Arc/<Center of circle>:
Diameter/<Radius> <1.1804>:
Center of circle:
Center of circle:
Center of circle: ↵

Arc

AutoCAD has 11 ways of drawing an arc, where as in CADian there is __ ways of drawings an arc. The comparison of methods of drawing Arc in AutoCAD and CADian is shown below:

AutoCAD	**CADian**
<3 Points>	<3 Points>
Start, Center, End	Start, Center, End
Start, Center, Angle	Start, Center, Angle
Start, Center, Length	Start, Center, Length
Start, End, Angle	Start, End, Angle
Start, End, Direction	Start, End, Direction
Start, End, Radius	Start, End, Radius
Center, Start, End	Center, Start, End
Center, Start, Angle	Center, Start, Angle
Center, Start, Length	Center, Start, Length
Continuous	Continuous
	Start, Direction, End
	Start, Radius, Angle
	Start, Radius, End point
	Start, Angle, Center
	Start, Angle, End

Rectangle

The rectangle command has additional options for creating a Square or a rectangle rotated to a particular angle.

Square: A square is a rectangle with all equal side. It prompts user to pick two ends of a side of square. A square will be created with length and rotation angle equal to the distance and angle betweens the two picked points respectively.

: Rectangle
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: s
Chamfer/Elevation/Fillet/Rotated/Thickness/Width/<Select first corner of rectangle>:
Second corner of square:
:

Rotated: This option creates a rectangle and rotates the same to a particular angle.
: Rectangle
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: r
Chamfer/Elevation/Fillet/Square/Thickness/Width/<Select first corner of rectangle>:
Other corner of rectangle:
Rotation angle for rectangle <0.0000>: 30
:

Donut

Donut can be defined as a circle with a particular width. Hence CADian unlike AutoCAD has provided all the basic options available in Circle command for Donut as well. Hence we find additional options like 2P , 3P , Rad Tan Tan.

2P: This option prompts the user to enter the Width of Donut. The rest of the prompt is similar to that of Circle defined using 2P option.

: Donut
2Point/3Point/RadTanTan/<Inside diameter of donut> <1.4948>: 2p
Width of donut <0.2474>: 0.25
First point on diameter: *pick the first point on the diameter*
Second point on diameter: *pick the second point on the diameter*

3P: This option prompts the user to enter the Width of Donut. The rest of the prompt is similar to that of Circle defined using 3P option.

: Donut
2Point/3Point/RadTanTan/<Inside diameter of donut> <3.2710>: 3p
Width of donut <0.2638>: 0.25
First point on donut: *pick first point on the Circumference of the Circle*
Second point: *pick second point on the circumference of the Circle*
Third point: *pick third point on the circumference of the Circle*

RadTanTan: This option prompts the user to enter the Width of Donut. The rest of the prompt is similar to that of Circle defined using RadTanTan option.

: Donut
2Point/3Point/RadTanTan/<Inside diameter of donut> <2.8769>: rtt
Width of donut <0.2638>: 0.25
Diameter of donut <3.1407>: 3
Select first tangent point: *Pick the first Tangent reference*
Select second tangent point: *Pick the second Tangent reference*

Plane (Solid)

In CADian Plane command and Solid command are exactly similar. Solid command in AutoCAD was little confusing, as one has to be very specific with the order of points picked. In CADian you can create the basic solids like Rectangle, Square, Triangle easily just by picking corner points, which thinking 'bout the order of the points picked.

Rectangle: This option will prompt user for two corner points of a rectangle. Once user provides these two corner points, a solid rectangle is previewed to the user and it further prompts user for the rotation angle. Once the user provides the angle then the solid rectangle is rotated along the angle provided. It will keep prompting user for other corner and rotation angle, unless and until user gives a null enter.

: Plane
Rectangle/Square/Triangle/<First point of plane>: r

First point of plane:
Other corner of rectangle:
Rotation angle for plane <0.0000>:

Other corner of rectangle:
Rotation angle for plane <0.0000>:

Other corner of rectangle:↵

Square: This option will prompt user for two points defining base of a square. Once user provides these two points. A solid square created with base defined by two points. It will keep prompting user for Second corner, unless and until user gives a null enter.

: Plane

Rectangle/Square/Triangle/<First point of plane>: s

First point of plane:
Second corner of square:
Second corner of square: ↵

Triangle: This option will prompt user for two points defining base of an equilateral triangle. Once user provides these two points. A solid triangle created with base defined by two points. It will keep prompting user for Second corner, unless and until user gives a null enter.

: Plane
Rectangle/Square/Triangle/<First point of plane>: t
First point of plane:
Second corner of triangle:
Second corner of triangle: ↵

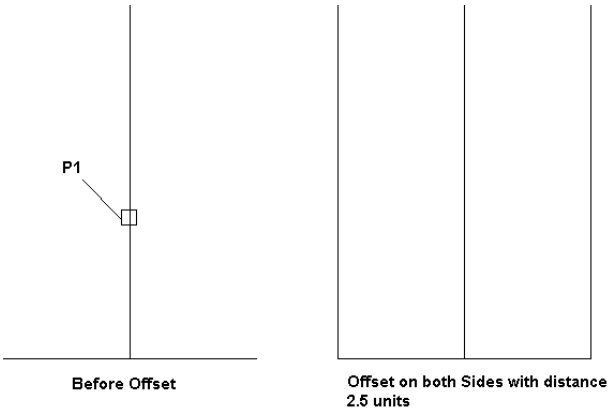
Snapshots

Msnapshot: This command works similar to Mslide command. Apart from sld files Msnapshot allows user to save the snap shot in Enhanced Metafile (Emf) and Windows Metafile (Wmf) file format as well.

Vsnapshot: This command works similar to Vslide command. Apart from sld files Vsnapshot allows user to view the snap shot in Enhanced Metafile (Emf) and Windows Metafile (Wmf) file format as well.

Offset

Both Sides: Normally when we offset any entity it asks for the side to offset, and offsets the entity in that side. There are frequent cases where user would like to offset an object symmetrically on both the sides. CADian support this kind of requirement, by providing both sides option in offset. User can either click on the side he wishes to offset, or type B for offsetting the entity on both sides.



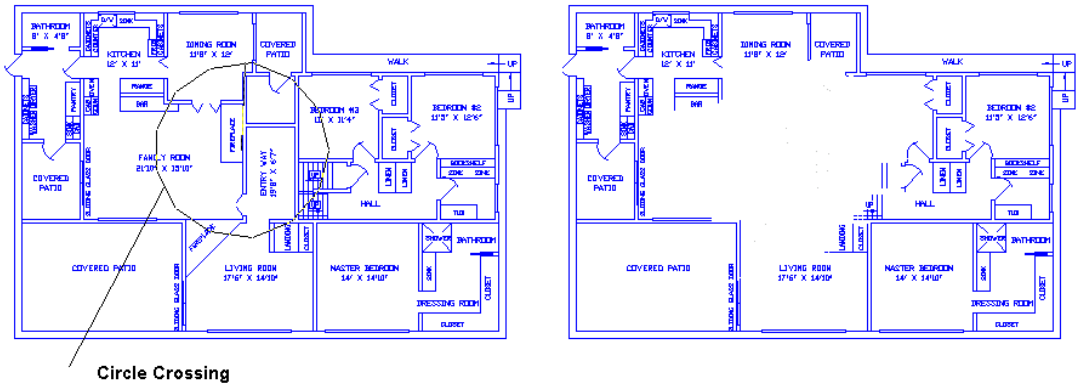
: Offset
Parallel: Through point/<Distance> <0.2000>: 2.5

Select entity:
Both sides/<Side for parallel copy>: b
Select entity:

Additional Options in Selection Set

Cadian has provided lot additional options for selection sets, apart from the existing AutoCAD options. These options are:

Circle crossing (CC): This option prompts user for center and radius of a circle, and selects the entities, which are completely inside the Circle or crossing the circle.

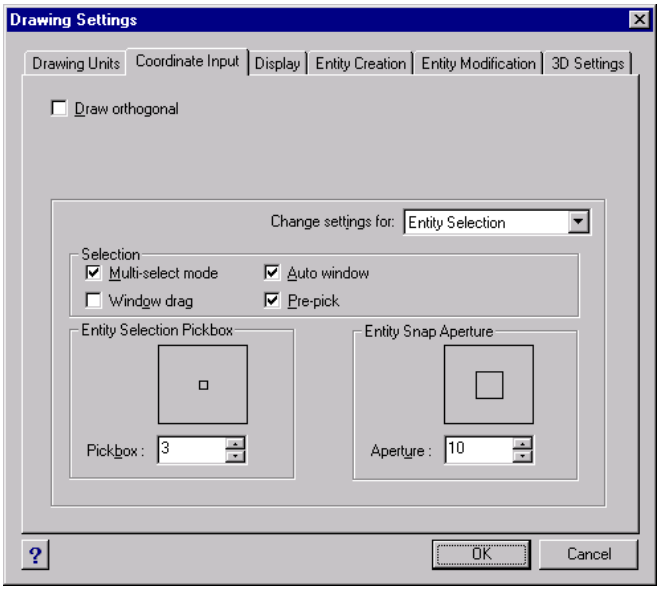


Circle Crossing

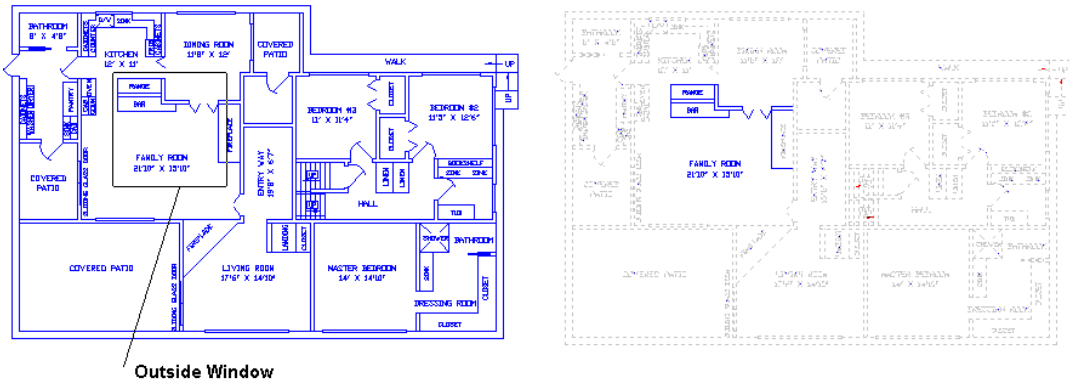
: Erase
Select entities to delete: cc

Center of circle:
Radius of circle:
Entities in set: 69
Select entities to delete:↵

Selection Method (D): This option displays the selection method tab, in the Drawing Settings Dialog box.



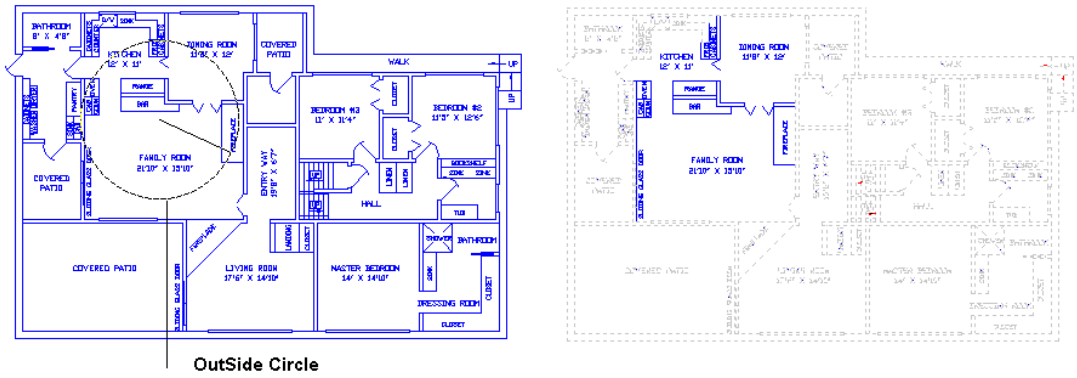
Outside Window (0): Prompts users for two corners of a rectangle, and selects all the entities, which are neither inside not crossing the rectangular area.



: Erase
Select entities to delete: o

First corner of exclusive window:
Opposite corner:
Select entities to delete:↵

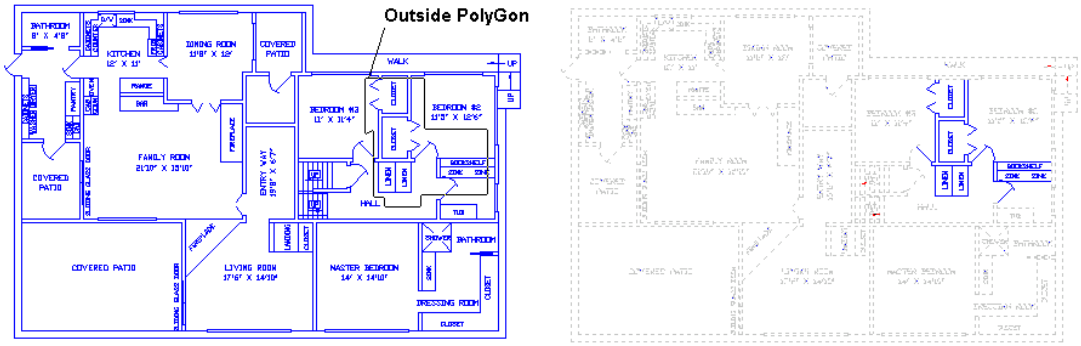
Outside Circle (OC): Prompts user for center and radius of circle, and selects all the objects, which are neither inside nor crossing the circular area.



: Erase
Select entities to delete: oc

Center of circle:
Radius of circle:
Select entities to delete:↵

Open Window (OP): This option prompts user for drawing a polygon, and then selects all the entities, which are neither inside nor crossing the polygon.

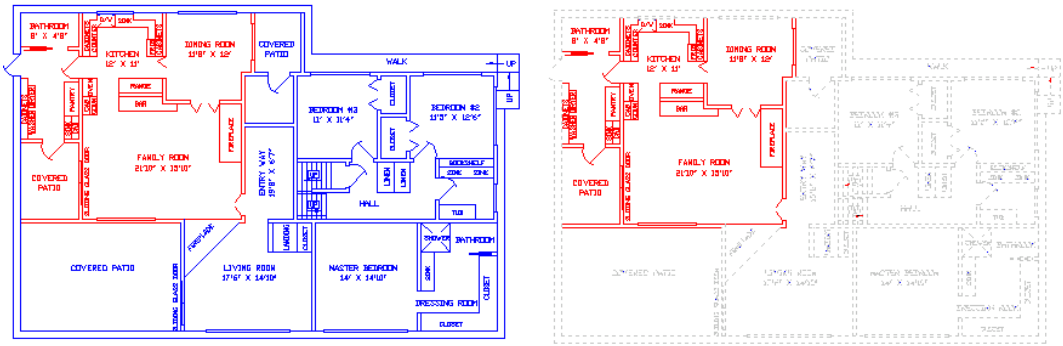


: Erase
Select entities to delete: OP

First vertex of outside polygon:
Next vertex:
Next vertex:
Next vertex:
Next vertex:
Next vertex:
Next vertex:

Select entities to delete: ↵

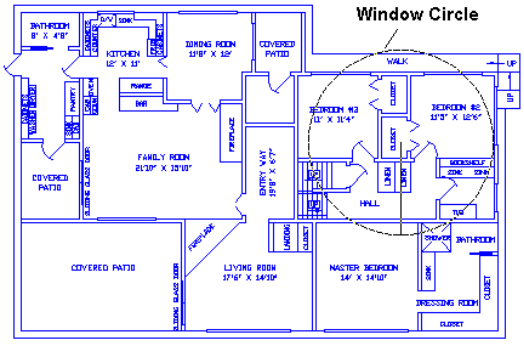
Properties (Pro): Selects all entities with specific properties from the drawing.

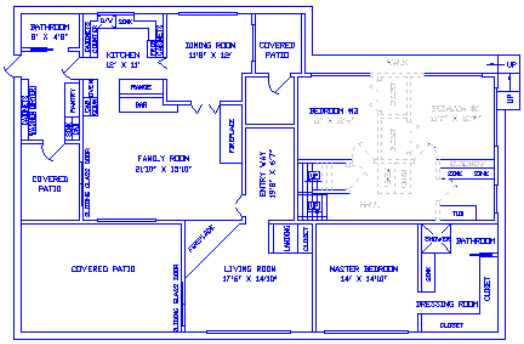


: Erase
Select entities to delete: pro

Select by properties: color
Color to select <BYLAYER>: blue
Select by properties:

Window Circle (WC): This option prompts users for Center and Radius of a Circle and selects all the entities within the circular area.





: Erase
Select entities to delete: wc

Center of circle:
Radius of circle:
Select entities to delete:

Alternate Commands

CADian has number of commands, which are not found in AutoCAD, though many of these commands have equivalent function in AutoCAD. User can very well invoke these commands in Cadian, by even giving the equivalent command of AutoCAD.

CADian Commands	AutoCAD Equivalent	Description
Coordinate	*F6 or Ctrl D*	*Controls display of Coordinates in Status bar.*
Delete	*Erase*	*Erases the selected entities.*
Dimension	*Dim*	*Switches to Dimension mode.*
EditLen	*Lengthen*	*Changes the length of an unclosed entity*
EditPline	*Pedit*	*Edits polylines or polymeshes*
EntProp	*Ddmodify/Ddchprop*	*Displays properties of Entities selected in a Dialog Box*
Esnap	*Osnap*	*Controls Entity Snapping*
Expblocks	*Ddinsert*	*Displays The CADian Explorer - Blocks.*
Expdimstyles	*Ddim*	*Displays The CADian Explorer - Dimension Styles*
Expfonts	*Ddstyle*	*Displays The CADian Explorer - Styles.*
Explayers	*Layer, Ddlmodes*	*Displays the CADian Explorer - Layers.*
Explorer	*Ddrename*	*Displays the CADian Explorer*
Expltypes	*Linetype ddltype*	*Displays the CADian Explorer - Linetypes.*
Expucs	*Dducs*	*Displays the CADian Explorer - UCS.*
Expviews	*Ddview*	*Displays the CADian Explorer - Views.*
Face	*3dface*	*Draws three-dimensional faces with three or four vertices.*
Flatten	***No equivalent***	***Sets thickness to zero at user specified Elevation.***
Font	*Style*	*Displays the Text Style dialog box.*
Freehand	*Sketch*	*Allows freehand sketching.*
Idpoint	*Id*	*Provides X, Y and Z Coordinates of a picked point*
Infline	*Xline*	*Draws a line of infinite length.*
Join	*Pedit join*	*Joins lines and arcs.*
Orthogonal	*Ortho*	*Toggles orthogonal mode.*

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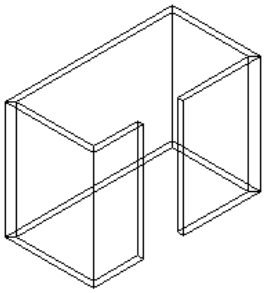
CADian Commands	AutoCAD Equivalent	Description
Mesh	*3dmesh*	*Draws a surface mesh.*
Parallel	*Offset*	*Copies an entity by a parallel offset Distance*
Plane	*Solid*	*Draws a two-dimensional solid plane.*
Pmthist	*F2*	*Switches between the Prompt History window and the graphics screen.*
Ppreview	*Preview*	*Plot Preview*
Qprint	***No Equivalent***	***Quickly prints the current view.***
Psetup	*Config*	*Displays the Print/Plot Setup dialog box.*
Recscript	***No Equivalent***	***Starts the Script Recorder.***
Saveall	***No Equivalent***	***Saves all currently open drawings.***
Setcolor	*Ddcolor*	*Displays the Color dialog box.*
Setdim	*Ddim*	*Displays the Dimension Settings Dialog box.*
Setesnap	*Osnap ddosnap*	*Displays the Drawing Settings dialog Box with the Coordinate Input tab Displayed.*
Setlayer	*ai_molc.*	*Sets the current layer based on the Selected entity's layer*
Settings	*No equivalent*	*Displays the Drawing Settings dialog Box.*
Setucs	*Dducs*	*Displays the User Coordinate Systems dialog box.*
Setvpoint	***No equivalent***	***Displays the Preset Viewpoints dialog Box.***
Stopscript	***No equivalent***	***No equivalent Stops running the script***
Tipofday	***No equivalent***	***Displays the Tip of the Day.***
Undelete	*Oops*	*Restores the last deleted entity.*
Viewpoint	*Vpoint*	*Sets the three-dimensional viewing Direction via the command line.*

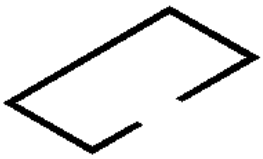
New Features

Flatten

Flatten will change the thickness of the selected entities to zero, and positions it at user specified elevation.



Pline With Thickness
before Flatten Command



Pline after Flatten Command

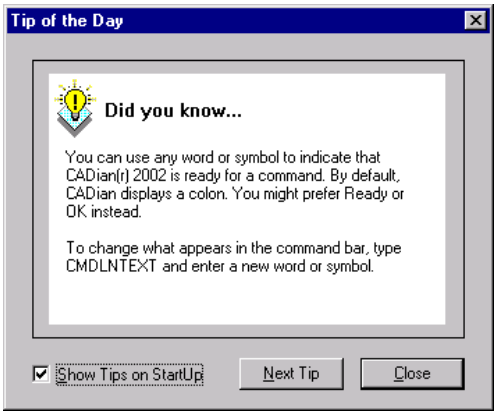
: Flatten
Select entities to flatten: *Pick Entities with thickness*
Select entities to flatten:

New UCS elevation <0.0000>: ↵

TipofDay

This command will display Cadian tips in a dialog box. Every time this command is given a new tip will be displayed.

: TipofDay



Next Tip: Will display the Next Tip of day

Show Tip on Startup: Will display the Tip when CADian is started.

SaveAll

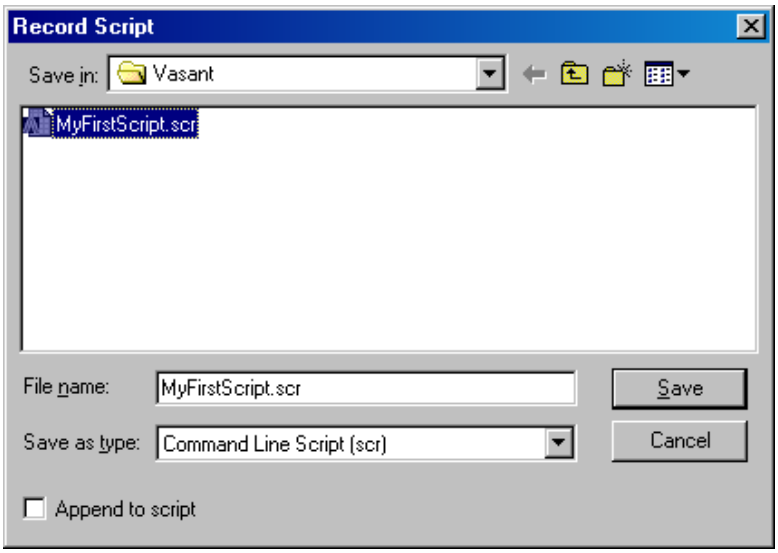
User can save all the drawings opened in the current session at one shot, by using the above command.
: SaveAll
:

Script

Script can be defined as collection of Commands of AutoCAD, which can be stored and run as and when needed.
Apart from having the script functionality, which works exactly similar to AutoCAD script, CADian provides user an additional facility of recording script. This command will automatically records all the commands keyed in by the user in a script file.

Script Recorder (RecScript)
This command initializes recording of Script. It will display dialog similar to save dialog box, prompt the user to give filename to save the script. This script file will be saved with an extension SCR. User can even append the existing script, by selecting an existing script file and clicking on 'Append to script' toggle box. If user selects an existing script file and fails to click on 'Append to script' toggle box before saving, then it is as good as overwriting the existing script.

: RecScript



Stop Script (StopScript)
Once the user starts recording the script. All the commands and user inputs given by user gets automatically recorded in a script file. This process will continue unless and until user terminates the recording of the script. The command StopScript terminates the recording of the script
: StopScript

Qprint

Qprint directly Prints/Plots the current display of the current drawing to the current Printer/Plotter, without prompting for user input.

: Qprint
:

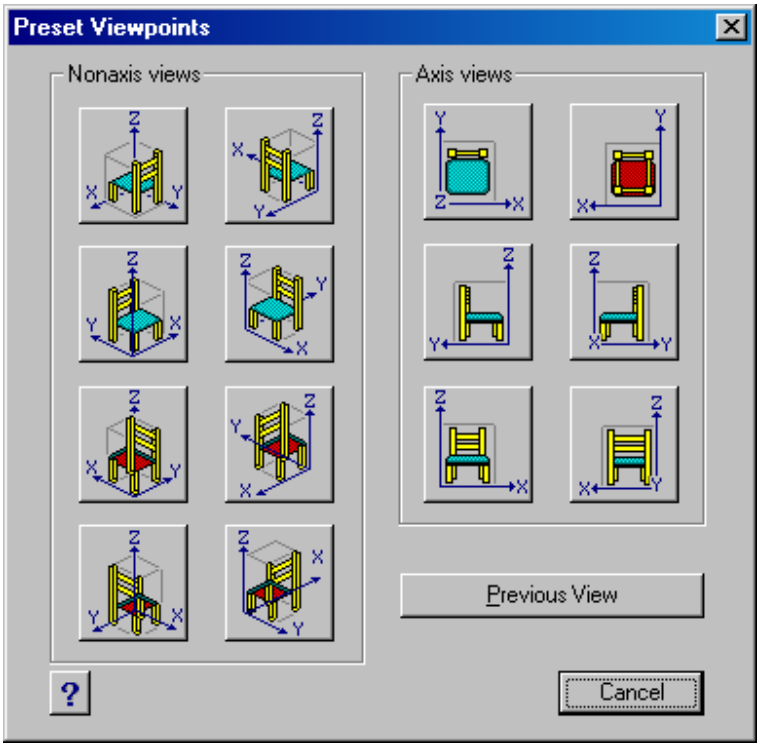
SetVPoint

To view a 3D Object from different views, we use command Vpoint. There are some standard views, which are used extensively by the users to view 3D object. SetVPoint helps user to switch to this standard viewpoint easily. SetVpoint displays a dialog box displaying all the standard views. The pictorial representation of each view helps user to choose the desired viewpoint.

The dialog is classified in three sections:

1. Non-axis views: Collection of standard views, where one can view all the three planes in equal proportion.
2. Axis views: Collection of standard views, where one can view all the two in equal proportion.
3. Previous views: Returns back to the previous view.

Command: SetVpoint
:



AutoCAD command not supported in CADian

Topic	Commands
File	Audit Recover Dxbin Wmfopts Dwfout Dwfoutd 3dsin 3dsout Psdrag Psfill Psin Psout
ACIS Solid	Ameconvert Soldraw Solprof Solview Stlout
ASE (SQL Extension)	Aseadmin Aseexport Aselinks Aserows Aseselec Asesqled
Internet	Attachurl Browser Detachur Inserturl Listurl Openurl Saveurl Selecturl
Render	Fog Matlib Replay Saveimg Scene Setuv Showmat Transparency

Landscape	Lsedit Lslib Lsnew
Others	Rtext Wipeout Arx Dsviewer Edge Mline Mledit Treestat Stats

AutoCAD objects not displayed in CADian

Complex linetypes: Complex components (text and shapes) are not displayed.

Proxy (zombie): Not displayed.

Images: Do not display if inside blocks and externally referenced drawings (Xrefs).

Arc aligned text: Text that is aligned along the curve of an arc.

Read text: Dynamically linked text that displays in a drawing but resides in an external file.

Wipeout masks: Masks to cover parts of drawings that you don't want plotted.

CADian Explorer

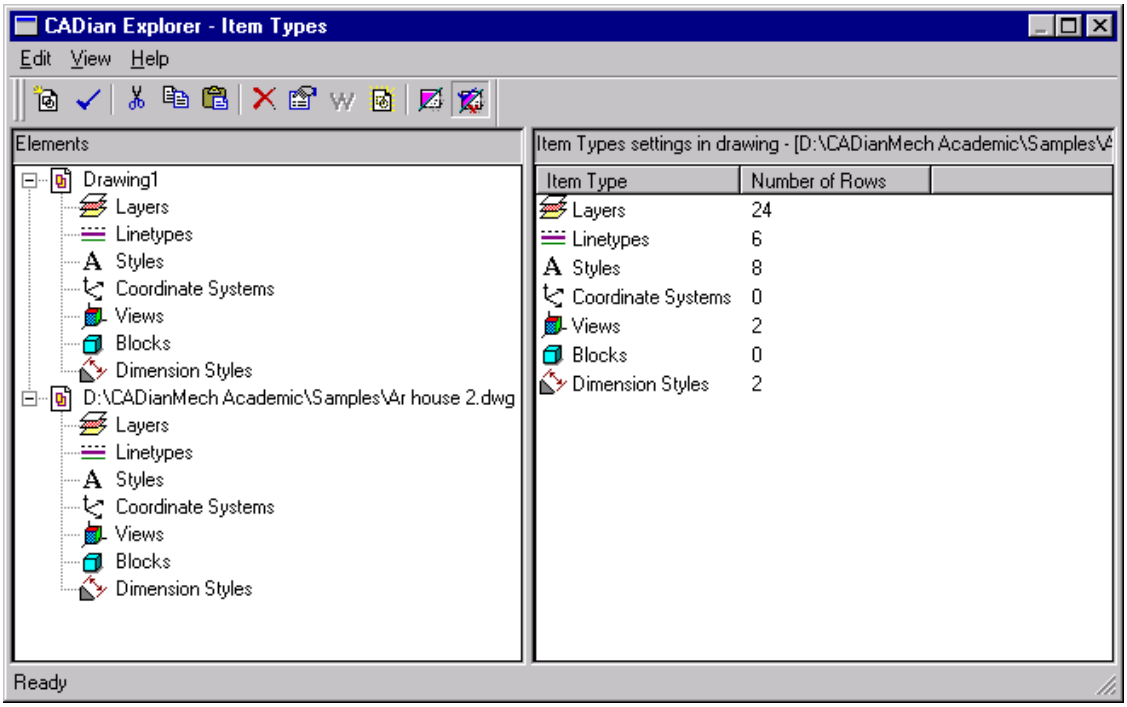
The CADian Explorer provides a powerful and convenient way to maintain and manage many of the features and settings of your drawings. You can use the CADian Explorer to work with layers, linetypes, text styles, coordinate systems, named views, blocks, and dimension styles within the current drawing or to copy this information between drawings.
We can use the CADian Explorer to:

- Manage elements related to settings and entities in your drawings.
- Organize information on layers and manage layers.
- Create and use linetypes.
- Load text fonts and create text styles.
- Select and control co ordinate systems.
- Save and Restore Named Views.
- Save insert and Manage Blocks.
- Copy, cut, and paste dimension styles between DWG files.

Manage elements related to settings and entities in your drawings.

: Explorer

:



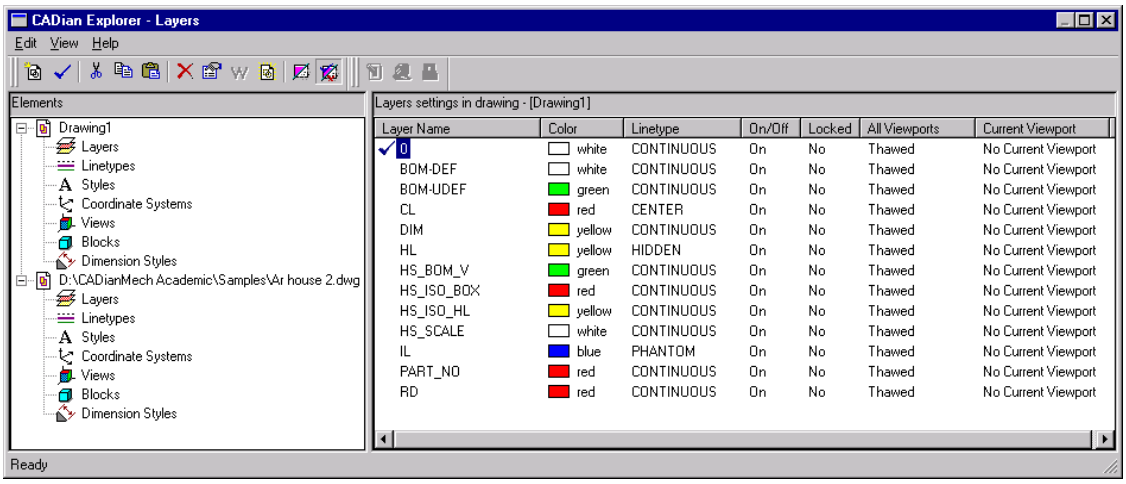
The CADian explorer will display properties of all the drawing files opened in the current session. User can double click on the properties under the drawing file trees structure to view all the entities corresponding to that property. Using CADian explorer you can edit or rename entities or even copy cut and paste the entities between the drawing files.

Organize information on layers and manage layers.

This option will display the layer properties of the current drawing in CADian explorer. The right hand side of the dialog will similar to the Layer dialog box in AutoCAD. A user can perform the following operations under this option:

- Create New Layer
- Rename a Layer
- Delete Layers
- Cut/Copy and Paste the Layers betweens drawings Files.
- Make a Particular Layer Current
- Edit Bylayer Color and Linetype properties for a particular Layer.
- Control Visibility of Layers by using On / Off or Freeze / Thaw options.
- Control Visibility of Layers in Viewports.
- Lock / Unlock layers for editing.

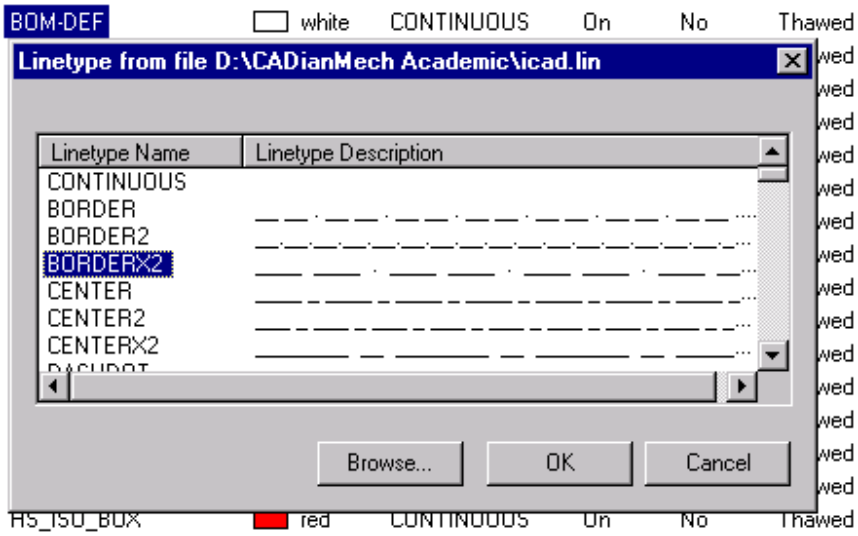
: Explayers or La
:
OR
Pull Down Menu: Setting => Explore Layers



You can perform the above stated operations by three ways:

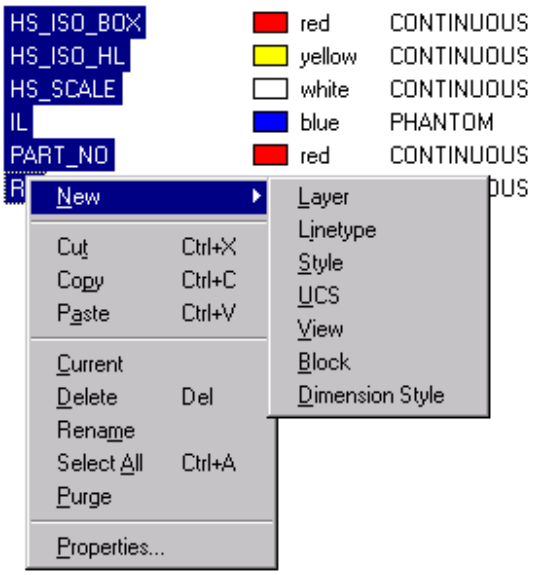
1. ***Click directly on the corresponding properties of the entities.***

   User can directly click on the layer or respective layer property you wish to edit (the way he used to do in AutoCAD).



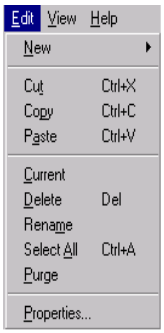
2. ***Using Right Click.***

   If user positions the cursor on top of the Layer or Layers (select using Shift or CTRL as in Windows) and then clicks the right button of the mouse, a cursor menu shown below will appear:



3. Using Pull Down Menu or Toolbars

User can use Edit and View Pull down menus to perform the same operations specified above. User can even click on corresponding toolbars.
Edit contains all the above options. The options under properties are present in the View menu as well. The submenus for both the menu items are displayed below:



The details of the above mentioned options with their Corresponding toolbars is given below:

New  ***:*** This option will be available for all the features in CADian Explorer. User can create new Layer by clicking on New => Layer option. It will automatically create a new layer with Default name as NewLayer1; the name will be highlighted for user to change it, if required.

Cut  ***:*** Cuts the layer and its properties, so that it can be pasted in a Different drawing.

Copies  ***:*** Copies the layer and its properties, so that it can be pasted in a Different drawing.

Paste  ***:*** Pastes the Layers in clipboard in the current expanded drawing in Cadian Explorer.

Current  ***:*** Sets the selected layer as current. User needs to select only one layer to make it current.

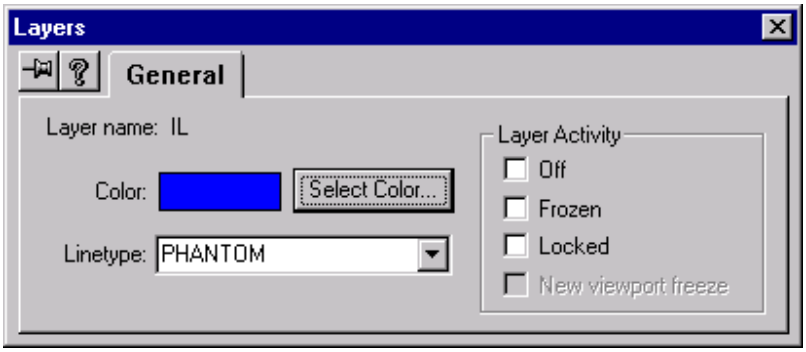
Delete  ***:*** Deletes the selected layers.

Rename: Renames the Selected Layer. User needs to select only one layer for renaming. Once user clicks on option rename the layer name will get highlighted for user to change the same.

Select All : Will select all the layers in the current selected drawing.

Purge  ***:*** Will purge all the unused layers. User will be prompted the layer names of the unused layers one by one. User can then decided whether he wishes to purge the corresponding layer by pressing Y for Purging or N for not purging. User can even purge all the layers at one shot by pressing A for Yes to All.

Properties  ***:*** Will display the layer properties dialog box as shown below:



Color: Will change the Bylayer Color of the selected of Layer/Layers

Linetype: Will change the Bylayer Linetype of the selected of Layer/Layers

Off  ***:*** Controls the visibility of the selected Layer/Layers. If Clicked then all the Layers Selected will be turned OFF and vice versa.

Frozen  ***:*** Controls the visibility of the selected Layer/Layers. If Checked then all the Layers Selected will be Frozen and vice versa.

Locked  ***:*** Controls the visibility of the selected Layer/Layers. If Checked then all the Layers Selected will be Locked for editing and vice versa.

Xref Layer Visibility : Controls display of Xrefed Layer in the Layer Dialog.

Create and use linetypes.

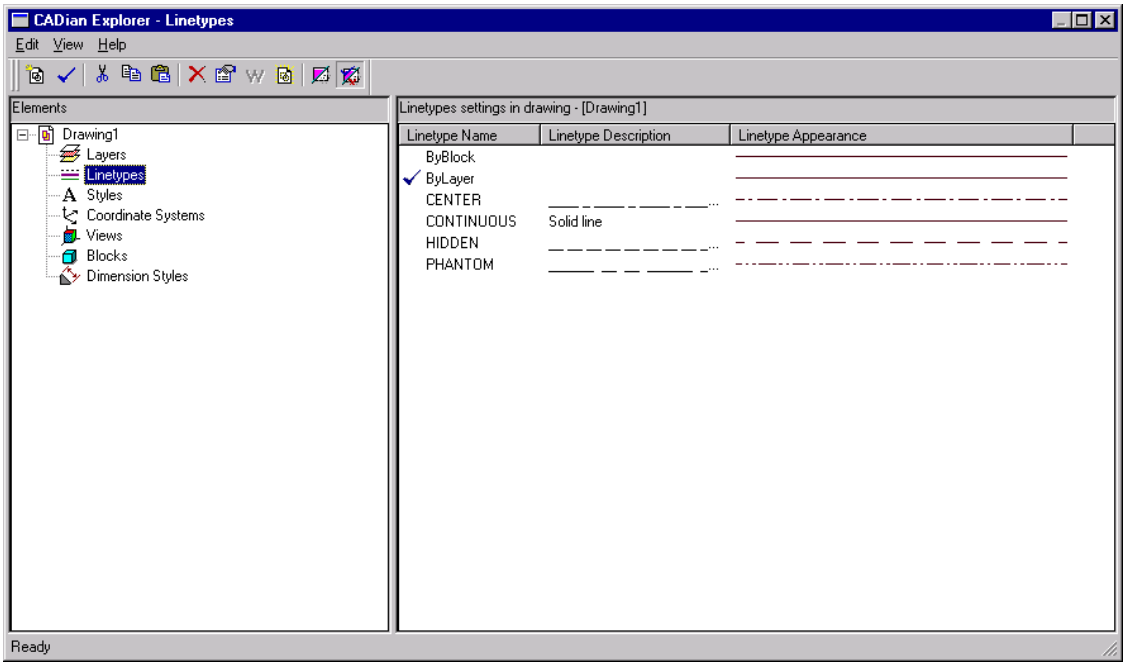
This option will display the Linetype properties of the current drawing in CADian explorer. The right hand side of the dialog will similar to the Linetype dialog box in AutoCAD. A user can perform the following operations under this option:

- Create New Linetype
- Rename a Linetype
- Delete Linetype
- Cut/Copy and Paste the Linetype betweens drawings Files.
- Make a Particular Linetype Current

: ExpLtypes or Lt
:

OR

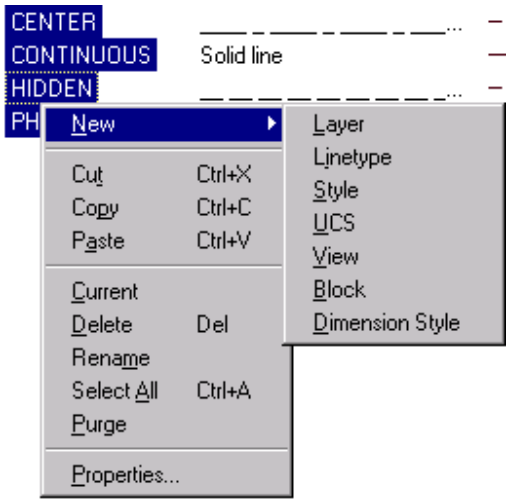
Pull Down Menu: Setting => Explore LineTypes



You can perform the above stated operations by three ways:

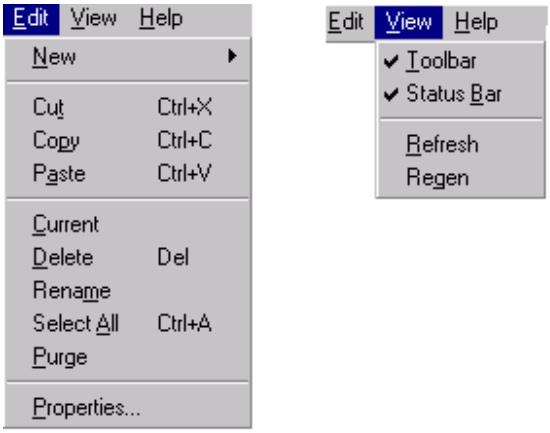
1. Using Right Click.

If user positions the cursor on top of the Linetype or Linetypes (select using Shift or CTRL as in Windows) and then clicks the right button of the mouse, a cursor menu shown below will appear:



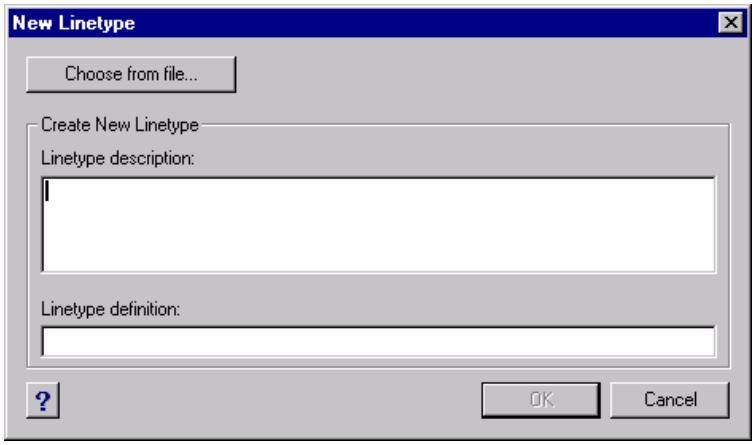
2. Using Pull Down Menu or Toolbars

User can use Edit and View Pull down menus to perform the same operations specified above. User can even click on corresponding toolbars.
Edit contains all the above options. The options under properties are present in the View menu as well. The submenus for both the menu items are displayed below:



The details of the above mentioned options with their Corresponding toolbars is given below:

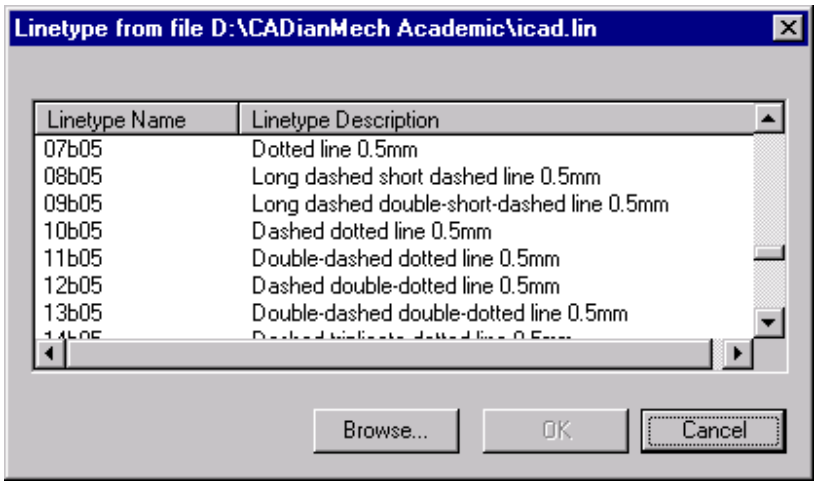
New  ***:*** User can create new Linetype by clicking on New => Linetype option. It will display dialog for creating a new Linetype.



If user wishes to create a new Linetype definition then he can give the Description and Linetype definition as displayed below.

It will create a new Linetype with default name as NewLinetype1 having specified description and Definition. The default name will be highlighted for user to change the same if he wishes to.

Choose From file: Allows user to choose the existing Linetypes which are not loaded. All the linetypes available will be displayed in a dialog box as shown below:



You can further browse to some other Linetype definition files by using browse option.

Cut  ***:*** Cuts the Linetype and its properties, so that it can be pasted in a Different drawing.

Copies  ***:*** Copies the Linetype and its properties, so that it can be pasted in a Different drawing.

Paste  ***:*** Pastes the Linetypes in clipboard in the current expanded drawing in Cadian Explorer.

Current  ***:*** Sets the selected Linetype as current. User needs to select only one Linetype to make it current.

Delete  ***:*** Deletes the selected Linetypes.

Rename: Renames the Selected Linetype. User needs to select only one Linetype for renaming. Once user clicks on option rename the Linetype name will get highlighted for user to change the same.

Select All : Will select all the Linetype in the current selected drawing.

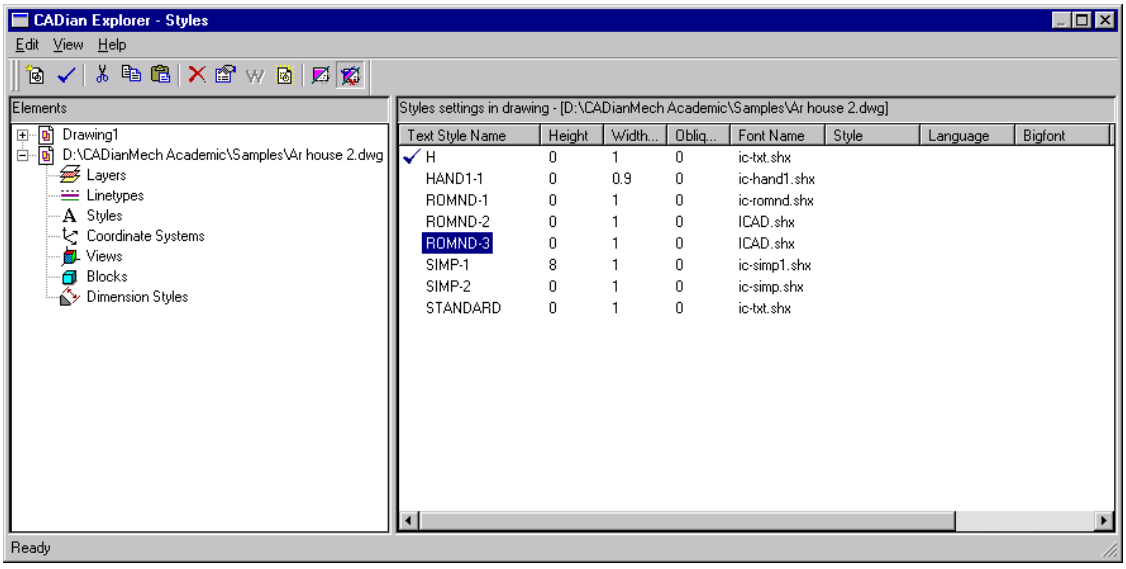
Purge  ***:*** Will purge all the unused Linetypes. User will be prompted the Linetype names of the unused Linetypes one by one. User can then decided whether he wishes to purge the corresponding Linetype by pressing Y for Purging or N for not purging. User can even purge all the Linetypes at one shot by pressing A for Yes to All.

Load text fonts and create text styles.

This option will display the Style properties of the current drawing in CADian explorer. The right hand side of the dialog will similar to the Style dialog box in AutoCAD. A user can perform the following operations under this option:

- Create New Style
- Rename a Style
- Delete Style
- Cut/Copy and Paste the Style betweens drawings Files.
- Make a Particular Style Current

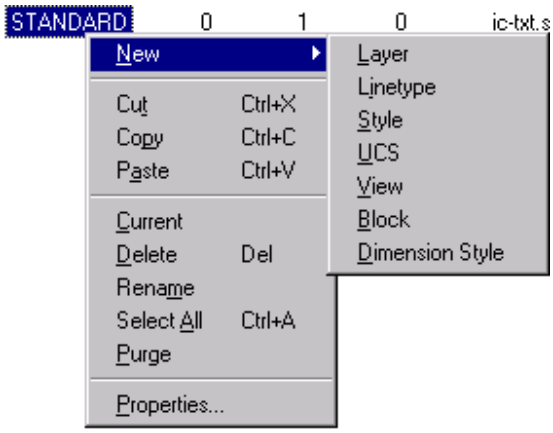
: Expstyles
:
OR
Pull Down Menu: Setting => Explore Styles



You can perform the above stated operations by three ways:

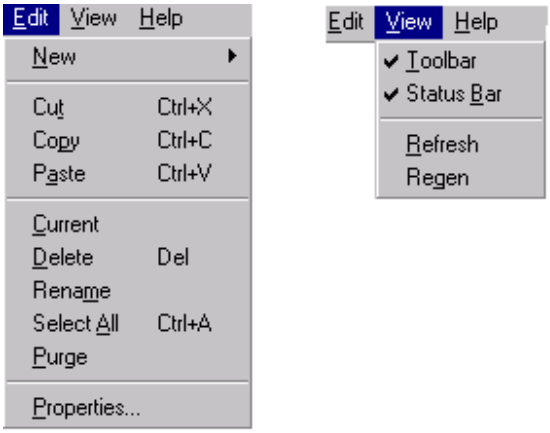
1. Using Right Click.

If user positions the cursor on top of the Style or Styles (select using Shift or CTRL as in Windows) and then clicks the right button of the mouse, a cursor menu shown below will appear:



2. Using Pull Down Menu or Toolbars

User can use Edit and View Pull down menus to perform the same operations specified above. User can even click on corresponding toolbars.
Edit contains all the above options. The options under properties are present in the View menu as well. The submenus for both the menu items are displayed below:



The details of the above mentioned options with their Corresponding toolbars is given below:

New  **:** This option will be available for all the features in CADian Explorer. User can create new Style by clicking on New => Style option. It will automatically create a new Style with Default name as NewStyle1, the name will be highlighted for user to change it, if required.

Cut  ***:*** Cuts the Style and its properties, so that it can be pasted in a Different drawing.

Copies  ***:*** Copies the Style and its properties, so that it can be pasted in a Different drawing.

Paste  ***:*** Pastes the Styles in clipboard in the current expanded drawing in Cadian Explorer.

Current  ***:*** Sets the selected Style as current. User needs to select only one Style to make it current.

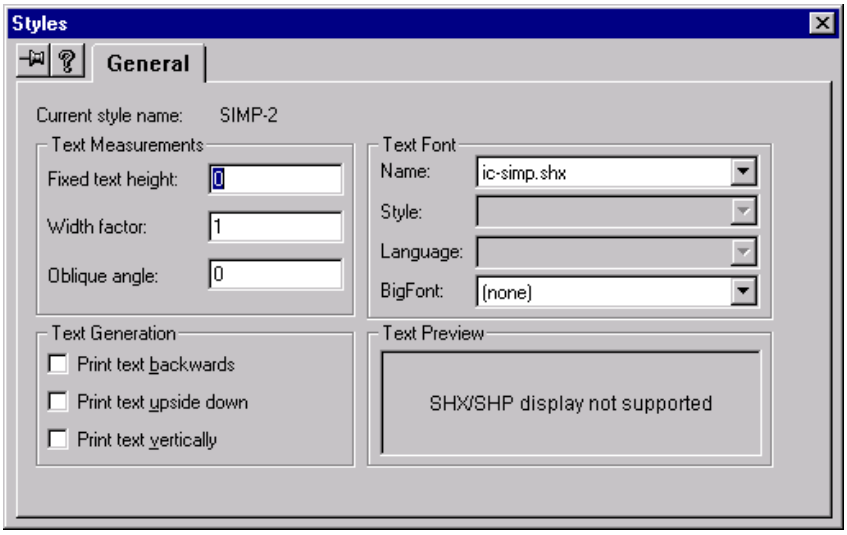
Delete  ***:*** Deletes the selected Styles.

Rename: Renames the Selected Style. User needs to select only one Style for renaming. Once user clicks on option rename the Style name will get highlighted for user to change the same.

Select All : Will Select all the Style in the current selected drawing.

Purge  ***:*** Will purge all the unused Styles. User will be prompted the Style names of the unused Styles one by one. User can then decided whether he wishes to purge the corresponding Style by pressing Y for Purging or N for not purging. User can even purge all the Styles at one shot by pressing A for Yes to All.

Properties **:** Will display the Style properties dialog box as shown below:



Text Measurement

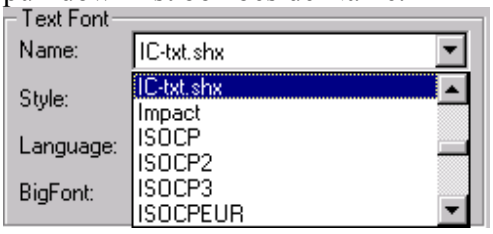
Fixed Text Height:
Sets the text height based on the value you enter. If you enter 0.0, CADian prompts for the text height each time you enter text using this style. Entering a height greater than 0.0 sets the text height for this style.

Width Factor
Sets the character Width ratio with respect to height of the text. Entering a value less than 1.0 condenses the text. Entering a value greater than 1.0 expands it.

Oblique Angle
Sets the oblique angle for the text under selected style. Oblique Angle will give text an effect of Italic Font.

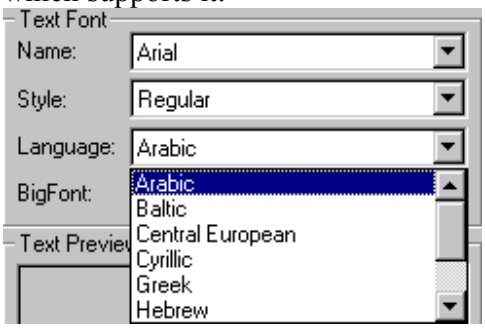
Text Font

Name
User can select the Font he wishes to assign from the list of predefined Fonts in the pull down list box beside Name.

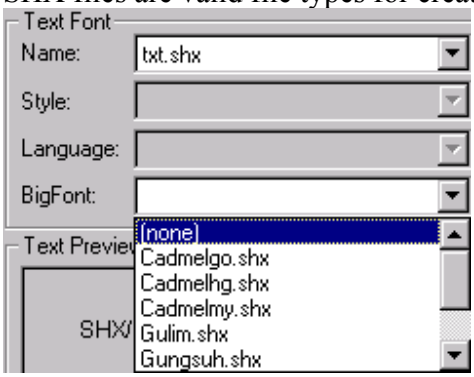


Style
Specifies font character formatting, such as Italic, oblique, bold, or regular. The style will be displayed only for the fonts, which supports it.

Language
Set the Language for the selected style. The Language is displayed only for the Font, which supports it.



Big Font
Use Big Font is available only if you specify an SHX file under Font Name. Only SHX files are valid file types for creating Big Fonts



Backwards
Displays the characters backwards.

Upside Down
Displays the characters upside down.

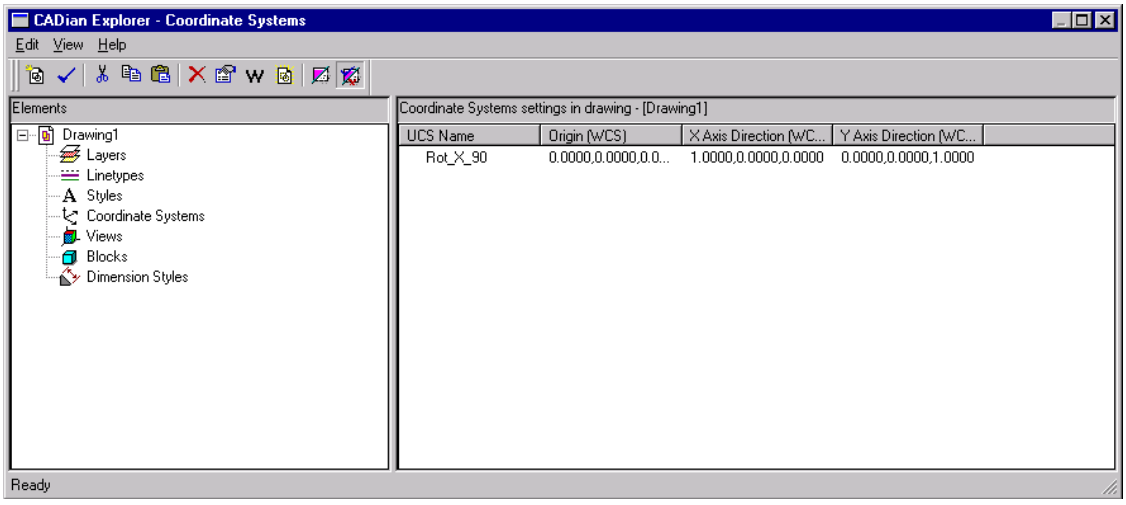
Vertical
Displays the characters aligned vertically. Vertical orientation is not available for TrueType fonts.

Select and control co ordinate systems (UCS).

This option will display the UCS properties of the current drawing in CADian explorer. The right hand side of the dialog will similar to the UCS dialog box in AutoCAD. A user can perform the following operations under this option:

- Create New UCS
- Rename a UCS
- Delete UCS
- Cut/Copy and Paste the UCS betweens drawings Files.
- Make a Particular UCS Current

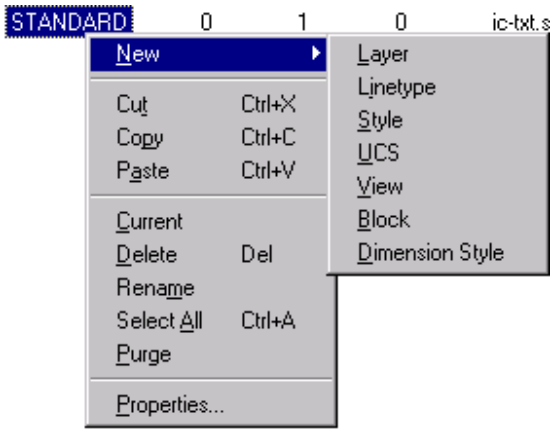
: ExpUCS or DDUCS
:
OR
Pull Down Menu: Setting => Explore UCSs



You can perform the above stated operations by three ways:

1. Using Right Click.

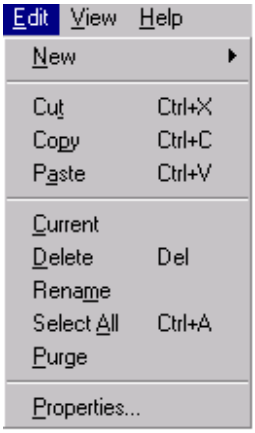
If user positions the cursor on top of the UCS or UCS's (select using Shift or CTRL as in Windows) and then clicks the right button of the mouse, a cursor menu shown below will appear:

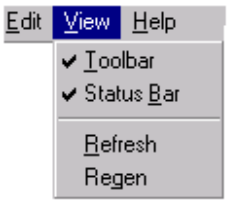


2. Using Pull Down Menu or Toolbars

User can use Edit and View Pull down menus to perform the same operations specified above. User can even click on corresponding toolbars.

Edit contains all the above options. The options under properties are present in the View menu as well. The submenus for both the menu items are displayed below:





The details of the above mentioned options with their Corresponding toolbars is given below:

New : This option will be available for all the features in CADian Explorer. User can create new UCS by clicking on New => UCS option. It will display the prompt of UCS command on command prompt, once specifies the UCS, it automatically create a new UCS with Default name as NewUCS1, the name will be highlighted for user to change it, if required.

Cut : Cuts the UCS and its properties, so that it can be pasted in a Different drawing.

Copies : Copies the UCS and its properties, so that it can be pasted in a Different drawing.

Paste : Pastes the UCS's in clipboard in the current expanded drawing in Cadian Explorer.

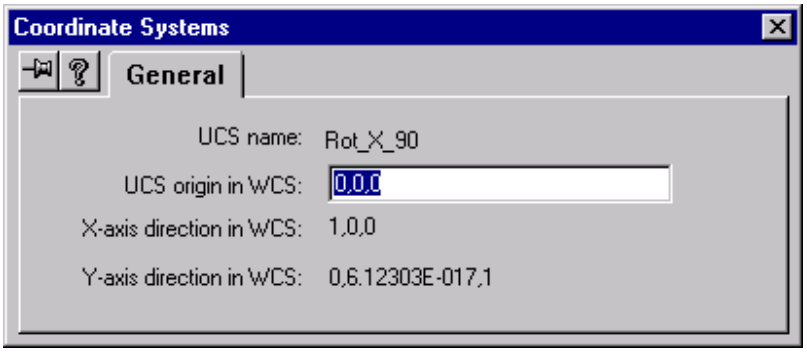
Current : Sets the selected UCS as current. User needs to select only one UCS to make it current.

Delete : Deletes the selected UCS's.

Rename: Renames the Selected UCS. User needs to select only one UCS for renaming. Once user clicks on option rename the UCS name will get highlighted for user to change the same.

Select All : Will select all the UCS in the current selected drawing.

Properties : Will display the UCS properties dialog box as shown below:



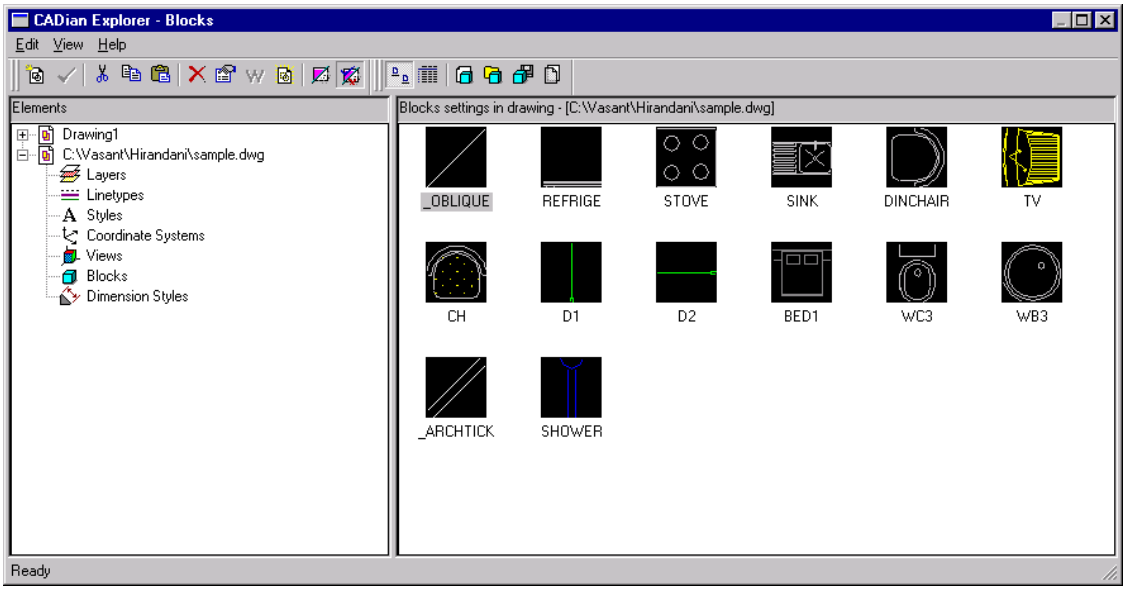
UCS Origin: Specifies the new origin for the selected UCS.

Save insert and Manage Blocks.

This option will display the Block properties of the current drawing in CADian explorer. The right hand side of the dialog will similar to the Block dialog box in AutoCAD. A user can perform the following operations under this option:

- Create New Block
- Rename a Block
- Delete Block
- Cut/Copy and Paste the Block betweens drawings Files.
- Make a Particular Block Current

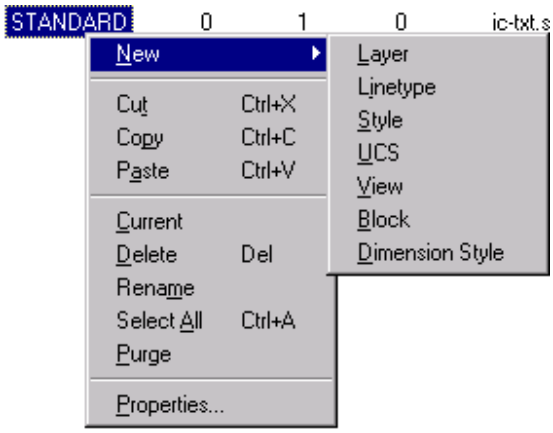
: ExpBlock or DDBlock
:
OR
Pull Down Menu: Setting => Explore Blocks



You can perform the above stated operations by three ways:

3. Using Right Click.

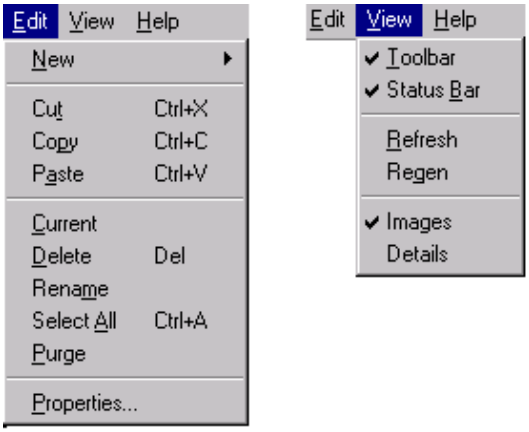
If user positions the cursor on top of the Block or Blocks (select using Shift or CTRL as in Windows) and then clicks the right button of the mouse, a cursor menu shown below will appear:



4. Using Pull Down Menu or Toolbars

User can use Edit and View Pull down menus to perform the same operations specified above. User can even click on corresponding toolbars.

Edit contains all the above options. The options under properties are present in the View menu as well. The submenus for both the menu items are displayed below:



Details: To view only the details of the block.
Image: To view the preview image of Blocks in the Explorer

The details of the above mentioned options with their Corresponding toolbars is given below:

New  **:** This option will be available for all the features in CADian Explorer. User can create new Block by clicking on New => Block option. It will display the prompt of Block command on command prompt, once specifies the Block, it automatically create a new Block with Default name as NewBlock1, the name will be highlighted for user to change it, if required.

Cut  **:** Cuts the Block and its properties, so that it can be pasted in a Different drawing.

Copies  **:** Copies the Block and its properties, so that it can be pasted in a Different drawing.

Paste  **:** Pastes the Blocks in clipboard in the current expanded drawing in Cadian Explorer.

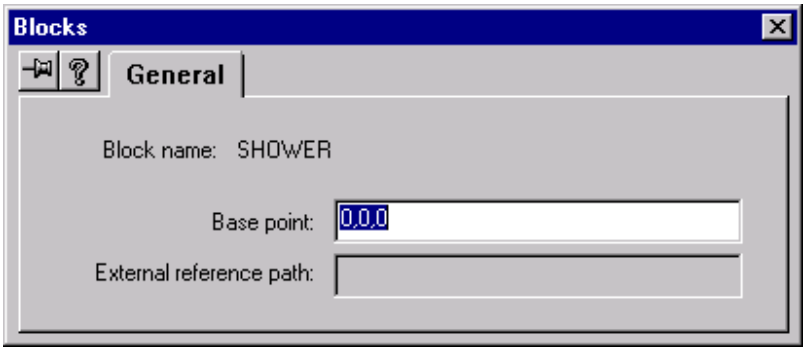
Current  **:** Sets the selected Block as current. User needs to select only one Block to make it current.

Delete  **:** Deletes the selected Blocks.

Rename: Renames the Selected Block. User needs to select only one Block for renaming. Once user clicks on option rename the Block name will get highlighted for user to change the same.

Select All : Will select all the Blocks in the current selected drawing.

Properties  **:** Will display the Block properties dialog box as shown below:



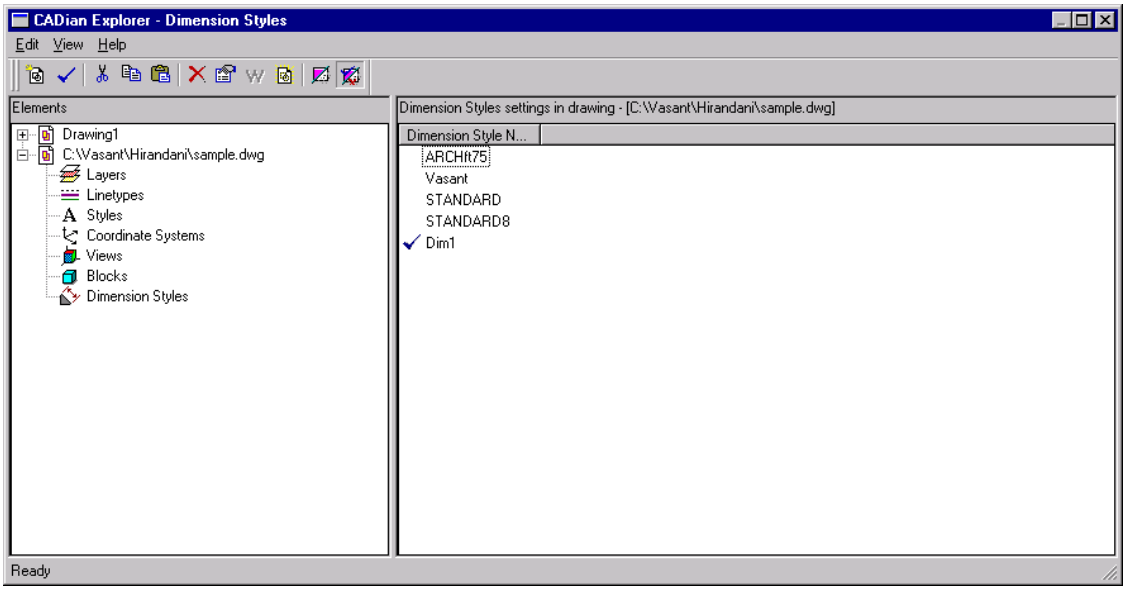
Base Point: Specifies the new base point for the selected Block.

Copy, cut, and paste dimension styles between DWG files

This option will display the Dimension style properties of the current drawing in CADian explorer. The right hand side of the dialog will similar to the Dimension style dialog box in AutoCAD. A user can perform the following operations under this option:

- Create New Dimension style
- Rename a Dimension style
- Delete Dimension style
- Cut/Copy and Paste the Dimension style betweens drawings Files.
- Make a Particular Dimension style Current

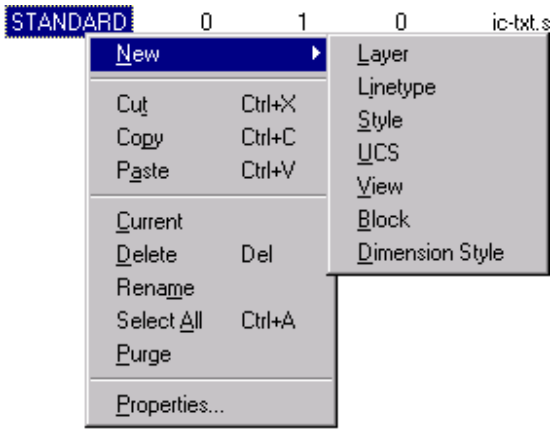
: ExpDimstyle style or DDim
:
OR
Pull Down Menu: Setting => Explore Dimension styles



You can perform the above stated operations by three ways:

5. Using Right Click.

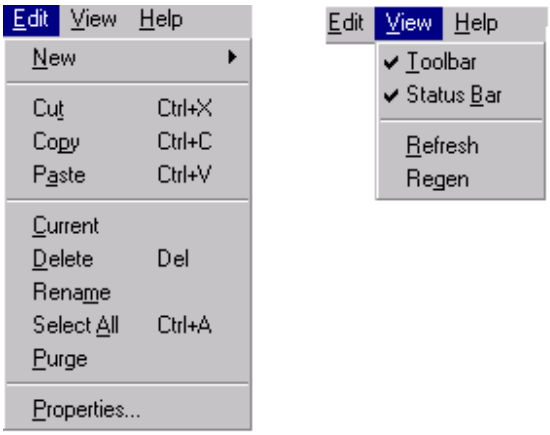
If user positions the cursor on top of the Dimension style or Dimension styles (select using Shift or CTRL as in Windows) and then clicks the right button of the mouse, a cursor menu shown below will appear:



6. Using Pull Down Menu or Toolbars

User can use Edit and View Pull down menus to perform the same operations specified above. User can even click on corresponding toolbars.

Edit contains all the above options. The options under properties are present in the View menu as well. The submenus for both the menu items are displayed below:



The details of the above mentioned options with their Corresponding toolbars is given below:

New : This option will be available for all the features in CADian Explorer. User can create new Dimension style by clicking on New => Dimension style option. It automatically create a new Dimension style with Default name as NewDimension style1, the name will be highlighted for user to change it, if required.

Cut : Cuts the Dimension style and its properties, so that it can be pasted in a Different drawing.

Copies : Copies the Dimension style and its properties, so that it can be pasted in a Different drawing.

Paste : Pastes the Dimension styles in clipboard in the current expanded drawing in Cadian Explorer.

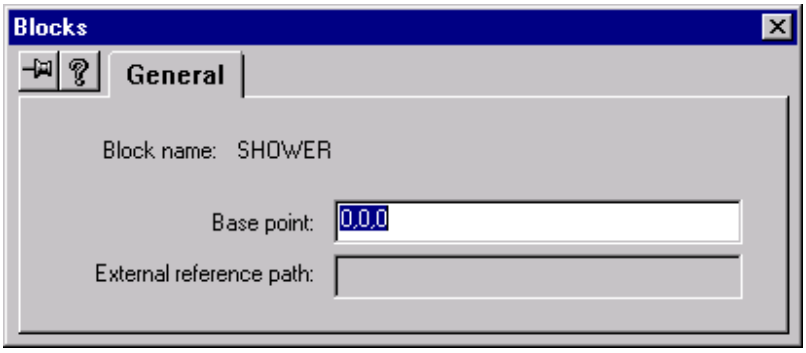
Current : Sets the selected Dimension style as current. User needs to select only one Dimension style to make it current.

Delete : Deletes the selected Dimension styles.

Rename: Renames the Selected Dimension style. User needs to select only one Dimension style for renaming. Once user clicks on option rename the Dimension style name will get highlighted for user to change the same.

Select All : Will select all the Dimension style in the current selected drawing.

Properties : Will display the Dimension style properties dialog box as shown below:



Base Point: Specifies the new base point for the selected Dimension style.

Entity Property Dialog

Every object you draw has a set of inherent properties including layer, Linetype, lineweight etc.

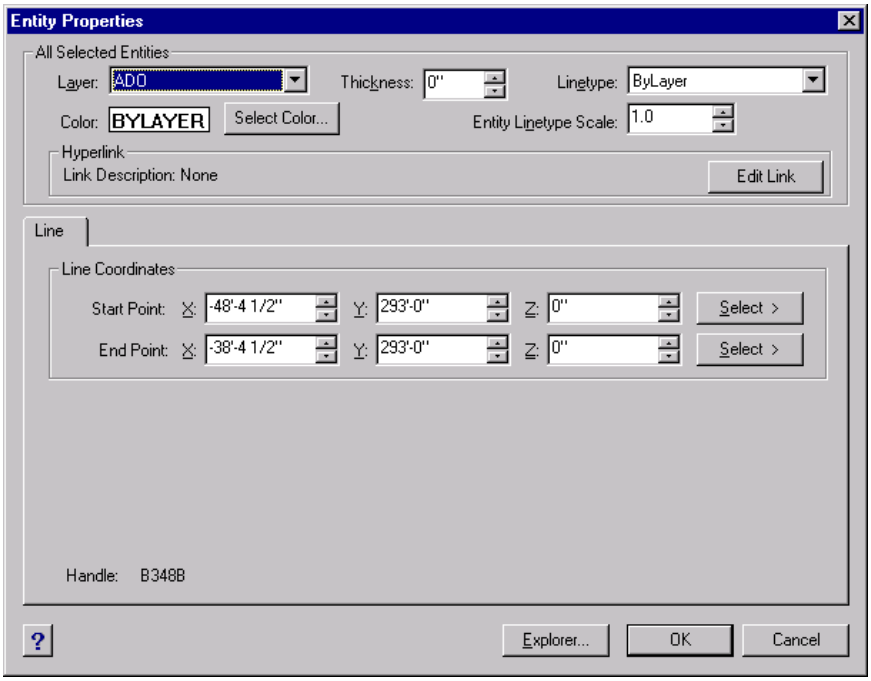
The properties can be broadly classified in two ways:

1. General Properties: Like Color, Layer Linetype, and Lineweight etc.
2. Individual Properties: This property will differ from one entity type to another, like Center and Radius for Circle, and Start point and End point for Line and so on.

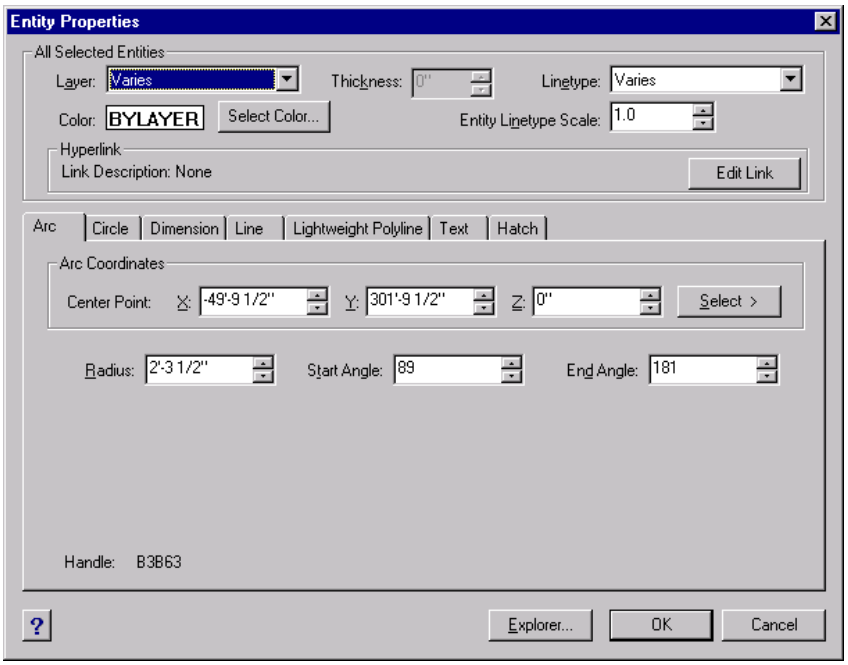
You can change Entity Properties by any of the following method:

:Ch↵ *OR* Mo↵ *OR* Ddmodify↵ *OR* EntProp↵
Select Objects: *Select entities you wish to modify*
Select Objects: ↵
:
Or
From Pull Down Menu: Modify => Properties
Or
By double clicking on an Entity
OR
Select Entities using Grips; give a right click and then click on the Properties.

Entity Property Dialog Box will appear, which is shown below:



If you have selected more than one entity the dialog will appear as follows:



The upper Half of the Entity Properties deals with general Properties; hence it will remain command to all the Entity Type. The other half deal with properties specific to Entity Types, hence it will from one Entity Type to another.

Layer: Will display all the Existing Layers in Pull down List beside Layer Text. if user selects a layer, then all the entities selected will be transferred in the selected Layer.

Linetype: Will display all the Existing Linetypes in Pull down List beside Linetype Text. if user selects a Linetype, then the selected Linetype property will be assigned to all the Entities selected.

Color: Will display all the Existing Colors in Pull down List beside Color Text. if user selects a Color, then the selected Color property will be assigned to all the Entities selected.

Thickness: if user sets thickness to value other than 0, then all the entities selected will have the thickness provided by the user.

LTScale: Changes the LTScale for the selected entities.

LineWeight: Will display all the Existing Lineweights in Pull down List beside Lineweight Text. If user selects a Lineweight, then the selected Lineweight property will be assigned to all the Entities selected.

HyperLink: Will edit the Hyperlink assigned to selected objects.

Entity Properties will vary from one Entity Type to another. A table is given below displaying the properties displayed for different Entity Types.

Entity Type	Properties
Line	Start Point
	End Point
Circle	Center Point
	Radius
Arc	Center
	Radius
	Start Angle
	End Angle
Text	Start Point
	Justification
	Height
	Rotation
	Width Factor
	Oblique angle
	Style
	Text
	Backward
	Upside Down
LWPolyline (For Each Vertices)	Continuous Linetype
	Close Polyline
	Polyline Elevation
	Location
	Starting and Ending Width
	Bulge
	Next Vertex or Previous Vertex
Hatch	Hatchedit
Dimension	Dimension Text
	Rotation
	Insertion Point
	Dimension Settings

Supporting AutoCAD Customization Files

The list of AutoCAD Customization files supported in CADian is given below:

LIN	Supported: Linetypes. Partially supported: Does not display text and shapes in complex linetypes.
MNU and MNS	Supported: Toolbar and Pop-up menus and macros. Not supported: Tablet, Buttons, Screen, Aux, Icon menus.
MLN	Not supported: The multiline definition file is used by the AutoCAD *mline* command.
PAT	Supported: Hatch patterns.
PGP	Supported: Command aliases. Not supported: External commands.
PSF	Not supported: PostScript fill pattern file is used by the AutoCAD *psfill* command.
SHP and SHX	Supported: Text fonts. Not supported: Shapes.
SLD	Supported: Slide files.
UNT	Supported: Unit translation file used by the LISP (cvunit) and SDS sds_cvunit functions to translate values from one unit of measurement to another.

Menu Compatibility

CADian reads both mnu and mns menu file formats of AutoCAD, even when menu macros include AutoLISP code. User can load and work on his existing AutoCAD menus. Following menus sections are supported by CADian.

Menu section	Definition	CADian support
***POP0	Cursor menu	Supported
***POP*n*	Pull-down menus	Supported
***AUX*n*	Auxiliary menus	Not supported
***BUTTON*n*	Button menus	Not supported
***ICON	Icon menus	Not supported
***SCREEN	Screen menus	Not supported
***TABLET*n*	Tablet menus	Not supported

Importing and exporting customization files

You can continue using aliases and menu files from AutoCAD by importing the appropriate file. You can import AutoCAD customization files and export CADian formats using the Customize dialog box. All of the files listed in the following table are in ASCII format, which means you can view and edit them with a text editor, such as Notepad.

Customizing files

Customization	Import file formats	Export file formats
Aliases	PGP: AutoCAD aliases ICA: CADian aliases ICA: CADian aliases	PGP: AutoCAD aliases
Keyboard	ICK: CADian keyboard	ICK: CADian keyboard
Menus	MNU: All AutoCAD menus ICM: CADian menu	ICM: CADian menu
Toolbars	MNU: All AutoCAD menus	None

Programming in CADian

CADian® 2001 supports more AutoCAD application programming interfaces (APIs) than any other software, but not all of the AutoCAD APIs are available in CADian. The following table summarizes the AutoCAD APIs CADian supports.

CADian support of the AutoCAD programming interface

AutoCAD programming interface	CADian support
Scripts (SCR files)	Supported
AutoLISP (LSP files)	Supported
Dialog Control Language (DCL files)	Supported
AutoCAD Development System (ADS)	Supported; source code must be recompiled
Direct Interactively Evaluated String Expression Language (Diesel)	Not supported
AutoCAD SQL Interface (ASI)	Not supported
Autodesk Threaded Language Application System Toolkit (Atlast)	Not needed
AutoCAD Runtime Extension (ARx)	Not supported

Understanding AutoLISP compatibility

CADian adds LISP functionality that you will find useful. The following table lists functions that are unique to CADian LISP.

Additional CADian LISP functions

Unique LISP function	Definition
(log10)	Returns log base 10.
(lpad)	Pads a text string with spaces to the left.
(rpad)	Pads a text string with spaces to the right.
(tan)	Returns the tangent.
(trim)	Trims spaces from a string.

Not all CADian LISP functions are completely compatible with AutoLISP functions. The following table identifies CADian LISP functions that are partially compatible with AutoLISP functions.

Partially compatible LISP functions

LISP function	Description
(menucmd)	Supports *P0* (cursor menu) and *P1* through *P16* (the pull-down menus), but does not support *A* (aux menus), *B* (button menus), *I* (icon menus), *M* (diesel expressions), *S* (screen menu), or *T* (tablet menus).
(osnap)	Supports an additional entity snap, pla, for planview (two-dimensional intersection).
(print1)	Does not support Unicode characters, such as \U+00B0 (the degree symbol) and M+Nxxxx (multibyte Unicode sequences).
(ssget) and *(ssadd)*	Supports additional selection modes: CC = Crossing Circle O = Outside OC = Outside Circle OP = Outside Polygon PO = POint