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Arc-1 command

Use the Arc-1 command to draw an arc if you know the points for the center and radius of the arc.

Angle

The angle is entered in Drawing Units in the Angle box in the Command Line.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Same as

Use the Same as button to have the arc drawn with the same angle as another arc in the drawing.

Shortcuts

Toolbox:



Keys: **A**

See Also: [Drawing Arcs](#), [Drawing Arc with Center and Radius](#)

Same As Button

The Same as button draws an entity using the same options as an entity already on the screen.

To do this, select the Same as button in the Command Line. Then select the entity in the drawing with the options you want. The options in the Command Line will change to reflect those of the selected entity.

Vector Entity

A vector entity is made up of straight line segments. These individual line segments can be seen by selecting the entity with the Point Select Mode from the Toolbox.

A vector entity is drawn when the Save as vector form box is selected with various DesignCAD commands.

Save As Vector Form Box

The Save as vector form box saves a drawing entity as a vector entity.

When an entity is saved in its vector form, the individual points used to make up the entity can be moved or deleted.

Vector

A vector is a straight line segment used to construct a vector entity. These individual line segments can be seen by selecting the entity with the Point Select Mode from the Toolbox.

A vector is drawn with the Line command.

Arc-2 command

Use the Arc-2 command to draw an arc using points set for the beginning and end of the arc.

Radius

The radius is entered in Drawing Units in the Radius box in the Command Line.


Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Same as

Use the Same as button to have the arc drawn with the same radius as another arc in the drawing.

Shortcut

Mouse: 

See Also: Drawing Arcs, Drawing Arc with Beginning and End


Arc-3 command

Use the Arc-3 command to draw an arc using any three points.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Shortcut

Mouse: 

See Also: [Drawing Arcs](#), [Drawing Arc with Three Points](#)


Arc-4 command

Use the Arc-4 command to draw an arc using points set for the center, beginning, and end of the arc.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Shortcut

Mouse: 

See Also: Drawing Arcs, Drawing Arc with Center, Beginning, and End


Arc-5 command

Use the Arc-5 command to draw an arc using points set for the beginning, end, and radius of the arc.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Shortcut

Mouse: 

See Also: Drawing Arcs, Drawing Arc with Beginning, End, and Radius

Arrange Icons command

Use the Arrange Icons command to straighten any icons on the screen and place them along the bottom of the screen. Different views of the drawing window become icons when they are minimized. These icons can be moved around the screen.

See Also: [Arranging Windows and Icons](#)

Icons

Icons are little graphical symbols or buttons illustrating different DesignCAD windows. When a DesignCAD drawing window is minimized with the Minimize button, it becomes an icon. The drawing window is maximized by double-clicking the icon.

View

A view is a window used by DesignCAD to display a section or part of the current drawing. Entities can be added, deleted, or edited in the views. The views can be saved as separate drawings.

Window

A window is a separate view of a drawing.

Arrow command

Use the Arrow command to draw an arrow through several points. The arrowhead is drawn at the last point.

Arrowhead Size

The size of the arrowhead is entered in the Arrowhead size box in the Command Line.


Arrowhead Type

The arrowhead type is selected in the Arrowhead Type box in the Command Line.

Same as

Use the Same as button to have the arrow drawn with the same size or type of arrowhead as another arrow in the drawing.

Shortcuts

Mouse: 

Keys: >

See Also: [Drawing Regular Lines](#)

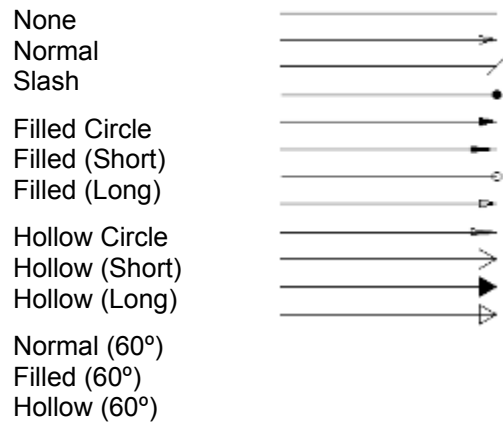
Arrowhead

An arrowhead is drawn at the end of a line drawn with the Arrow command. The arrowhead type and arrowhead size can be changed in the Command Line at the top of the screen.

Arrowhead Types

The Arrow and Dimension commands draw arrows through a set of points. At the end of the arrow is the arrowhead.

There are 12 arrowhead types available, these are:



Arrowhead Size Box

The Arrowhead Size box in the Command Line allows you to change the size of an arrowhead.

Attribute command

Use the Attribute command to assign information to an object in the drawing. This information can later be extracted from the drawing file and used with other programs for applications such as parts lists and bills of materials.

Text

The text for the attribute is entered in this box in the Command Line. Up to 80 characters can be entered.

Size


The size of the attribute is entered in the Size box in the Command Line.

Show attributes

Attributes will not be displayed on the screen unless the Show Attribute option is selected in the Options menu or the Show attributes box is selected in the Command Line. Whether the attribute is displayed on the screen or not, it will be copied, deleted, etc., as a normal entity.

Attributes can be extracted and used by the Material List command. This command creates a list of all attributes in a drawing and outputs this information to the printer or a disk file for use by other applications.

Shortcuts

Mouse: 

Keys: \$

See Also: Drawing Attributes, Show Attribute option

Bill of Materials

The Bill of Materials is a list of attributes in a drawing generated by the Material List command.

Balloon command

Use the Balloon command to add information to a drawing in a text balloon.

Text

The text is entered in the Text box in the Command Line.

Balloon Size

The size of the balloon drawn around the text is entered in the Balloon Size box in the Command Line.


Dimension Line button

The arrowhead type and size can be set by choosing the Dimension Line button in the Command Line.

Same as

Choose the Same as button to have the balloon drawn with the same arrowhead and text size as another balloon in the drawing.

Shortcut

Mouse: 

See Also: [Drawing a Balloon around Information](#)


Bezier Curve command

Use the Bezier Curve command to draw a bezier curve through four points. The second and third points are control points.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Shortcut

Mouse: 

See Also: [Drawing Bezier Curves](#)

Control Points

Control points are used with the Bezier Curve command to determine the tangents of the curve at the endpoints. The farther the control points are from the first and last points, the more they influence the curve.


Box command

Use the Box command to draw a box or rectangle defined by two points.

Align to any angle

The box can be drawn aligned to any angle by selecting the Align to any angle box in the Command Line. When this box is selected, two points are set for the first side of the box. The points can be set at any angle. A third point is set for the opposite edge of the box.

Shortcuts

Mouse: 


Keys:]

See Also: [Drawing Simple Polygons](#), [Drawing a Box](#)

Break Line command

Use the Break Line command to break a line entity consisting of two or more line segments into separate line entities. Each of the lines will be defined by two endpoints.

Shortcuts

Mouse: 

Keys: |

See Also: [Breaking a Line](#), [Breaking Lines](#)

Cascade command


Use the Cascade command to arrange all open views on the screen so that a small amount of each window is visible.

See Also: [Cascading Windows](#)

Center of Gravity command

Use the Center of Gravity command to find the geographic center of an entity.

Shortcut


Mouse: 

See Also: [Choosing Point Commands](#), [Setting Points](#), [Center of Gravity Move command](#)

Center of Gravity Move command

Use the Center of Gravity Move command to move the pointer to the geographic center of an entity without setting a point.

Shortcut

Mouse: 

See Also: [Choosing Point Commands](#), [Setting Points](#), [Center of Gravity command](#)

Chamfer command


Use the Chamfer command to "cut off" the corner of a line or arc entity with a straight line.

The "chamfer depth" is entered in the Chamfer box in the Command Line.

Keep original lines

The original lines used to make up the corner of the object will be removed, unless the Keep original lines box in the Command Line is selected.

Shortcuts

Mouse: 

Keys: **Ctrl+F**

See Also: [Chamfering Lines](#)

Chamfer Depth

The chamfer depth is the depth, or radius, of the line that cuts off the corner of the entity.


Circle-1 command

Use the Circle-1 command to draw a circle using points set for the center and radius.

Save as vector form

Selecting the [Save as vector form box](#) in the Command Line saves the circle as a [vector entity](#).

Shortcuts

Mouse: 

Keys: **O**

See Also: [Drawing Circles](#), [Drawing Circle with Center and Radius](#)


Circle-2 command

Use the Circle-2 command to draw a circle using two points set for the diameter.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the circle as a vector entity.

Shortcut

Mouse: 

See Also: [Drawing Circles](#), [Drawing Circle with Diameter](#)


Circle-3 command

Use the Circle-3 command to draw a circle by setting any three points.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the circle as a vector entity.

Shortcut

Mouse: 

See Also: [Drawing Circles](#), [Drawing Circle with Three Points](#)

Circle-4 command

Use the Circle-4 command to draw a circle with a point set for the center.

Radius

The radius length is entered in Drawing Units in the Radius box in the Command Line.


Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Same as

Use the Same as button to have the circle drawn with the same radius as another circle in the drawing.

Shortcut

Mouse: 

See Also: [Drawing Circles](#), [Drawing Circle with Center and Radius Length](#)

Circular Array command

Use the Circular Array command to duplicate selected objects a certain number of times in a circle or arc.

No. of copies

The number of copies to be duplicated is entered in the No. of copies box in the Command Line.

Angle

The angle of the circle or arc is entered in the Angle box in the Command Line.


Use original as the 1st copy

If the Use original as 1st copy box is selected in the Command Line, the original entity will be used as the location of the first copy.

Fixed handle

Use the Fixed handle box in the Command Line to have the center point remain in the same position when you set a point for the primary handle.

Shortcut

Mouse: 

See Also: [Copying Objects in a Circle](#)

Tangent Two Lines command

Use the Circle Tangent Two Lines command to draw a circle tangent to two lines.


Radius

The radius of the circle is entered in Drawing Units in the Radius box in the Command Line.

Same as

The circle can be drawn using the same radius as a circle in the drawing by choosing the Same as button in the Command Line.

Shortcut


Mouse: 

See Also: [Drawing Tangent Circles](#)

Tangent Three Lines command

Use the Circle Tangent Three Lines command to draw a circle tangent to three lines. The radius of the circle is determined by the placement of the three lines.

Shortcut

Mouse: 

See Also: [Drawing Tangent Circles](#)

Close command

Use the Close command to close all windows containing the active drawing. When this command is selected you are asked if you want to save the current drawing. Select the Yes button to save the current drawing. Select the No button to close the current drawing without saving it. Select the Cancel button to return to the drawing screen without saving or closing the current drawing. If you close a drawing without saving, you lose all changes made since the last time you saved it.

You can also close a drawing by using the Close command in the [drawing Control-menu](#).

Shortcut

Keys: **Ctrl+F4**

See Also: [Closing a Drawing](#), [Closing a Drawing and Exiting DesignCAD](#)

Drawing Control-Menu

The drawing Control-menu is a menu with commands that control a drawing window, for example you can move, resize and close a drawing window.

Close All command

Use the Close All command to close all the open windows.

See Also: [Closing a Window](#)

Close command (Control menu)

Use the Close command to close the active window or dialog box.

Double-clicking a Control-menu box is the same as choosing the Close command.

Shortcuts

Keys: **Ctrl+F4** closes a drawing window
 Alt+F4 closes the DesignCAD window or a dialog box

See Also: [Closing a View](#)

Combine command

Use the Combine command to combine two or more lines into a single line. Each line that is to be combined must be connected to the next line. Therefore, each line selected for this command must share an endpoint with the previous line.

If the lines being combined are drawn in different colors or line types, the line resulting after they are combined will use the characteristics of the first line selected. If the lines are selected at the same time, the combined line will use the characteristics of the first line drawn.

See Also: [Combining Lines](#)

Apply Button

The Apply button lets you "apply" a selected option to an entity in the drawing.

For example, to have an object drawn in a different color, first select the object. Then select the new color in the color table. Next, select the Apply button.

Color Table

The color table is the list of colors displayed when the Show Color Table command is selected.

These colors can be edited with the Set Color command, or by double-clicking a color in the color table. The new colors can be saved with the Save Color Table command. A new color table can be loaded with the Load Color Table command.

Static Zoom

To "Static Zoom" a drawing means to permanently change the zoom of the drawing. When the Static Zoom button is selected in the Command Line, the actual size and position of the drawing changes, rather than just the view of the drawing.

Show Hot Toolbox option

Use the Show Hot Toolbox option to create a [Hot Toolbox](#) containing only those commands that you use most often.

Commands are selected in the Main or Snap Toolboxes and "dragged" to the Hot Toolbox.

Shortcut

Keys: **Shift+Alt+T**

See Also: [Creating Hot Toolbox](#)

Main Toolbox

The Toolbox can be displayed on the screen to give quick mouse access to the most-commonly used DesignCAD commands.

To hide or display the Toolbox, choose the [Show Main Toolbox option](#) in the View menu.

See Also: [Showing Main Toolbox](#), [Hiding Main Toolbox](#), [Changing Toolbox Columns](#), [Customizing DesignCAD](#)

Crosshair option

Use the Crosshair option to activate the [cursor crosshair](#). You can use the cursor crosshairs to line up points and entities on the screen.

Shortcut

Keys: **Ctrl+H**

See Also: [Displaying the Cursor Crosshairs](#)

Cursor Crosshairs

The crosshairs are horizontal and vertical lines attached to the pointer on the screen.


Curve command

Use the Curve command to draw a curve through three or more points. Up to 200 points can be set for the curve.

Save as vector form

The curve can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Shortcuts

Mouse: 

Keys: **C**

See Also: [Drawing Curves](#), [Converting Curves](#)

Curve To Line command

Use the Curve To Line command to change a curve entity to a line entity

See Also: [Converting Lines](#)


Cutoff command

Use the Cutoff command to "cut off" a section of a drawing. Then use other DesignCAD commands to erase, copy, move, or edit the section.

Cut selected entities only

The Cut selected entities only box in the Command Line allows you to select the entities you want to cut. Entities within the selection box but not selected will not be cut.

Shortcut

Mouse: 

See Also: [Breaking Lines](#)

Erase Last command

Use the Erase Last command to delete the last line or entity drawn. This command can be repeated until each entity is deleted.

Shortcut

Keys: **Ctrl+E**

See Also: [Deleting Entities](#)

Dimension command

Use the Dimension command to draw dimensioning information about an object. Dimensions can be drawn vertically, horizontally, or aligned to any angle.

HOR

Choose this button to have the dimension measured horizontally.

VER

Choose this button to have the dimension measured vertically.

ALO

Choose this button to have the dimension aligned to any angle.

The following dimension options can be changed with the dimension commands:

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size

Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcuts

Mouse:



Keys: @

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Dimensioning Lines](#)

Dimension Line button



Use the Dimension Line button to set the position, dimension offset, arrowhead type, and arrowhead size of the dimension.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the dialog box.

See Also: Position of Dimension Arrows, Dimension Arrowhead Type, Dimension Arrowhead Size

Dimension Extension Line button



Use the Dimension Extension Line button to set the overshoot length, fixed length, or variable length of the extension line for the dimension.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the dialog box.

See Also: Dimension Options, Changing Dimension Options, Extension Line

Dimension Text button



Use the Dimension Text button to set the format, precision, location, orientation, and size of the text information for the dimension.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the dialog box.

See Also: [Dimension Options](#), [Changing Dimension Options](#), [Dimension Text](#)

Dimension Prefix and Suffix button



Use the Dimension Prefix and Suffix button to set the prefix and suffix for the dimension text.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the dialog box.

See Also: [Dimension Options](#), [Changing Dimension Options](#), [Dimension Prefix](#), [Dimension Suffix](#)

Dimension Tolerance button



Use the Dimension Tolerance button to set the positive and negative tolerance information for the drawing as well as the size of the tolerance.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the dialog box.

See Also: [Dimension Options](#), [Changing Dimension Options](#), [Tolerancing Information](#)

Dimension Angle command

Use the Dimension Angle command to draw dimensioning information for an angle. Points are set for the center, beginning, and end of the angle to be measured. A fourth point is set for the location of the dimension text information.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size

Save as vector form

Selecting the Save as vector form box in the Command Line saves the dimension as a vector entity.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: Adding Dimensions, Changing Dimensions, Drawing Angle Dimensions

Dimension Arc command

Use the Dimension Arc command to dimension the length of an arc. Points are set on the arc and at the location for the dimension text information.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size


Save as vector form

Selecting the Save as vector form box in the Command Line saves the dimension as a vector entity.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Dimensioning Arcs](#)

Dimension Coordinate command

Use the Dimension Coordinate command to draw coordinate dimensions relative to a base point. The first point is the drawing origin, or reference point, from which subsequent points are dimensioned.

If the second point is set above or below the first point, the X-value, or horizontal distance, from the reference point will be placed in the dimension. If the second point is set to the right or left of the first point, the Y-value, or vertical distance, from the reference point will be placed into the dimension.

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix


Save as vector form

Selecting the Save as vector form box in the Command Line saves the dimension as a vector entity.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: Adding Dimensions, Changing Dimensions, Dimensioning Lines

Drawing Origin

The origin of a drawing is the location of the point with the coordinates 0, 0.

The origin lies in the lower left corner of the DesignCAD screen before any zooms, pans, or rotates are performed.

When the Dimension Coordinate command is selected, a point is set for the origin. This will be the point that the subsequent points are measured from.

When the Origin command is selected, a point is set for the new origin of the drawing. This point is used as the origin until it is changed or another drawing is retrieved.

Reference Point

A reference point is a point in the drawing used to define a unit of measurement or a coordinate position.

When the Dimension Coordinate command is selected a point is set for the reference point so that subsequent points can be measured from this location.

Dimension Diameter command

Use the Dimension Diameter command to draw diameter dimensions for a circle or arc.

Pre-Defined Extension

Choose this button to have the dimension drawn with a pre-defined extension line.

Dimension Outside Circle

Choose this button to have the dimension text and arrowheads drawn outside the circle.

Dimension Inside Circle

Choose this button to have the dimension arrowheads drawn inside the circle.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size

Save as vector form

Selecting the Save as vector form box in the Command Line saves the dimension as a vector entity.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Dimensioning Circles](#)

Dimension Radius command

Use the Dimension Radius command to draw radius dimensions for a circle or arc.

Pre-Defined Extension

Choose this button to have the dimension drawn with a pre-defined extension line.

Dimension Outside Circle

Choose this button to have the dimension text and arrowheads drawn outside the circle.

Dimension Inside Circle

Choose this button to have the dimension arrowheads drawn inside the circle.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size

Save as vector form

Selecting the Save as vector form box in the Command Line saves the dimension as a vector entity.

Same as

The dimension can be drawn with the same options as another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse:



See Also: Adding Dimensions, Changing Dimensions, Dimensioning Circles

Dimension Distance Only command

Use the Dimension Distance Only command to dimension a line without extension lines and arrows.

HOR

Choose this button to have the dimension measured horizontally.

VER

Choose this button to have the dimension measured vertically.

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size


Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Dimensioning Lines](#)

Dimension Extended command

Use the Dimension Extended command to draw successive dimensions extended along several points.

HOR

Choose this button to have the dimension measured horizontally.

VER

Choose this button to have the dimension measured vertically.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size


Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Drawing Extended Dimensions](#)

Dimension Baseline command

Use the Dimension Baseline command to draw baseline dimensions.

HOR

Choose this button to have the dimension measured horizontally.

VER

Choose this button to have the dimension measured vertically.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size


Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Dimensioning Lines](#)

Dimension Progressive command

Use the Dimension Progressive command to draw the dimensions of a line progressively from a base point.

HOR

Choose this button to have the dimension measured horizontally.

VER

Choose this button to have the dimension measured vertically.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size


Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Drawing Progressive Dimensions](#)

Dimension Progressive Radius command

Use the Dimension Progressive Radius command to draw radius dimensions progressively.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Extension Line

Overshoot Length

Fixed Length

Variable Length

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size

Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: Adding Dimensions, Changing Dimensions, Drawing Progressive Dimensions

Dimension Chamfer command

Use the Dimension Chamfer command to draw the dimensions for a chamfered line.

Dimension Line

Position of Dimension Arrows

Dimension Arrowhead Type

Dimension Arrowhead Size

Dimension Text

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix and Suffix

Dimension Prefix

Dimension Suffix

Dimension Tolerance

Positive and Negative Dimension Tolerance

Positive or Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size


Save as vector form

The dimension can be saved as a vector entity by choosing the Save as vector form box in the Command Line.

Same as

The dimension can be drawn with the same options of another dimension entity in the drawing by selecting the Same as button in the Command Line.

Shortcut

Mouse: 

See Also: [Adding Dimensions](#), [Changing Dimensions](#), [Dimensioning Lines](#)

Display Grid

The display grid is a series of "guide lines" drawn on the screen. The display grid helps locate points in a drawing. The grid is not printed when the drawing is printed.

Set Color command

Use the Set Color command to change the color specifications of the colors in the [color table](#).

The changes made to the color table can be saved with the [Save Color Table command](#). The new color table can be loaded with the [Load Color Table command](#).

Dialog box options

Red Set the new color for the red value in this box.

Green Set the new color for the green value in this box.

Blue Set the new color for the blue value in this box.

Current The original color is shown in this box.

New The new color is shown in this box.

Shortcut

Mouse: Double-click the color in the color table.

See Also: [Customizing the Color Table](#), [Save Color Table command](#)


Ellipse command

Use the Ellipse command to draw an ellipse, or oval using three points.

Save as vector form

Selecting the [Save as vector form box](#) in the Command Line saves the ellipse as a [vector entity](#).

Shortcut

Mouse: 

See Also: [Drawing Elliptical Circles](#)


Elliptical Arc command

Use the Elliptical Arc command to draw an elliptical arc, or part of an ellipse or oval, between two points.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the arc as a vector entity.

Shortcut

Mouse: 

See Also: [Drawing Elliptical Arcs](#)

Display Grid option

Use the Display Grid option to activate the [display_grid](#).

The display grid type and size are set with the [Grid Options command](#).

Shortcut

Keys: **G**

See Also: [Setting the Display Grid](#)

Snap Grid option

Use the Snap Grid option to activate the [snap_grid](#).

The snap grid size is set with the [Grid Options command](#).

Shortcut

Keys: **Ctrl+G**

See Also: [Setting Grids](#), [Setting the Grid Sizes](#)

Entity Select command

Use the Entity Select command to select all entities of a certain type in the drawing.

The following entities can be selected with the Entity Select command:

<u>Vector</u>	<u>Symbol</u>
<u>Ellipse</u>	<u>Point mark</u>
<u>Text</u>	<u>Angular Dimension</u>
<u>Curve</u>	<u>Linear Dimension</u>
<u>Back Arc</u>	<u>Diameter/Radius Dimension</u>
<u>Ellipse Arc</u>	<u>Arc Dimension</u>
<u>Bezier Curve</u>	<u>Radius Progress Dimension</u>
<u>Attribute</u>	<u>Linear Progress Dimension</u>
<u>Circle or Arc</u>	<u>Chamfer Dimension</u>
<u>Hatch</u>	<u>Coordinate Dimension</u>
<u>Text Arc</u>	<u>Bitmap</u>
<u>Arrow</u>	

See Also: [Selecting Entities](#), [Selecting Certain Entities](#)

Erase command

Use the Erase command to selectively erase lines or entities in the drawing. Set a point on each line or entity to be erased.

More than one entity can be erased at a time with this command by setting points on several entities or setting points in opposite corners of a rectangular selection box. All entities selected within this box will be erased.

Shortcut

Keys: **E**

See Also: [Erasing Entities](#)

Exit command

Use the Exit command to end your DesignCAD session. You can also use the Close command on the application Control-menu to exit DesignCAD. DesignCAD prompts you to save any open drawing with unsaved changes.

Shortcuts

Mouse: Double-click the application Control-menu button.

Keys: **Alt+F4**

See Also: [Exiting DesignCAD](#)

Export DXF command

Use the Export DXF command to export a drawing in a DXF drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Extending Lines](#)

Extend command

Use the Extend command to extend or shorten a line by moving one of the points defining the end of the line. The line maintains the same angle after it has been extended.

See Also: [Extending Lines](#), [Editing Lines](#)

Fillet command

Use the Fillet command to "round off" the corner of a line or arc entity with a smooth arc.

Fillet Radius

The "fillet radius" is entered in Drawing Units in the Fillet radius box in the Command Line.


Keep original lines

The original lines used to make up the corner of the entity being filleted are removed unless the Keep original lines box is selected in the Command Line.

Same as

The line can be filleted with the same radius as another line in the drawing by selecting the Same as button in the Command Line.

Shortcuts

Mouse: 

Keys: **F**

See Also: [Filleting Lines](#)

Fill Wide Lines option

Use the Fill Wide Lines option to fill all lines in the drawing with a width greater than 0.00 Drawing Units.

See Also: Filling Wide Lines, Changing Line Characteristics

Fit to Window command

Use the Fit to Window command to zoom a drawing so that it fills the entire drawing window.

Shortcut

Keys: **Ctrl+W**

See Also: Zooming a Drawings, Restoring a Drawing to Former Size

Character Font

The character font is the typeface that the text is drawn in.

Load Color Table command

Use the Load Color Table command to load a different color table onto the screen.

The color table is displayed when the Show Color Table option is selected. The color table can be changed with the Set Color command. The new color table can be saved with the Save Color Table command.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type


DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: Getting Color Table, Loading Color Table

Gravity command

Use the Gravity command to snap to the nearest point in a drawing and set a point.

Shortcuts

Mouse: 


Keys: .

See Also: [Choosing Point Commands](#), [Setting Points](#), [Gravity Move command](#)

Gravity Move command

Use the Gravity Move command to "snap" the pointer to the nearest point without setting a point.

Shortcuts

Mouse: 

Keys: ,

See Also: [Choosing Point Commands](#), [Setting Points](#), [Gravity command](#)

Grid Options command

Use the Grid Options command to set the display grid size and type, and the snap grid size.

Dialog box options

Display Grid Type Choose the display grid type from this box.

Display Grid Size Enter the display grid size in this box.

Snap Grid Size Enter the snap grid size in this box.

Enable display grid This box enables or disables the display grid.

Enable snap grid This box enables or disables the snap grid.

Save Options Choose this box to save the grid options to the DCW.INI file.

See Also: [Setting Grids](#), [Setting the Grid Sizes](#)

Group Define command

The Group Define command defines one or more entities as a single object, or Group, so that the entities can be selected and edited as a single object.

See Also: [Defining a Group](#)

Group Explode command

The Group Explode command "explodes," or "undefines," a Group so that the individual entities can be selected and edited separately.

See Also: [Undefining a Group](#)

Hatch command

Use the Hatch command to add hatch patterns to an area.

Hatch Angle

The angle of the hatch pattern is entered in the Hatch Angle box in the Command Line.

Hatch Scale

The size of the pattern is entered in the Hatch Scale box in the Command Line.

Hatch Pattern

The hatch pattern is selected by choosing the Hatch Pattern button in the Command Line.

Same as

The hatch can be drawn using the same angle, scale, and pattern of other hatch areas in the drawing by choosing the Same as button in the Command Line. Then choose the hatch area in the drawing.

See Also: [Hatching a Selected Area](#), [Hatch Patterns](#)

Hatch Line command

Use the Hatch Line command to add hatch patterns to an area enclosed by a line.

Hatch Angle

The angle of the hatch pattern is entered in the Hatch Angle box in the Command Line.

Hatch Scale

The size of the pattern is entered in the Hatch Scale box in the Command Line.

Hatch Pattern

The hatch pattern is selected by choosing the Hatch Pattern button in the Command Line.

Same as

The hatch can be drawn using the same angle, scale, and pattern of other hatch areas in the drawing by choosing the Same as button in the Command Line. Then choose the hatch area in the drawing.

See Also: [Hatching a Line Entity](#), [Hatch Patterns](#)

Hatch Fill command

Use the Hatch Fill command to add hatch patterns inside an area.

Hatch Angle

The angle of the hatch pattern is entered in the Hatch Angle box in the Command Line.

Hatch Scale

The size of the pattern is entered in the Hatch Scale box in the Command Line.

Hatch Pattern

The hatch pattern is selected by choosing the Hatch Pattern button in the Command Line.

Selection Only

If the entity is selected before the command is chosen, you can choose the Selection Only box to limit the hatch boundary to only those lines that are selected.

Same as

The hatch can be drawn using the same angle, scale, and pattern of other hatch areas in the drawing by choosing the Same as button in the Command Line. Then choose the hatch area in the drawing.

See Also: [Hatching an Enclosed Area](#), [Hatch Patterns](#)

Help Index command

Use the Help Index command to display help for DesignCAD commands.

To use the Help Index

Click an underlined topic.

-or-

Press the **Tab** key until you highlight the topic you want, then press **Enter**.

To scroll in the Help window

Click the scroll bars on the right side of the screen.

-or-

Press the up or down arrow keys.

To return to the previous topic

Click the Back button.

-or-

Press **B** to return to the previous topic.

To open the Search feature in Help

Click the Search button.

-or-

Press **S** to select the Search feature.

To close the Help window

Choose Exit from the Help File menu.

-or-

Press **Alt+F** to open the Help File menu.

Press **x** to choose the Exit command.

Shortcuts

Keys: **F1**

-or-

Shift+F1

See Also: [Finding Information in Help](#), [Help Instructions](#)

Options command

Use the Options command to change default options for DesignCAD.

This command allows you to change:

[Toolbox](#)

[Symbol Toolbox](#)

[Macro](#)

[Coordinate Bar](#)

[View](#)

[Drawing](#)

[Text](#)

[Cursor](#)

[Grid](#)

[General Dimension](#)

[Dimension Text](#)

[Dimension Prefix and Suffix](#)

[Digitizer Menu](#)

[Miscellaneous](#)

See Also: [Changing Default Options](#)

Import DXF command

Use the Import DXF command to open a drawing in DXF drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type


DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Exporting Drawings](#), [Converting Files](#)

Intersect-1 command

Use the Intersect-1 command to snap to the intersection of two lines.

Shortcuts

Mouse: 


Keys: **N**

See Also: [Choosing Point Commands](#), [Setting Points](#), [Intersect-1 Move command](#)

Intersect-2 command

Use the Intersect-2 command to snap to the intersection of two lines by setting two points.

Shortcut


Mouse: 

See Also: [Choosing Point Commands](#), [Setting Points](#), [Intersect-2 Move command](#)

Intersect-1 Move command

Use the Intersect-1 Move command to "snap" the pointer to the intersection of a line without setting a point.

Shortcuts

Mouse: 


Keys: **Ctrl+N**

See Also: [Choosing Point Commands](#), [Setting Points](#), [Intersect-1 command](#)

Intersect-2 Move command

Use the Intersect-2 Move command to "snap" the pointer to the intersection of two lines without setting a point.

Shortcut

Mouse: 

See Also: [Choosing Point Commands](#), [Setting Points](#), [Intersect-2 command](#)

Join command

Use the Join command to connect the points of two or more lines in the drawing. Set two points in opposite corners of the area to be joined. The endpoints within the selection box will all be moved to one location.

See Also: [Joining Lines](#)

Layer Setup command

Use the Layer Setup command to set the current layer for the drawing, name a layer, and display the layer commands.

A DesignCAD drawing can have up to 256 layers or subdivisions.

Dialog Box Options

Current layer

When you create a new drawing, the entities are added to the current layer. The current layer is shown in the Coordinate Bar at the top of the screen. The current layer defaults to Layer 1 unless otherwise specified.

No.

You can set the number of the current layer in this box.

Name

You can set the name of the current layer in this box.

Multilayer

Choose the Multilayer box to manipulate entities in the layers.

Visible/Invisible

Each layer can be set to visible or invisible status. A visible, or active, layer is displayed on the screen. An invisible, or inactive, layer is not visible on the screen.

Editable/Uneditable

A visible layer can be edited or can be "uneditable." An invisible layer cannot be editable.

Layer Names

Layers can be named for easy identification.

Commands

Choose the Commands button to open any of these commands:

[Select Layer](#)

[Move Layer](#)

[Save Layer](#)

[Delete Layer](#)

[Set Layer Color](#)

[Set Layer Line Style](#)

[Separate by Color](#)

See Also: [Editing Layers](#), [Manipulating Layers](#), [Setting Layer Options](#)

Current Layer

The current layer is the layer currently being used. The current layer is displayed in the Layer box in the Coordinate Bar at the top of the screen.

Active Layer

Layers can be active or inactive. Active layers are visible on the screen.

Inactive Layer

Layers can be active or inactive. Inactive layers are not visible on the screen.

Set Layer Color command

Use the Set Layer Color command to set the color for an entire layer. All entities in the layer will be changed to the new color. Subsequent entities drawn in the layer will be drawn in the current color shown in the Toolbox.

See Also: [Manipulating Layers](#)

Delete Layer command

Use the Delete Layer command to delete all the objects in a layer. All the objects in the current layer will be deleted.

This command erases existing objects only. It does not prevent objects from being drawn in the current layer after the command is issued.

See Also: [Editing Layers](#), [Manipulating Layers](#)

Set Layer Line Style command

Use the Set Layer Line Style command to set the type, width, and scale of the lines in a layer. The lines in the layer will be drawn in the new line style. Subsequent lines drawn in the layer will be drawn using the current line style shown in the Toolbox.

Dialog box options

Line Type Choose the line type from this box.

Scale Enter the scale for the line in this box.

Width Enter the width for the line in this box.

See Also: [Manipulating Layers](#)

Move Layer command

Use the Move Layer command to move all the objects in a layer to a different layer. All the objects in the current layer will be moved to the new layer. The current layer will be empty.

This command changes existing objects only. It does not prevent objects from being drawn in the current layer after the command is issued.

See Also: [Manipulating Layers](#), [Selecting and Moving Layers](#)

Save Layer command

Use the Save Layer command to save an entire layer as a separate drawing.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Manipulating Layers](#)

Select Layer command

Use the Select Layer command to select all the objects in a layer. All the objects in the current layer will be selected.

See Also: [Manipulating Layers](#), [Selecting and Moving Layers](#)

Separate by Color command


Use the Separate by Color command to separate the objects of the current layer into separate layers according to color. All the objects drawn in Color 1 will be moved to Layer 1, objects in Color 2 will be moved to Layer 2, etc.

See Also: [Manipulating Layers](#), [Separating Colors into Layers](#)

Line command

Use the Line command to draw a line between points. Up to 200 points can be set with this command.

Shortcuts

Mouse: 

Keys: **V**

See Also: [Line Types](#), [Drawing Lines](#)

Angle, Distance Two Points command

Use the Angle, Distance-Two Points command to calculate the angle and distance between two points and displays that information in a dialog box.

Dialog box options

Angle precision

This box sets the number of decimal places for angular measurement:

Angle format

This box sets the format for angular measurement information:

Distance precision

This box sets the number of decimal places for linear measurement:

Distance format

This box sets the format for linear measurement information:

Shortcuts

Mouse:



Keys: <

See Also: [Calculating Angle of Two Points](#), [Adding Angle of Objects to Drawing](#)

Angle Between Two Lines command

Use the Angle Between Two Lines command to find the angle between two lines.

Dialog box options


Precision

The angular precision box sets the number of decimal places for angular measurement:

Format

The angular format box sets the format for angular measurement information:

Shortcut


Mouse: 

See Also: [Calculating Angle of Two Lines](#), [Displaying Angle of Two Lines](#)

Line Snap command

Use the Line Snap command to snap to the nearest line in the drawing and set a point.

Shortcuts

Mouse: 


Keys: **K**

See Also: [Choosing Point Commands](#), [Setting Points](#), [Line Snap Move command](#)

Line Snap Move command

Use the Line Snap Move command to "snap" the pointer to the nearest line without setting a point.

Shortcuts

Mouse: 
Keys: **Ctrl+K**

See Also: [Choosing Point Commands](#), [Setting Points](#), [Line Snap command](#)

Line To Curve command

Use the Line To Curve command to change a line entity to a curve entity

See Also: [Converting Lines](#)

Merge command

Use the Merge command to merge two or more drawings into one drawing. The second drawing will be scaled to the Drawing Units of the drawing on the screen.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type


DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Merging Drawings](#)

Midpoint command

Use the Midpoint command to snap to the midpoint of the nearest line.

Shortcuts

Mouse: 


Keys: **F2**

See Also: [Choosing Point Commands](#), [Setting Points](#), [Midpoint Move command](#)

Midpoint Move command

Use the Midpoint Move command to "snap" the pointer to the midpoint of the nearest line without setting a point.


Shortcuts

Mouse: 
Keys: **Ctrl+F2**

See Also: [Choosing Point Commands](#), [Setting Points](#), [Midpoint command](#)

Minimize command (Control menu)

Use the Minimize command to reduce the DesignCAD window to an icon.

Clicking the  in the upper-right corner of the window is the same as choosing the Minimize command.

See Also: [Minimizing Windows](#)

Maximize command (Control menu)

Use the Maximize command to enlarge the active window to fill the available space.

A drawing window expands to fill the DesignCAD window.
The DesignCAD window expands to fill the entire screen.

Clicking the  in the upper-right corner of the window is the same as choosing the Maximize command.

Shortcut

Mouse: Double-click title bar

See Also: [Maximizing Windows](#)

Move command (Control menu)

Use the Move command to display a four-headed arrow so you can move the active window or dialog box with the arrow keys.

Note: This command is not available if the window is maximized.

See Also: [Moving and Sizing Windows](#)

Next command (Control menu)

Use the Next command to switch to the next open drawing window. DesignCAD determines which window by the order in which you opened the windows.

Shortcut


Keys: **Ctrl+F6**

See Also: [Switching to a Different Drawing Window](#)

New command

Use the New command to create a new drawing in DesignCAD. You can open an existing drawing with the [Open command](#).

Shortcuts

Mouse: 

Keys: Y

See Also: [Creating a New Drawing](#), [New Drawings](#)

Open command

Use the Open command to open an existing file in a new window. You can have several drawings open at once, depending on the size of each drawing.

If you have a drawing on your screen that you no longer want to use, choose the Close command to close it.

You can create new drawings with the [New command](#).

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the [drive](#) in which you want to store the drawing.

Directories

Select the [directory](#) in which you want to store the drawing.


Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

Set Handles

The Set Handles box allows you to set handles of the drawing to determine how it will be loaded. If handles were set before the drawing was saved, up to two points can be set to designate the primary and secondary handles. If no handles were set when the drawing was saved, up to four points can be set for the lower left, lower right, upper left, and upper right corners of the drawing respectively.

Shortcuts

Mouse: 

Keys: **Ctrl+F12**

See Also: [Opening a Drawing](#), [Opening an Existing Drawing](#), [Opening Drawings](#)

Directory

A directory is a subdivision of a disk that works like a filing system to help you organize your drawings. For example, you can create a directory called SYMBOL to store all your symbols.

Drive

The drive is the mechanism in your computer that turns a disk to retrieve and store information. Personal computers often have one hard drive labeled C and two drives labeled A and B that read removable disks.

Origin command

Use the Origin command to move the drawing origin of the drawing. The drawing origin is the location of coordinate 0, 0. It lies in the lower left corner of the screen before any zooms, pans, rotates, etc., are performed.

The Zoom, Rotate, and Pan commands may change the location of the origin on the screen, but it remains at the same point on the drawing unless it is changed with the Origin command, Zoom Static command, or at the time the drawing is retrieved.

See Also: [Setting a New Origin](#)

Original Size command

Use the Original Size command to return the drawing to its original size after a zoom command.

Shortcut


Keys: **Ctrl+O**

See Also: [Zooming a Drawing](#)

Ortho Line command

Use the Ortho Line command to draw a straight line exactly vertically or horizontally. Up to 200 points can be set with this command.

Shortcut


Mouse: 

See Also: [Drawing Regular Lines](#)

Pan command

Use the Pan command to pan, or move, the drawing across the screen.

Shortcut


Mouse: 

See Also: [Zooming Entire Drawing](#), [Panning Drawings](#)

Parallel-1 command

Use the Parallel-1 command to draw a line parallel to any other line, curve, circle, arc, ellipse, or elliptical arc.

Shortcuts

Mouse: 

Keys: =

See Also: [Drawing Parallel Lines](#)


Parallel-2 command

Use the Parallel-2 command to draw a line parallel to any other line or curve.

Distance

The distance the parallel line is drawn from the original line is set in the Distance box in the Command Line.

Shortcut


Mouse: 

See Also: [Drawing Parallel Lines](#)

Perpendicular To Line command

Use the Perpendicular To Line command to draw a line perpendicular from a point in the drawing to an existing line.

Shortcut

Mouse: 

See Also: [Drawing Perpendicular Lines](#)


Perpendicular From Line command

Use the Perpendicular From Line command to draw a perpendicular line to any other line or curve.

Distance

The distance of the perpendicular line is entered in the Distance box in the Command Line.

Shortcut

Mouse: 

See Also: [Drawing Perpendicular Lines](#)

Point Mark command

Use the Point Mark command to mark a point in the drawing with a small cross, box, circle, or combination of these objects.

Cross

Choose this button to have the point mark drawn as a single cross.

Cross and Box

Choose this button to have the point mark drawn as a cross inside a box.

Cross and Circle

Choose this button to have the point mark drawn as a cross inside a circle.

Cross, Circle, and Box

Choose this button to have the point mark drawn as a cross inside a circle and a box.


Point Mark Size

Enter the size for the point mark in this box.

Same as

Choose this button to have the point mark drawn the same as a point mark in the drawing. Then select the point mark in the drawing.

Shortcut

Mouse: 

See Also: [Marking Points](#)

Point Move command

Use the Point Move command to move or delete a point in the drawing.

Shortcut

Keys: *

See Also: [Moving Points](#)

Polygon-1 command

Use the Polygon-1 command to draw a regular polygon by setting two points for the first side.


No. of sides

The number of sides of the polygon is entered in the No. of sides box in the Command Line.

Same as

Select the Same as button to have the polygon drawn with the same number of sides as another polygon in the drawing.

Shortcut

Mouse: 

See Also: [Drawing Simple Polygons](#), [Drawing a Polygon with Corner Points](#)

Polygon-2 command

Use the Polygon-2 command to draw a regular polygon by setting a point for the center and a point for the corner of the polygon.


No. of sides

The number of sides for the entity is entered in the No. of sides box in the Command Line.

Same as

Select the Same as button to have the polygon drawn with the same number of sides as another polygon in the drawing.

Shortcut

Mouse: 

See Also: [Drawing Simple Polygons](#), [Drawing a Polygon with Center and Corner](#)

Print command

Use the Print command to print DesignCAD drawings. To specify a printer and its connection, use the [Printer Setup command](#).

Dialog Box Options

Drawing Size

The drawing size is shown in the Drawing Size box at the top of the dialog box. Choose the units of measurement from the Units box.

Print Area

The print area displays the scale, height, and width of the drawing.

Center Drawing Choose the Center Drawing box to have the drawing centered on the paper.

Fit to Paper Choose the Fit to Paper box to have the drawing fill the entire paper.

Panel

The Panel box displays the number of panels the drawing will use.

Mark Choose the type of panel mark from the Mark box.

Mark Panel Number This box determines if panels will be numbered or not.

Number of Copies

You can enter the number of copies to be printed in this box.

Paper Size

The Paper Size box shows the size of the paper loaded in the printer.

Margin You can change the size of the top, bottom, left, and right paper margins.

Orientation Drawings are either printed in Portrait or LandScape orientation.

Print Selection

This box is available when an entity is selected before the Print command is issued. Choosing this box allows you to print only the selected entity.

Print to File

Choosing this box saves the drawing file to disk.


Preview

Choose this button to preview the entire drawing before you print it.

Printer Setup

Choose this button to setup the printer.

Shortcuts

Mouse: 

Keys: **Ctrl+Shift+F12**

See Also: [Printing](#), [Printing and Plotting Drawings](#)

D.P.I. -- Dots Per Inch

The dots per inch, or d.p.i., determine how many dots the printer uses to output per inch in the drawing.

Printer Setup command

Use Printer Setup command to select a printer.

Dialog Box Options

Printer

Select the printer you want to use; only installed printers appear. You install printers and configure ports through the Windows Control Panel. For information on installing printers, see your system documentation.

Use Printer Font Only

This command allows you to set the printer to use printer fonts only. This aids in printing text at lower resolutions.

Setup

Displays Setup dialog box for the selected printer. The dialog box that appears depends on the printer.

See Also: [Print Options dialog box](#), [Printer Setup](#)

Print Options Dialog Box

Use the Print Options dialog box to select printing options for your printer.

Dialog Box Options

Units

Choose units of measurement used in your drawing.

- Inches
- Meters
- Centimeters
- Millimeters

Print Area

Enter the measurements of the drawing to be printed.

- Scale
- Height
- Width

Center Drawing

Select this box to center the drawing on the paper.

Fit to Paper

Select this box to have the drawing fill the paper.

Panel

Choose the type of panel mark to be used to mark panels.

- None
- Corner
- Outline Box

Mark Panel Number

Select this box to mark the panels with a corresponding number.

Number of Copies

Enter the number of copies you want printed.

Paper Size

Enter the margins for the printing area of your paper.

- Top
- Bottom
- Left
- Right

Orientation

Choose the Page Orientation for your drawing.

- Portrait
- LandScape

Print Selection

Select this box to print only the objects selected.

Print to File

Select this box to send the drawing to a disk rather than a printer.

Preview

Select this option to preview your drawing before it is printed.

Printer Setup

Select this option to display the Printer Setup command.

See Also: [Printing](#), [Printer Setup](#)

Font Cartridges

If your printer uses a font cartridge, it is important to select it; otherwise you won't have access in DesignCAD to all of the fonts available on your printer.

Dithering

Dithering determines how DesignCAD prints a drawing.

Point Relative command

Use the Point Relative command to set a point using relative X and Y coordinates.

Dialog box options

DX Enter the Delta X coordinate in this box.

DY Enter the Delta Y coordinate in this box.

Move cursor without setting point Choose this box to move the cursor without setting a point.

Shortcut

Keys:

See Also: [Choosing Point Commands](#), [Points and Command Selection](#)

Point XY command

Use the Point XY command to set a point using X and Y coordinates.

Dialog box options

X Enter the X coordinate in this box.

Y Enter the Y coordinate in this box.

Move cursor without setting point Choose this box to move the cursor without setting a point.

Shortcut

Keys: :

See Also: [Choosing Point Commands](#), [Points and Command Selection](#)

Point Polar command

Use the Point Polar command to set a point using polar coordinates.

Dialog box options

Distance Enter the distance from the last point in this box.

Angle Enter the angle from the last point in this box.

Move cursor without setting point Choose this box to move the cursor without setting a point.

Shortcut

Keys: ;

See Also: [Choosing Point Commands](#), [Points and Command Selection](#)

Pullout command

Use the Pullout command to add information to a drawing with a pullout line.

Text

The text is entered in the Text box in the Command Line.

Dimension Line button

The arrowhead type and size can be set by choosing the Dimension Line button in the Command Line.


Dimension Text button

The text location, orientation, and size can be set by choosing the Dimension Text button in the Command Line.

Same as

Choose the Same as button to have the balloon drawn with the same arrowhead and text size as another balloon in the drawing.

Shortcut

Mouse: 

See Also: [Pulling Out Information in a Drawing](#)


Quarter Circle command

Use the Quarter Circle command to draw a quarter circle, or 90 degree arc, between two points.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the quarter circle as a vector entity.

Shortcuts

Mouse: 

Keys: (

See Also: [Drawing Part of a Circle](#)

Record Macro command

The Record Macro command creates a macro. After selecting the command, type up to eight characters for the macro name. Every action you perform becomes part of the macro.

The Record Options command can be selected to save certain options with the macro.

When a macro is being recorded, a small dot appears on the Status Bar at the bottom of the screen.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters.

DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

Note: Macros must be stored in the current directory that DesignCAD was installed in to appear in the Macro Options dialog box.

See Also: [Using Macros](#), [Creating Macros](#), [Recording Macro Options](#)

Execute Macro command

The Execute Macro command executes a macro. The macro name is chosen from the File Name box. A point is set for the "startup" cursor position.

The macro will be performed starting at the point set for the "startup" cursor position.

When a macro is being executed, a small triangle appears on the Status Bar at the bottom of the screen.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters.

DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Using Macros](#), [Running a Macro](#)

Stop Recording Macro command

Choose the Stop Recording Macro command after performing the steps to record a macro.

A macro is recorded with the [Record Macro command](#).

Shortcut

Keys: **&**

See Also: [Using Macros](#)

Record Options command

The Record Options command is active while the macro is being executed. This command allows you to save and change options within the macro.

Use this command to record options such as layer, color, line style and number of points set while recording a macro.

Dialog box options

Record Layer Information

Choose this box to record layer information such as layer color, layer name, and current layer with the macro.

Record Color Information

Choose this box to record the current color of the drawing with the macro.

Record Line Style Information

Choose this box to record the current line style and width of the lines drawn within the macro.

Record Input Information

Choose this box to record most information entered in the Command Line or the Text Block dialog box. If the Record Input Information box is disabled, most information entered in the Command Line or Text Block dialog box will not be recorded with the macro.

If both the Record Input Information and the Record Points Set boxes are disabled, neither input or point information will be recorded. When the macro is executed, DesignCAD will execute the commands, stopping when it reaches the input for the points. The macro will pause until points are set or until **Enter** is pressed. This allows interactive macros.

If the Record Points Set box is enabled and the Record Input Information box is disabled, the point information will be recorded and the input information will not. When the macro is executed, DesignCAD will execute the commands, set the points, and wait for further information to be entered or **Enter** to be pressed.

Record Command Options

Choose this box to record all drawing options including color, line type, dimension options, and layer options. However, some information entered in the Command Line or in the Text Block dialog box is not recorded with the Macro Record Options command.

Record Points Set

Choose this box to record any points set within the macro.

See Also: [Using Macros](#), [Recording Macro Options](#)

Redo and Undo commands

Use the Redo command to "undo" the previous Undo command, returning the drawing to its previous state. This command can only be selected after an Undo command.

Use the Undo command to "undo" the last command issued, such as a move, copy, or scale command.

Shortcut

Keys: **Ctrl+Z**

See Also: [Undoing and Redoing Commands](#)

Regenerate command

Use the Regenerate command to quickly refresh the screen.

The Regenerate command is a quick way to refresh the screen and regenerate any lines that look as if they have been partially erased.

Shortcut

Keys: **Ctrl+R**

See Also: [Repairing Lines](#)

Repeat command

The Repeat command repeats the last command completed.

Shortcut

Keys: **F3**

See Also: [Repeating Commands](#)

Array command

Use the Array command to duplicate selected objects a certain number of times in a straight line, at any angle and offset, or in a rectangular array.

No. of copies

The number of times the objects are to be copied is entered in the No. of copies box in the Command Line.

No. of rows

The number of rows the objects are to be copied in is entered in the No. of rows box in the Command Line.


Use original as 1st copy

Select the Use original as 1st copy box in the Command Line to have the first copy drawn in the same location as the selected object.

Fixed handle

Use the Fixed handle box in the Command Line to have the center point remain in the same position when you set a point for the primary handle.


Shortcut

Mouse: 

See Also: [Copying Objects in a Row](#)

Restore command (Control menu)

Use the Restore command to return the active window to its size and position before you chose the Maximize or Minimize command.

Clicking the  in the upper-right corner of a maximized window is the same as choosing the Restore command.

Shortcut

Mouse: Double-click the title bar

See Also: [Restoring a Drawing Window to Its Former Size](#)

Save command

Use the Save command to save the current drawing and its current name, location, and file format. When you save a drawing for the first time, DesignCAD displays the Save As dialog box so you can name your drawing. If you want to change the name, location, or format of an existing drawing before you save it, choose the Save As command.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

Shortcuts

Mouse:



Keys: **Shift+F12**

See Also: [Saving Drawings](#), [Saving a Drawing](#), [Saving a New, Unnamed Drawing](#)

Save As command

Use the Save As command to save the current drawing with a specified file format, or lock a file so that it can be read only.

Note: To save a drawing with its existing name, format, and location, use the [Save command](#).

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

Shortcut

Keys: **F12**

See Also: [Saving an Existing Drawing](#), [Saving Your Drawings](#)

Save Color Table command

Use the Save Color Table command to save changes made to the color table to disk.

The color table is displayed when the Show Color Table option is selected. The color table can be changed with the Set Color command. The new color table can be loaded with the Load Color Table command.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: Saving Color Table

Segment command

Use the Segment command to break a line, arc, circle, or curve entity into segments of equal length.

No. of sections


The number of segments the entity is broken into is entered in the No. of sections box in the Command Line.

See Also: [Segmenting Entities](#)

Select Mode command

Use the Select Mode to select entities within the drawing.

Shortcut

Mouse: 

See Also: [Selecting Objects](#)

Select All command

Use the Select All command to select all the entities in a drawing. You can then use other DesignCAD commands to edit the highlighted entities.

See Also: [Selecting Entities](#), [Selecting All Entities](#)

Select Previous command

Use the Select Previous command to select the last object selected.

This command is useful when selecting complicated entities.

See Also: [Selecting Entities](#), [Selecting Last Entity](#)

Load Selection command

Use the Load Selection command to load a drawing and select it. Up to four points can be set for the lower left, lower right, upper left, and upper right corners of the drawing. When the points are set, or **Enter** is pressed, the drawing is loaded and the entities are selected.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

Shortcut

Keys: **Ctrl+F9**

See Also: [Loading and Selecting Objects](#)

Select Ortho command

Use the Select Ortho command to force a line or lines to be vertical or horizontal. All line segments within 10 degrees of vertical or horizontal will be moved so that they lie exactly vertical or horizontal.

See Also: [Straightening Lines](#)

Select Rotate command

Use the Select Rotate command to rotate selected entities.

Rotate Angle

The angle the objects are to be rotated is entered in the Rotate Angle box in the Command Line.

Rotate Stepping Angle

The rate at which the rotate angle changes is entered in the Rotate Stepping Angle box in the Command Line.

See Also: [Rotating Objects](#)

Save Selection command

Use the Save Selection command to save selected objects to disk as a separate drawing.

The new drawing can be loaded later with the [Open](#) or [Load Symbol](#) commands.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Saving Selected Objects](#)

Select Scale command

Use the Select Scale command to scale objects along the X or Y axis. This command "stretches" objects to make them taller, shorter, or wider.

Scale X

The amount the object is to be scaled along the X axis is entered in the Scale X box in the Command Line.

Scale Y

The amount the object is to be scaled along the Y axis is entered in the Scale Y box in the Command Line.

See Also: [Scaling Objects with Select Scale](#), [Scaling Objects with Selection Box](#), [Scaling Objects with Constant Aspect Ratio](#)

Select Zoom command

Use the Select Zoom command to zoom, or scale, objects to be larger or smaller.

Zoom factor

The zoom factor is entered in the Zoom factor box in the Command Line.

This command does not make a new copy of the objects. The original objects are resized.

See Also: [Zooming Selected Objects](#)


Semi-Circle command

Use the Semi-Circle command to draw a semi-circle, or 180 degree arc, between two points.

Save as vector form

Selecting the Save as vector form box in the Command Line saves the semi-circle as a vector entity.

Shortcuts

Mouse: 

Keys:)

See Also: [Drawing Part of a Circle](#)

Show Attribute option

Use the Show Attribute option to display attributes drawn with the [Attribute command](#).

Shortcut

Keys: **Shift+Alt+A**

See Also: [Displaying Attributes](#)


Point Select Mode command

Use the Point Select Mode to display all the points making up an entity.

You can use the Point Select Mode to cut a line into two sections.

The Point Select Mode can also be used to bend a line, or add a vertex to a line entity.

Shortcut

Mouse: 

See Also: [Selecting Objects](#)

Size command (Control menu)

Use the Size command to display a four-headed arrow so you can size the active window with the arrow keys.

Note: This command is not available if the window is maximized.

See Also: [Sizing a Window](#)

Vertex

A vertex is a corner added to a line entity with the Point Select Mode.

Show Coordinate Bar option

Use the Show Coordinate Bar option to display the Coordinate Bar at the top of the screen.

Shortcut

Keys: **Ctrl+Alt+B**

See Also: Showing the Coordinate Bar

Coordinate Bar

The Coordinate Bar displays the current X and Y coordinates of the cursor, the Delta X and Delta Y coordinates of the cursor, and the current layer of the drawing.

The Show Polar Coordinates option changes the Coordinate Bar to display polar coordinates, or the angle and distance from the last point.

See Also: Showing Coordinate Bar, Hiding Coordinate Bar

Delta X Coordinate

The Delta X coordinate is the horizontal distance from the cursor to the last point set. The Delta X coordinate is shown in the Coordinate Bar at the top of the screen.

Delta Y Coordinate

The Delta Y coordinate is the vertical distance from the cursor to the last point set. The Delta Y coordinate is shown in the Coordinate Bar at the top of the screen.

Show Info Box option

Use the Show Info Box option to display an Info Box on the screen. The Info Box displays information on a selected entity such as the entity type, color, area, length, and number of points. The information shown in the Info Box depends on the type of entity selected.

Shortcut

Keys: **Ctrl+I**

See Also: [Showing Info Box](#)

Show Snap Toolbox option

Use the Show Snap Toolbox option to display the [Snap Toolbox](#) on the screen.

Shortcut

Keys: **Shift+Alt+N**

See Also: [Showing Snap Toolbox](#)

Snap Toolbox

The Snap Toolbox can be displayed on the screen to provide quick mouse access to the most-commonly used DesignCAD point commands.

To hide the Snap Toolbox, choose the [Show Snap Toolbox option](#) in the View menu.

See Also: [Showing the Snap Toolbox](#), [Hiding Snap Toolbox](#), [Changing Toolbox columns](#), [Customizing DesignCAD](#)

Show Status Bar option

Use the Show Status Bar option to display the [Status Bar](#) at the bottom of the screen.

Shortcut

Keys: **Shift+Alt+B**

See Also: [Showing the Status Bar](#)

Status Bar

The Status Bar at the bottom of the screen displays command information, the current zoom factor of the drawing, and the number of points set. When a macro is being recorded, a small dot appears in the Status Bar. When a macro is being executed, a small triangle appears in the Status Bar.

See Also: [Showing Status Bar](#), [Hiding Status Bar](#)

Show Polar Coordinates option

Use the Show Polar Coordinates option to display the polar coordinates of points in the drawing in the Coordinate Bar.

The Options command allows you to change the format and precision of the polar coordinates displayed in the Coordinate Bar.

Shortcut

Keys: **J**

See Also: [Changing Coordinate Bar Information](#)

Polar Coordinates

Polar coordinates are the angle and distance of a point from another point.

Show Main Toolbox option

Use the Show Main Toolbox option to display the [Toolbox](#) on the screen.

Shortcut


Keys: **Shift+Alt+M**

See Also: [Showing Main Toolbox](#)

Sketch command

Use the Sketch command to draw a freehand curve following the movement of the cursor.

Shortcut

Mouse: 

See Also: [Cleaning Up a Line](#)

Smooth command

Use the Smooth command to smooth a line or curve in the drawing. If the entity selected with this command is a line, it will be converted to a curve. If the entity selected is a curve, it will be converted to a smoother curve.

See Also: [Smoothing Lines](#)

Stretch command

Use the Stretch command to stretch objects. The area to be stretched is defined by two points set in opposite corners. Then a point is set for the original location of the point. A second point is set for the new position of this point. The objects contained in the area are stretched so that the first point is moved to the new location.

See Also: [Stretching Objects](#), [Stretching a Line](#)

Switch To command (DesignCAD Control menu)

Use the Switch To command to display a list of all open applications.

Shortcut

Keys: **Ctrl+Esc**

See Also: [Switching to a Different Drawing Window](#)

Symbol Explode command

Use the Symbol Explode command to "undefine" a symbol so that the individual entities can be manipulated separately.

See Also: [Manipulating Symbols](#)

Load Symbol command

Use the Load Symbol command to load a DesignCAD symbol into the drawing. When a symbol is loaded, an "insertion entity" is added to the drawing. The symbol file (.DW2 file) must be present when the symbol is loaded. If the symbol file is modified, the change will be reflected in all the drawing using that symbol.

Points are set for the lower left and lower right corners of the symbol.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters.

DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type


DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Loading Symbols](#), [Installing Symbols](#)

Tangent To Circle command

Use the Tangent To Circle command to draw a line tangent from a point to a circle, ellipse, arc, or elliptical arc.

Shortcut


Mouse: 

See Also: [Drawing Tangent Lines](#)

Tangent From Circle command

Use the Tangent From Circle command to draw a line tangent to two circles.

Shortcut


Mouse: 

See Also: [Drawing Tangent Lines](#)

Tangent Between Circles command

Use the Tangent Between Circles command to draw a line tangent to a circle, ellipse, arc, or elliptical arc from a point on the entity.

Shortcut

Mouse: 

See Also: [Drawing Tangent Lines](#)

Text command

Use the Text command to draw a string of text.

Text

The text is entered in the Text box in the Command Line. Up to 80 characters can be entered.

Size

The size of the text is entered in the Size box in the Command Line.

Font

Use the Font box in the Command Line to change the font.

Angle

The angle of the text is entered in the Angle box in the Command Line. The angle of the text is also determined by the location of the points set with the Text command.

Orientation

Text can be left-justified, centered, or right-justified by selecting the orientation in the Orientation box in the Command Line.

Bold

Choose this button to have the text drawn in bold.

Italic

Choose this button to have the text drawn in italic.

Text Block

Several lines of text can be entered by choosing the Text Block button in the Command Line. The text is entered in the text window. Pressing **Enter** starts a new line. The space between the lines is set in the Line Spacing box.

Same as

Text can be drawn with the same options as other text in the drawing by selecting the Same as button in the Command Line.

After all the options are set, points are set for the lower-left and lower-right corners of the text string.

Shortcuts

Mouse: 

Keys: T

See Also: [Drawing Text](#), [Text Options](#), [Aligning Text](#)

Tick Marks command

Use the Tick Marks command to draw short lines, or tick marks, along a line, curve, ellipse, or circle entity. These tick marks are spaced at equal intervals along the entity.

Sections

The number of tick marks drawn is entered in the No. of tick marks box in the Command Line.

Divisions

The number of divisions per tick mark is entered in the No. of divisions per tick mark box in the Command Line.


Large tick mark

The size of the large tick marks is entered in Drawing Units in the Large Tick mark box in the Command Line.

Small tick mark

The size of the small tick marks is entered in Drawing Units in the Small Tick mark box in the Command Line.

Shortcut

Mouse: 

See Also: [Drawing Short Line Segments](#)

Tile Horizontal command

Use the Tile Horizontal command to tile all open windows horizontally on the screen.

See Also: [Tiling Windows](#)

Tile Vertical command

Use the Tile Vertical command to tile all open windows vertically on the screen.

See Also: [Tiling Windows](#)


Trim One Line command

Use the Trim One Line command to "trim" one line or arc so that its endpoint is precisely at its intersection with another line.

Trim shorter end

Choose the Trim shorter end box in the Command Line to have the shorter end erased when the line is trimmed.

Shortcut

Mouse: 

See Also: [Trimming Lines](#)


Trim Two Lines command

Use the Trim Two Lines command to "trim" two non-parallel lines or arcs so they meet precisely at a corner. Each line is maintained at its original angle and is extended or shortened to meet the other line.

Trim shorter end

Choose the Trim shorter end box in the Command Line to have the shorter end erased when the line is trimmed.

Shortcuts

Mouse: 


Keys: **B**

See Also: [Trimming Lines](#)

Trim Between Lines command

Use the Trim Between Lines command to "trim" a line between its intersection with two other lines.

Shortcut

Mouse: 

See Also: [Trimming Lines](#)

Trim Double Lines command

Use the Trim Double Lines command to "trim" two sets of parallel lines at their intersection.


Select trim area

Choose the Select trim area box in the Command Line to trim the lines within a specified area.

Select trim lines and area

Choose the Select trim lines and area box in the Command Line to trim certain lines within an area.

Shortcuts

Mouse: 

Keys: **Ctrl+B**

See Also: [Trimming Lines](#)

Undo and Redo commands

Use the Undo command to "undo" the last command issued, such as a move, copy, or scale command.

Use the Redo command to "undo" the previous Undo command, returning the drawing to its previous state. This command can only be selected after an Undo command.

Shortcut

Keys: **Ctrl+Z**

See Also: [Undoing and Redoing Commands](#)

Unerase command

Use the Unerase command to "unerase" the previous Erase or Erase Last command. This command can be repeated to "unerase" each erased entity, as long as no other command is selected.

Shortcut

Keys: !

See Also: [Unerasing Commands](#), ["Unerasing" Entities](#)

Units command

Use the Units command to set the Drawing Units of a specified distance in the drawing.

Set two points for the distance to be measured.

Original

The original distance between these two points will be displayed in the Original box.

New

Type the distance these points represent in the New distance box.

Shortcut

Keys: **U**

See Also: [Measuring the Units](#)

Drawing Units

Drawing Units are the units of measurement for a drawing. These units can represent inches, millimeters, feet, etc.

New Window command

Use the New Window command to open a new view of the drawing.

DesignCAD allows several different windows, or views, to be opened simultaneously. These views can be "zoomed" or "panned" independently.

The current view information can be saved to a file using the View Save command. The View Read command can be used to read a view file from disk.

See Also: Opening a New Window, Closing a Window

Zoom command

Use the Zoom command to temporarily change the size of the current drawing based on a "zoom factor." Points can be set for the "zoom origin" and "zoom destination."

Zoom factor

The zoom factor for the drawing is entered in the Zoom factor box in the Command Line.

Static Zoom

The Static Zoom button can be selected to permanently change the zoom of the drawing.

Shortcut

Keys: **Z**

See Also: Zooming Entire Drawing, Zooming a Drawing

Static Zoom button

Use the Static Zoom button to permanently change the zoom of the drawing. When Static Zoom is selected, the actual size and position of the drawing changes, rather than just the view of the drawing.

Zoom Factor

The "zoom factor" is the relative size the drawing is to be increased or decreased.

Zoom Origin

The zoom origin is the point into which the drawing is to be zoomed.

Zoom Destination


The zoom destination is the location to which the first point, or zoom origin, will be moved when the drawing is zoomed.

Zoom In command

Use the Zoom In command to zoom into a drawing based on one point and a zoom factor.

The zoom factor is based on the percentage defined in [View Options](#) of the Options command.

Shortcuts

Mouse: 

Keys: +

See Also: [Zooming a Drawing](#)

Zoom Window


The zoom window determines the area to be zoomed to fill the screen. Two points are set in opposite corners of the area to be zoomed.

Zoom Out command

Use the Zoom Out command to zoom out of a drawing based on one point and a zoom factor.

The zoom factor is based on the percentage defined in the [Options command](#).

Shortcuts

Mouse: 

Keys: -

See Also: [Zooming a Drawing](#)

Zoom Previous command

Use the Zoom Previous command to return the drawing to the previous configuration before the last zoom.

Shortcut

Keys: **Ctrl+P**

See Also: [Zooming a Drawing](#)

Group

A Group is several entities defined as a single object.

Fillet Radius

The fillet radius is the radius of the arc used to "fillet" two lines with the Fillet command.

Active Drawing Window

The active drawing window is the window, or view, of the drawing containing the cursor. DesignCAD commands such as pan and zoom will affect only this window.

Arrow Keys

"Arrow keys" is the collective name for the up arrow, down arrow, left arrow, and right arrow keys. These keys can be used to move the cursor on the drawing screen, select commands from the Command Menu, or select information boxes in dialog boxes.

Click

Clicking the mouse button means to point to an item on your screen and then quickly press and release the mouse button. You choose entities on the screen and move around in a drawing by clicking. Double-clicking, pointing to an item and quickly pressing the mouse button twice, is a convenient shortcut for many of the tasks you will perform in DesignCAD.

Coordinate

The coordinate is the X and Y location of a point on the screen. These X, Y, Delta X, and Delta Y coordinates are displayed in the Coordinate Bar at the top of the screen.

Defaults

Preset option settings such as text size, paper size, and arrowhead type set when DesignCAD is installed.

Dialog Box

The dialog box displays the available command options for you to review or change.

Drag

Dragging the mouse means to hold down the mouse button while moving the mouse.

Drawing Window

The drawing window is the rectangular area on your screen in which you view and work on drawings.

Application Window

An application window is a window that contains a running application. The window displays the menus and provides the workspace for a drawing. The application window shares its borders and title bar with drawing windows that are fully enlarged.

File

A file is a drawing that has been created, then saved under a unique filename. In DesignCAD, all drawings are stored as files.

File Format

The file format refers to the format in which data is stored in a file. DesignCAD stores drawings in .DW2 format.

LandScape

LandScape refers to the horizontal page orientation used with the Print command. The opposite orientation is "portrait," or vertical, orientation.

Macro

A Macro is a sequence of actions that is named and stored. When you run a Macro, DesignCAD performs all of its assigned actions in sequence. Macros can be assigned to keystrokes.

Measurement

A measurement is a measured distance.

Menu

A menu is a list of commands that drops down from the Command Menu bar. The Command Menu bar is displayed across the top of an application window and lists menu names, such as File and Edit.

Merge

To merge means to combine one or more drawings into a single drawing with the Merge command.

Message

A message is a notice on the screen that informs you of a problem or asks for more information. Messages appear in the Status Line at the bottom of your screen, or in a message box.

Normal View

The normal view is the view you see when you start DesignCAD.

Showing Hot Toolbox

Choose the Hot Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Hot Toolbox](#), [Customizing DesignCAD](#)

Options

Options are the choices you have in a dialog box.

Path

The path is the drive, directory, and filename for a drawing. For example, the complete path for DesignCAD 2-D for Windows might be C:\WDC\WDC.EXE.

Portrait

Portrait refers to the vertical page orientation chosen with the Print command. The opposite of portrait is "landscape," or horizontal, orientation.

Printer Driver (File)

The printer driver is a file for each of the printers supported by DesignCAD. The file includes codes for controlling the printer, a list of the fonts available on the printer, and the dimensions of the characters in each font. These files have the extension .PRD. You select a printer driver with the Print Setup command in the File menu.

Read-Only

Read-only refers to a drawing attribute that allows you to view, but not make changes to, a drawing using the same filename. In DesignCAD, you can designate that a drawing be read-only when you open it to prevent accidental changes.

Scale

The scale, or height and width of an object can be changed with DesignCAD.

Units

The units of a drawing are the distance one Drawing Unit represents.

X Coordinate

The X coordinate is the location of the cursor on the X, or horizontal, grid. The X coordinate is displayed in the Coordinate Bar at the top of the screen.

Y Coordinate

The Y coordinate is the location of the cursor on the Y, or vertical, grid. The Y coordinate is displayed in the Coordinate Bar at the top of the screen.

Filename

The filename includes the name and extension of a drawing. A filename can contain up to eight characters. A filename extension includes three characters.

Graphics Tablet

A graphics tablet, or digitizer, is used to move the cursor, set points, and enter commands. With a graphics tablet, you can "trace" a drawing from paper onto the screen.

Hot Toolbox

A Hot Toolbox can be created in DesignCAD to contain those commands that you use most often.

See Also: [Showing Hot Toolbox](#), [Hiding Hot Toolbox](#), [Changing Toolbox columns](#), [Customizing DesignCAD](#)

Rubber Band Line

A rubber band line is drawn while commands are being set with some DesignCAD commands to show how the completed line, arc, circle, or arc will be drawn.

TrueType Fonts

TrueType fonts are used in DesignCAD to eliminate the rough edges printed with some fonts. With TrueType fonts, the printer output more closely resembles the fonts shown on the screen.

AUTOEXEC Macro

The AUTOEXEC Macro can be created to run at the beginning of each DesignCAD session. This Macro can be used to perform tasks necessary for your specific application.

Endpoint

An endpoint is a point at the end of a line to designate the end of the line. This endpoint can be moved with DesignCAD commands. Pressing the **Ctrl** and **Shift** keys allows you to delete an endpoint in Point Select Mode.

Entity

An entity is the combination of lines used to create an object in DesignCAD. A single entity can be drawn with one line or several lines. An entity can be copied, scaled, or saved with DesignCAD commands.

Highlighting Box

The highlighting box is the highlighted lines and select nodes that appear when an object is selected with the Select command.

Highlighting Nodes

The highlighting nodes, or select nodes, are small highlighted squares used to mark the corners of an object selected with the Select command.

Select Node

The select nodes, or highlighting nodes, are small highlighted squares used to mark the corners of an object selected with the Select command.

Nodes

A node is a small highlighted square used to mark the edges of the highlighting box drawn should be shown when an object is selected with the Select command.

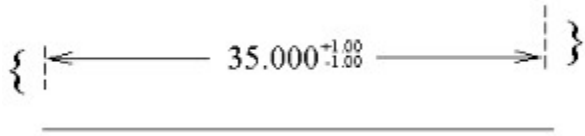
Parts List

A parts list is a list of all the attributes in a drawing. This list is compiled by the Material List command.

Selection Box

The selection box is the highlighted lines and select nodes that appear when an object is selected with the Select command.

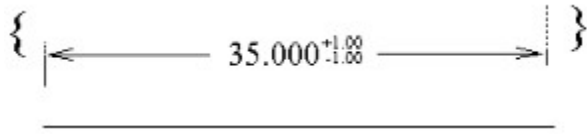
Extension Line



The extension line is the part of the dimension that extends from the point to be dimensioned to the arrowheads of the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Extension Line Overshoot



The extension line overshoot is the distance the extension line extends past the dimension line.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Overshoot

The overshoot is the distance the extension line extends, or "overshoots," past the dimension line.

Precision

The precision of a dimension determines the numeric value that the dimension will be accurate to. For example, you can have the dimension precision set at .0001 Drawing Units, or 100 Drawing Units.

Prefix

The dimension prefix is the characters drawn before the dimension text.

Suffix

The dimension suffix is the characters drawn after the dimension text.

Terminator

The terminator is the arrowhead drawn at the end of the line drawn with the Arrow or Dimension commands. There are 12 arrowhead types.

Tolerance

The dimension tolerance is the amount of variation from the accuracy. These numbers can be displayed as positive or negative values.

Layer

A layer is a subdivision of a DesignCAD drawing. A DesignCAD drawing can have up to 256 layers. Dividing a drawing into layers allows you to plot or print only certain layers. You can save individual layers as separate drawings, delete entire layers, move the objects in one layer to another layer, or change the color, line type, or line width of the objects in a particular layer.

Symbol

A symbol is an independent DesignCAD drawing, or commonly-used figure. To create a symbol, simply draw it and save it.

You can also load a pre-drawn DesignCAD symbol with the [Load Symbol command](#).

Insertion Entity

An "insertion entity" to the drawing file is added to a drawing when a symbol is loaded with the Symbol Load command. The symbol file must be present when the symbol is loaded. If the symbol file is modified, the change will be reflected in all the drawings using that symbol.

Section

A section is a part of the drawing contained in a rectangle defined by dragging the cursor. The defined section is then used with other drawing commands.

Intensity

Intensity refers to the lightness or darkness of a drawing printed with the Print command.

Tick Mark

A tick mark is a small line drawn with the Tick Marks command.

Application Control-menu

The Application Control-menu, in the upper-left corner of the DesignCAD screen displays a menu with commands for sizing and moving the DesignCAD window, switching to other applications, and closing DesignCAD.

Control-Menu

A menu that includes commands with which you can control the size and position of the DesignCAD window and switch to another application.



Shortcut

Keys: **Alt+Space bar**

Attributes

An attribute is information assigned to an entity in a drawing. These attributes can be shown or hidden on the screen.


The Material List command compiles all of the attributes in a drawing into a parts list.

Copy command

Use the Copy command to copy selected text or objects onto the [Clipboard](#). This command is unavailable if you have no selected text or objects.

Copying text or objects to the Clipboard replaces the contents previously stored there.

Shortcuts

Mouse: 

Keys: **Ctrl+C**

See Also: [Copying Objects](#)

Clipboard


The Clipboard is a temporary storage area for cut or copied text or graphic objects. You can paste the contents of the Clipboard into any DesignCAD drawing or into a file of another application. The Clipboard holds the information until you cut or copy another piece of text or a graphic.

Cut command

Use the Cut command to remove selected text or objects from a drawing and put it on the [Clipboard](#). This command is unavailable if you have no selected text or objects.

Cutting text or objects to the Clipboard replaces the contents previously stored there.

Shortcuts

Mouse: 


Keys: **Ctrl+X**

See Also: [Cutting Objects](#)

Paste command

Use the Paste command to insert a copy of the [Clipboard](#) contents into the drawing. This command is unavailable if the Clipboard is empty.

Shortcuts

Mouse: 

Keys: **Ctrl+V**

See Also: [Pasting Objects](#)

Import IGES command

Use the Import IGES command to import a drawing in IGES drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Exporting Drawings](#), [Converting Files](#)

Import HPGL command

Use the Import HPGL command to import a drawing in HPGL drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Exporting Drawings](#), [Converting Files](#)

Import Text command

Use the Import Text command to import a drawing in Text drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Exporting Drawings](#), [Converting Files](#)

Import XY command

Use the Import XY command to import a drawing in XY drawing file format.

Dialog box options

Straight line

Choose the Straight line button to have a straight line drawn between the X and Y coordinate points.

Curved line

Choose the Curved line button to have a curved line drawn between the X and Y coordinate points.

Small squares

Choose the Small squares button to have a small square drawn at each X and Y coordinate location.

Small circles

Choose the Small circles button to have a small circle drawn at each X and Y coordinate location.

Mark size

If the Small squares or Small circles buttons are chosen, enter the size of the squares or circles in the Mark size box.

Color

Choose the Color box to determine the color of the entities.

See Also: [Exporting Drawings](#), [Converting Files](#)

Export IGES command

Use the Export IGES command to export a drawing in IGES drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Exporting Drawings](#), [Converting Files](#)

Export WordPerfect command

Use the Export WordPerfect command to export a drawing in WordPerfect drawing file format.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Exporting Drawings](#), [Converting Files](#)

Show Color Table option

Use the Show Color Table option to display the color table and set the color to be used for subsequent lines or entities drawn in the drawing. The color table in the Toolbox displays the current drawing color.

Dialog box options

Same as

Use the Same as button to have the current color reflect the same color as an entity in the drawing.

First select the command, then choose the color in the color table. Then, choose the Same as button in the color table.

Apply button

To have an entity drawn in a different color select the Apply button.

The color table can be edited with the Set Color command. The new color table can be saved with the Save Color Table command. A new color table can be loaded with the Load Color Table command.

Shortcut

Mouse:



Keys: **H**

See Also: [Setting Drawing Color](#), [Changing Colors](#), [Changing Entity Color](#)

Show Line Style Box option

Use the Show Line Style Box option to display the line type box.

Dialog box options

Line type

The line type is chosen from the Line type box. There are 13 line types available with DesignCAD.

Scale

The Scale of the line is entered in the Scale box in the Line Type box.

Width

The Width of the line is entered in the Width box in the Line Type box.

Fill Wide Line

Choose the Fill Wide Line box to have lines with a width greater than 0.00 filled.

Apply button

Use the Apply button to change the line type, scale, or width of an entity in the drawing.

Same as

Use the Same as button to have the line drawn with the same type, scale, or width of an entity in the drawing.

Shortcuts

Mouse:



Keys: /

See Also: [Line Types](#), [Changing Line Characteristics](#)

Text Options command

Use the Text Options command to display the options for text.

This command changes the font, style, size, angle, and spacing for text. You can also display attributes using this command.

Dialog box options

Font

Use the Font box to choose the font for text in the drawing. The fonts available depend on the fonts installed with your printer.

Font Style

Use the Font Style box to determine the style of the text. The font style can be either normal, italic, bold, or bold italic.

Size

Enter the size for the text in the Size box.

Angle

Enter the angle for the text in the Angle box.

Spacing

Enter the spacing between lines for the Text Block dialog box in the Spacing box.

Display Attribute

Choose the Display Attribute box to have attributes created with the Attribute command displayed in the drawing.

Save Options

Choose the Save Options box to save the text options to the DCW.INI file.

Sample

The Sample box shows an example of the new font and style.

OK

Choose the OK button to use the new text options.

Cancel

Choose the Cancel button to return to the drawing without changing the text options.

Shortcut

Keys: **F7**

See Also: [Text Options](#), [Changing Text Options](#)

Set Handles command

Use the Set Handles command to set handles for a drawing before saving it. One or two points can be set for the lower left and lower right corners of the drawing. If no points are set with this command, the drawing handles default to the lower left and lower right corners of the screen.

See Also: [Setting Drawing Handles](#)

DOS Keystrokes option

Use the DOS Keystrokes option to use the keystrokes associated with DesignCAD for DOS. This command helps you to learn the DesignCAD drawing commands if you are familiar with DOS or are not familiar with Windows.

See Also: [Using DOS Keystrokes](#), [DOS Keystroke Commands](#)

How to Use Help command

Use the How to Use Help command to get more information on how to use the Help Index command. The How to Use Help command explains how to choose a Help topic, use commands and buttons, jump from topic to topic, copy and print Help topics.

See Also: [Finding Information in Help](#), [Help Instructions](#)

About DesignCAD command

Use the About DesignCAD command to display the version number of your copy of DesignCAD for Windows, the copyright notice, installed math coprocessors, and the available disk space and memory.

See Also: [Finding Information on DesignCAD](#)

Parts of DesignCAD Screen

Command Line

Command Menu bar

Coordinate Bar

Rulers

Drawing Area

Main Toolbox

Maximize button

Minimize button

Restore button

Scroll bars

Snap Toolbox

Status Bar

Title Bar


Window Control-menu

Rulers

The vertical and horizontal rulers help you align entities and keep track of where you are in the drawing.

See Also: [Showing Rulers](#), [Hiding Rulers](#)

Maximize Button

The Maximize button  is located at the right end of the title bar in the application and drawing windows when they are not fully enlarged.

When you click the maximize button, the following occurs:


The application window enlarges to fill the screen.

A drawing window enlarges to fill the DesignCAD window.

Note: Maximized windows have no borders.

See Also: Maximize command, Maximizing Windows

Minimize Button

The Minimize button  is located at the right end of the title bar in an application window.

Clicking the Minimize button is the same as choosing Minimize from the application Control-menu. The application window shrinks to an icon.

See Also: Minimize command, Minimizing Windows

Title Bar

The Title Bar is located along the top of a window and contains the name of the application and drawing.

To move the window, drag the title bar.

Note: You can also move dialog boxes by dragging their title bars.

A title bar may contain the following elements:

Application Control-menu button

Drawing Control-menu button

Maximize button


Minimize button

Name of the application

Name of the drawing

Restore button

Restore Button

The Restore button  is located at the right end of the title bar in a maximized application window.

Clicking the Restore button is the same as choosing Restore from the application Control-menu: The application window returns to its previous size.

See Also: [Restore command](#), [Restoring a Drawing Window to Its Former Size](#)

Window Control-Menu

The window Control-menu is located at the left end of the title bar in an application window.

To display commands that control the size and position of an application window, click the application Control-menu box or press **Alt+Space bar**.

The window Control-menu contains these commands:

- Restore command
- Move command
- Size command
- Minimize command
- Maximize command
- Close command
- Next command

Command Menu Bar

The Command Menu bar is located below the title bar, along the top of the application window, and contains DesignCAD commands.

To view all the commands in a menu, do one of the following:

Click the menu name.

-or-

Press **Alt** and the underlined letter of the menu name.

See Also: [Choosing Commands](#), [Using Menus](#)

Command Line

The Command Line is located below the Command Menu bar, at the top of the drawing screen. Before a command is selected, or when a command that does not require any information to be input, the Command Line contains icons for file management commands such as Open, Save, Print, Copy, Cut, and Paste. The Command Line also contains any user-defined macro buttons.

When a command is selected, the Command Line allows you to enter text, and other options for each command. For example, when the Text command is selected the following information is displayed in the Command Line:

The Text is entered in the Text box. The font can be changed in the Font box. The size and angle of the text can be changed in the Size and Angle boxes. The orientation, left, right, or centered, is entered in the Orientation box. Text can be drawn in bold or italic font by choosing the Bold and Italic buttons. Text can be drawn in a block by choosing the Text Block command. Text can be drawn with the same options as other text in the drawing by choosing the Same As button and setting a point on the text in the drawing.

See Also: [Parts of the DesignCAD Screen](#)

Scroll Bars

The Scroll bars are displayed at the right and bottom edges of the drawing window. The scroll boxes inside the scroll bars indicate your vertical and horizontal location in a drawing. You can use the mouse to scroll to other parts of the drawing.

Vertical bar scroll

Item Mouse action

- Click to scroll up one unit.
- Drag to move to an approximate location in the drawing.
- Click to scroll down one unit.

Horizontal scroll bar

Use the horizontal scroll bar to move left and right, just as you use the vertical scroll bar to move up and down.

See Also: [Parts of the DesignCAD Screen](#)

Drawing Area

The Drawing Area is the area in which the entities appear in your drawing. The scroll bars can be used to move the drawing inside the drawing area.

Use the rulers at the top and left of the drawing area to position the cursor within the drawing area.

See Also: [Scroll Bars](#), [Rulers](#)

Alphabetic Listing

-A-

[Aligning text](#)
[Angle dimension](#)
[Arranging icons](#)
[Arranging views](#)

-B-

[Bending lines](#)
[Bold text](#)
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-C-

[Canceling a menu](#)
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[Changing Toolbox columns](#)
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[Creating drawings](#)
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-D-

[Definitions of terms](#)
[Deleting entities](#)
[Deleting a layer](#)
[Deleting points](#)
[Dimension basics](#)
[Dimension options](#)
[Drawing angle dimensions](#)

[Dimensioning arcs](#)
[Dimensioning circles](#)
[Dimensioning lines](#)
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-E-

[Editing drawings](#)
[Editing entities](#)
[Editing layers](#)
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-G-

[Getting color table](#)

-H-

[Help Instructions](#)
[Hiding Coordinate Bar](#)
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[Hiding Snap Toolbox](#)
[Hiding Status Bar](#)
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[Inserting an entity](#)
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-J-

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-L-

[LandScape page orientation](#)
[Left-justified text](#)
[Loading a color table](#)
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-M-

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Maximizing drawings
Maximizing windows
Menus
Merging drawings
Minimizing windows
Mirroring objects
Mouse shortcuts
Moving objects
Moving objects with handles
Moving windows

-N-

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-O-

Opening a file created by another application
Opening drawings
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-P-

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-R-

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Regenerating the screen
Repairing lines
Repeating commands
Retrieving files
Right-justified text
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-S-

Saving color table
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Saving Layers
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Segmenting entities
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-U-

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Undoing commands
Unerasing commands
Units of measurement
Unselecting entities
Using menus
Using symbols
Using the mouse

-V-

Viewing drawings

-W-

Windows

-Z-

Zooming a drawing

Aligning Text

Left-aligning text

After the Text command is selected, choose the Left alignment option in the Orientation box in the Command Line.

Set a point for the lower left corner of the text.

Set a point for the lower right corner of the text.

Centering text

After the Text command is selected, choose the Center alignment option in the Orientation box in the Command Line.

Set a point for the center of the text.

Set a point for the lower right corner of the text.

Right-aligning text

After the Text command is selected, choose the Right alignment option in the Orientation box in the Command Line.

Set a point for the lower right corner of the text.

Set a point for the lower left corner of the text.

See Also: [Text Options command](#), [Centered Text](#), [Left-justified Text](#), [Right-justified Text](#), [Text Alignment](#), [Text Options](#), [Changing Text Options](#).

Angle Dimension

Drawing dimensions aligned to any angle

- Choose the Dimension command.
- Choose the Align to any angle button in the Command Line.
- Set a point for the beginning of the dimension.
- Set a point for the end of the dimension.
- Set a point for the location of the text information.

Drawing angled dimensions without extension lines

- Choose the Dimension Distance Only command.
- Choose the Align to any angle button in the Command Line.
- Set a point for the beginning of the dimension.
- Set a point for the end of the dimension.
- Set a point for the location of the text information.

Drawing angle dimensions

- Choose the Dimension Angle command.
- Set a point for the center, or vertex, of the angle to be measured.
- Set a point for the beginning of the angle to be dimensioned.
- Set a point for the end of the angle to be measured.
- Set a point for the location of the text information.

See Also: [Angular Text Format](#), [Angular Text Precision](#), [Dimension Angle command](#), [Dimension command](#), [Dimension Distance Only command](#), [Dimension Basics](#), [Dimension Options](#), [Drawing Angle Dimensions](#)

Arranging Icons

Different views of the drawing window become icons when they are minimized. You can move these icons around the screen.

Arranging Icons

Choose the Arrange Icons command from the Window menu.

See Also: [Arrange Icons command](#), [Minimize command](#), [Maximize command](#), [Maximizing Windows](#), [Minimizing Windows](#)

Arranging Views

Cascading windows

Choose the Cascade command from the Window menu.

Tiling windows

Choose the Tile Horizontal command from the Window menu.

-or-

Choose the Tile Vertical command from the Window menu.

See Also: [Cascade command](#), [Tile Horizontal command](#), [Tile Vertical command](#), [Cascading Windows](#), [Minimize command](#), [Maximize command](#), [Maximizing Windows](#), [Minimizing Windows](#)

Bending Lines

Activate the Point Select Mode.

Select the line to be bent.

Move the pointer to point to the place in the line that is to be bent.

Set a point.

Move the pointer to the new corner, or vertex, of that line.

Set a point for the new corner of the entity.

See Also: [Editing Lines](#), [Moving Points](#)

Bold Text

Normal text

DesignCAD

Bold text

DesignCAD

Changing text style with command

After the Text command is selected, choose the Bold button in the Command Line.

Changing text style with Text Options

Choose the Text Options command from the Options menu.

Choose the Bold in the Font Style box.

Choose the OK button.

Text can be changed to bold after it is drawn with the [Show Info Box option](#).

Changing style of selected text

Select the text to be changed.

Choose the Show Info Box option.

Choose the Bold button.

See Also: [Text Options](#), [Changing Text Options](#), [Text Options command](#)

Breaking a Line

Select the line to be broken.
Choose the Break Line command.

See Also: [Break Line command](#), [Editing Lines](#)

Canceling a Command

Cancel button

Choosing the Cancel button discards the options you have selected, closes the dialog box, and returns you to your drawing. When DesignCAD completes an action that cannot be canceled, the Cancel button changes to the Close button.

See Also: [Undoing an Action](#)

Cascading Windows

The Cascade command organizes your open windows by lining them up on the screen so that all the title bars show.

Choose the Cascade command from the Window menu.

See Also: [Cascade command](#), [Arranging Windows and Icons](#), [Moving Windows](#), [Opening Views](#), [Tiling Windows](#), [Manipulating the Views](#), [Windows](#)

Centered Text

After the Text command is selected, choose the Center alignment option in the Orientation box in the Command Line.

Set a point for the center of the text.

Set a point for the lower right corner of the text.

See Also: [Aligning Text](#), [Text Alignment](#), [Text Options command](#), [Changing Text Options](#), [Text Options](#)

Chamfering Lines

Choose the Chamfer command.

Type the chamfer depth in the Chamfer depth box in the Command Line.

To keep the original line after the chamfer has been cut, choose the Keep original lines box.

Set a point on the first line to be chamfered.

Set a point on the second line.

See Also: Chamfer command, Editing Lines

Changing Toolbox Columns

You can change the number of columns in a toolbox, using the Options command in DesignCAD.

Changing Main Toolbox columns

Choose the Options command from the Options menu.
Choose the Toolbox category.
Type the new number of columns for the Main Toolbox.

Changing Snap Toolbox columns

Choose the Options command from the Options menu.
Choose the Toolbox category.
Type the new number of columns for the Snap Toolbox.

Changing Hot Toolbox columns

Choose the Options command from the Options menu.
Choose the Toolbox category.
Type the new number of columns for the Hot Toolbox.

See Also: [Options command](#), [Customizing Toolbox Columns](#)

Cleaning up a Line

Select the line to be smoothed.
Choose the Smooth command.

See Also: [Smooth command](#), [Smoothing Lines](#)

Clearing the Screen

Choose the New command from the File menu.

When asked if you want to save the drawing, choose the Yes button to save the drawing.

Choose the No button to clear the screen without saving the drawing.

See Also: [New command](#), [Erasing the Screen](#), [Closing a Drawing](#)

Clicking the Mouse

Pointing to an item on your screen and then quickly pressing and releasing the mouse button is called clicking. You choose entities on the screen and move around in a drawing by clicking. Double-clicking, pointing to an item and quickly pressing the mouse button twice, is a convenient shortcut for many of the tasks you will perform in DesignCAD.

See Also: [Mouse Techniques](#), [Using the Mouse](#)

Closing a View

Double-click the drawing Control-menu box in the upper-left corner of the drawing window.

-or-

Press **Ctrl+F4** to choose the Close command.

See Also: Control-menu Close command

Changing Entity Color

Select the entity to be changed.
Choose the Show Color Table option.
Choose the color.
Choose the Apply button.

Setting color same as an entity

Choose the Show Color Table option.
Choose the Same as box.
Select the entity with the color you want.
Select the entity to be changed.
Choose the Apply button.

Changing current color of selected object

Select the object to be changed.
Choose the Show Info Box option.
Choose the Color button.
Choose the color.

See Also: [Show Color Table option](#), [Show Info Box option](#)

Combining Drawings

DesignCAD allows you to merge more than one drawing into a single drawing. DesignCAD opens the first drawing, then merges the second drawing on top of it. The Drawing Units of the second drawing are matched to those of the drawing already on the screen.

Merging a drawing

Choose the Merge command from the File menu.

Choose the file you want to merge with the current drawing from the File Name box.

Set a point for the lower left corner of the drawing.

Set a point for the lower right corner of the drawing.

Set a point for the upper left corner of the drawing.

Set a point for the upper right corner of the drawing.

See Also: [Merge command](#)

Combining Lines

Activate the Select Mode.

Select the first line.

Press **Shift** and set a point on the next line.

Choose the Combine command from the Edit menu.

See Also: [Combine command](#), [Editing Lines](#)

Converting Curves

Select the curve to be changed.
Choose the Curve To Line command.

See Also: [Curve To Line command](#), [Converting Lines](#)

Converting Drawings

DesignCAD allows you to convert, or transfer, drawing files to and from other programs. A file saved in a drawing application usually includes formatting codes not recognized by other applications.

DesignCAD allows you to import drawings from other drawing file formats into DesignCAD.

Importing DXF file formats

- Choose Import DXF from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing IGES file formats

- Choose Import IGES from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing HPGL file formats

- Choose Import HPGL from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing Text file formats

- Choose Import Text from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing XY file formats

- Choose Import XY from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

DesignCAD allows you to export drawings from other drawing file formats into DesignCAD.

Exporting DXF file formats

- Display the drawing you want to export on the screen.
- Choose Export DXF from the File menu.

Exporting IGES file formats

- Display the drawing you want to export on the screen.
- Choose Export IGES from the File menu.

Exporting WordPerfect file formats

- Display the drawing you want to export on the screen.
- Choose Export WordPerfect from the File menu.

See Also: [Compatible File Formats](#)

Converting Lines

Converting a curve

Select the curve to be changed.
Choose the Curve To Line command.

Curving a line

Select the line to be changed.
Choose the Line To Curve command.

See Also: [Curve To Line command](#), [Line To Curve command](#)

Copying Objects

Activate the Select Mode.
Select the object to be copied.
Move the pointer to the area inside the selection box.
Press **Ctrl** and set a point.
Move the selection box to the new location.
Set a point.

See Also: [Copy command](#), [Copying Objects with Handles](#), [Mirroring Objects](#)

Copying Objects with Handles

Copying an object to an exact location

Activate the Select Mode.

Select the object to be copied.

Press **Ctrl** and click the left mouse button inside the selection box.

Set a point for the handle.

Set a point for the new location of the handle.

-or-

Press **Alt+Ins** to activate the selection box.

Use the arrow keys to move the cursor until the selection box surrounds the object.

Press **Ins**.

Move the pointer inside the selection box.

Press **Ins**.

Move the cursor to the point for the handle.

Set a point.

Move the cursor to the new location of the handle.

Set a point.

Copying several objects to an exact location

Activate the Select Mode.

Select the first object.

Press **Shift** while selecting other objects.

Press **Ctrl** and click the left mouse button inside the selection box.

Set a point for the handle of the objects.

Set a point for the new location of the handle.

-or-

Press **Shift+Ins** to set a point on the other object.

Move the cursor inside the selection box.

Press **Ins**.

Move the pointer inside the selection box.

Press **Ins**.

Set a point for the handle of the objects.

Set a point for the new location of the handle.

See Also: [Copying Objects](#)

Creating Hot Toolbox

Choose the Show Hot Toolbox option from the View menu.
Choose a command from the Main Toolbox.
Press and hold down the left mouse button.
Drag the icon to the Hot Toolbox.
Release the mouse button.

-or-

Choose the Options command.
Choose the Toolbox category.
Choose the icon in the Available Tools box.
Choose the Add button.
Choose the OK button.

Removing icon from Hot Toolbox

Choose the Options command.
Choose the Toolbox category.
Choose the icon in the Current Content box.
Choose the Delete button.
Choose the OK button.

Closing a Toolbox

Choose the Toolbox Control-menu box in the upper left corner of the Toolbox.

See Also: [Hot Toolbox](#), [Changing Toolbox Columns](#), [Customizing Toolbox Columns](#)

Customizing the Color Table

The colors in the [color table](#) can be changed to meet your specifications.

Choose the Set Color command from the Options menu.
Move the sliding box or click the arrows to change the red, green, and blue color values.
Choose the OK button.

Saving the Color Table

Customize the color table with the Set Color command.
Choose the Save Color Table command from the Options menu.
Type a name for the color table in the File Name box.
Choose the OK button.

See Also: [Set Color command](#), [Save Color Table command](#)

Customizing DesignCAD

DesignCAD allows you to determine how the screen looks by moving, changing, and deleting items such as the color table, Toolboxes, Command Line, Status Bar, and Coordinate Bar.

DesignCAD also lets you create your own commands by writing macros and using the BasicCAD programming language.

Customizing the Screen

[Customizing the Color Table](#)

[Creating Hot Toolbox](#)

[Creating Symbol Toolbox](#)

[Showing and Hiding Information](#)

[Customizing Toolbox Columns](#)

Using Macros

[Creating Macros](#)

[Running a Macro](#)

Using BasicCAD

[Running BasicCAD Programs](#)

Customizing Toolbox Columns

You can change the number of columns in a toolbox in DesignCAD.

Changing Main Toolbox columns

Choose the Options command from the Options menu.
Choose the Toolbox category.
Type the new number of columns for the Main Toolbox.
Choose the OK button.

Changing Snap Toolbox columns

Choose the Options command from the Options menu.
Choose the Toolbox category.
Type the new number of columns for the Snap Toolbox.
Choose the OK button.

Changing Hot Toolbox columns

Choose the Options command from the Options menu.
Choose the Toolbox category.
Type the new number of columns for the Hot Toolbox.
Choose the OK button.

See Also: [Options command](#)

Cutting Lines

Activate the Point Select Mode.

Select the entity to be cut.

Press the **Ctrl** key and move the pointer near the line until the scissors appear.

Press the **Ctrl** key and set a point.

See Also: [Editing Lines](#)

Deleting Objects

Select the entity to be erased.
Choose the Erase command from the Edit menu.

Erasing several entities

Activate the Select Mode.
Select the first entity.
Press **Shift** while selecting each entity.
Choose the Erase command from the Edit menu.

Deleting last entity drawn

Choose the Erase Last command from the Edit menu.

See Also: [Erase command](#), [Erase Last command](#), [Editing Entities](#), [Using the Clipboard](#)

Deleting a Layer

Choose the layer to be deleted in the Layer box in the Coordinate Bar.
Choose the Delete Layer command in the Options menu.

See Also: [Delete Layer command](#), [Editing Layers](#)

Deleting Points

Select the entity to be changed.

Press **Ctrl+Shift** while setting point on point to be deleted.

-or-

Activate the Point Select Mode.

Select the entity to be changed.

Set two points on the point to be deleted.

See Also: [Moving Points](#), [Editing Lines](#)

Dimension Basics

There are several options and characteristics that must be understood when using the dimension commands.

[Understanding Dimension Basics](#)
[Dimension options](#)

Dimension Options

When the Dimension command is selected, several dimension options are shown in the Command Line. These options can be changed for each dimension.

Dimension Line button

Position of Dimension Arrows

Dimension Offset

Dimension Arrowhead Type

Dimension Arrowhead Size

Extension Line button

Overshoot Length

Fixed Length

Variable length

Dimension Text button

Linear Text Format

Angular Text Format

Linear Text Precision

Angular Text Precision

Dimension Text Location

Dimension Text Orientation

Dimension Text Size

Dimension Prefix, Suffix button

Dimension Prefix

Dimension Suffix

Dimension Tolerance button

Positive or Negative Dimension Tolerance

Positive and Negative Dimension Tolerance

No Dimension Tolerance

Dimension Tolerance Size

Saving a Vector Dimension

Dragging the Mouse

DesignCAD allow you to "drag" the mouse to choose entities on the screen. To do this, hold down the left mouse button and move the mouse so that the rubber band box on the screen encloses the entities to be selected. Once all the entities are enclosed within the rubber band box let go of the left mouse button.

See Also: [Mouse Techniques](#), [Using the Mouse](#)

Editing Drawings

Editing Entities

Editing Lines

Erasing an Entity

Deleting Entities

Segmenting Entities

Stretching Objects

Editing Entities

Combining Lines

Editing Lines

Bending Lines

Breaking Lines

Chamfering Lines

Combining Lines

Cutting Lines

Deleting Points

Extending Lines

Filleting Lines

Joining Lines

Moving Points

Shortening Lines

Smoothing Lines

Straightening Lines

Stretching a Line

Trimming Lines

Editing Tools

Erasing an Entity

Regenerating the Screen

Undoing and Redoing Commands

"Unerasing" Objects

Using the Clipboard

Erasing an Entity

Select the entity to be erased.
Choose the Erase command from the Edit menu.

-or-

Select the entity to be erased.
Press **Delete** key.

Erasing several entities

Choose the first entity.
Press **Shift** while selecting each entity.
Choose the Erase command from the Edit menu.

-or-

Choose the first entity.
Press **Shift** while selecting each entity.
Press **Delete** key.

Deleting last entity drawn

Choose the Erase Last command from the Edit menu.

See Also: [Erase command](#), [Erase Last command](#), [Deleting Entities](#)

Error Message

If you try to choose a command without meeting certain guidelines, such as selecting entities, an error message will appear in the center of the screen.

Removing an error message

Choose the OK button.

-or-

Press **Enter**.

See Also: [Error Messages](#)

Exiting DesignCAD

If you have made changes to a drawing since you last saved, DesignCAD asks whether or not you want to save the changes before quitting, even if you close Windows without closing DesignCAD first.

Exiting DesignCAD

Choose the Exit command from the File menu.

Shortcut

Mouse: Double-click the application Control-menu button

Keys: **Alt+F4**

See Also: [Exit command](#)

Extending Lines

Select the line to be extended.
Choose the Extend command from the Edit menu.
Set a point on the endpoint to be extended.
Move the pointer to the new location for the endpoint.
Set a point for the new endpoint.
-or-
Activate the Point Select Mode.
Press **Shift** key and select the endpoint.
Move mouse in desired direction.
Set a point for the new endpoint.

See Also: [Extend command](#), [Editing Lines](#)

File Formats

[ASCII Text Files](#)

[AutoCAD DXF Files](#)

[DesignCAD 2-D ASCII Files](#)

[DesignCAD 2-D Binary Files](#)

[DesignCAD 3-D ASCII Files](#)

[DesignCAD 3-D Binary Files](#)

[Hewlett Packard Plotter \(HPGL\) Files](#)

[IGES \(Initial Graphics Exchange Standard\) Files](#)

[WordPerfect Files](#)

[X, Y Coordinate Files](#)

Opening a File Created by Another Application

In DesignCAD, you can open files created with different applications.

DesignCAD imports these file formats:

[Hewlett Packard Plotter \(HPGL\) Files](#)

[ASCII Text Files](#)

[X, Y Coordinate Files](#)

[AutoCAD DXF Files](#)

[IGES \(Initial Graphics Exchange Standard\) Files](#)

To open a file created by another application

Choose one of the Import commands from the File menu.

See Also: [Importing Drawings](#), [Exporting Drawings](#)

Filleting Lines

Choose the Fillet command.

Type the fillet radius in the Fillet radius box in the Command Line.

To keep the original line after the fillet has been cut, click the Keep original lines box.

To have the fillet cut with the same radius as another fillet in the drawing, choose the Same as button.

Then set a point on the fillet in the drawing.

Set a point on the first line to be filleted.

Set a point on the second line.

See Also: [Fillet command](#), [Editing Lines](#)

Changing Text Font

A font is a collection of type faces or styles used to give your text a particular look. Your choice of fonts depends on the printer fonts you have installed.

To change the text font

Choose the Text Options command in the Options menu.
In the Font box, the font type.
Choose the OK button.

Changing current font of selected entity

Select the entity to be changed.
Choose the Show Info Box option.
Choose the Font button.
Choose the font.

See Also: [Text Options command](#), [Show Info Box option](#), [Text Options](#), [Changing Text Options](#)

Help Instructions

You can find information in Help by using the index or Search feature. To return to the Help Index after you read the instructions below, click the Back button or press **B**.

For complete instructions on how to use Help, press **F1** while the Help window is active.

To use the Help Index

Choose an underlined topic.

To scroll in the Help window

Move the scroll bars with a mouse.

-or-

Press the up or down arrow keys.

To return to the previous topic

Choose the Back button.

To open the Search feature in Help

Choose the Search button.

To close the Help window

Choose the Exit command from the Help File menu.

See Also: [Help Index command](#)

Hiding Coordinate Bar

Choose the Coordinate Bar option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Coordinate Bar](#), [Customizing DesignCAD](#)

Hiding Info Box

Choose the Info Box option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Info Box](#), [Customizing DesignCAD](#)

Hiding Main Toolbox

Choose the Main Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Main Toolbox](#), [Customizing DesignCAD](#)

Hiding Snap Toolbox

Choose the Snap Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Snap Toolbox](#), [Customizing DesignCAD](#)

Hiding Status Bar

Choose the Status Bar option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Status Bar](#), [Customizing DesignCAD](#)

Inserting an Entity

Choose the Load Symbol command from the File menu.

Choose the symbol name from the File Name box.

Choose the OK button.

Set a point for the lower left corner of the symbol.

Set a point for the lower right corner of the symbol.

See Also: [Merge command](#), [Load Symbol command](#), [Load Selection command](#), [Loading Objects](#), [Loading Symbols](#)

Intersecting Lines

Choose the Intersect-1 command.
Set a point on one of the lines.

Choose the Intersect-2 command.
Set a point on one of the lines.
Set a point on the intersecting line.

Snapping to intersection without setting a point

Choose the Intersect-1 Move command.
Set a point on one of the lines.

Choose the Intersect-2 Move command.
Set a point on one of the lines.
Set a point on the intersecting line.

See Also: [Intersect-1 command](#), [Intersect-2 command](#)

Italic Text

Normal text

DesignCAD

Italic text



Setting text style within command

After the Text command is selected, choose the Italic button in the Command Line.

Setting text style with Text Options command

Choose the Text Options command in the Options menu.

Choose Italic in the Font Style box.

Choose the OK button.

Changing text style of selected text

Select the text.

Choose Show Info Box from the View menu.

Click the Italic button in the Info Box.

See Also: [Text Options](#), [Changing Text Options](#), [Text Options command](#)

Joining Lines

Choose the Join command from the Edit menu.
Set a point in one corner of the area to be joined.
Set a second point in the opposite corner of the area.

See Also: [Join command](#), [Editing Lines](#)

Shortcut Keys

A	<u>Arc-1</u>
<	<u>Angle, Distance-Two Points</u>
>	<u>Arrow</u>
\$	<u>Attribute</u>
]	<u>Box</u>
 	<u>Break Line</u>
Ctrl+F	<u>Chamfer</u>
O	<u>Circle-1</u>
Ctrl+F4	<u>Close</u>
Ctrl+C	<u>Copy</u>
Ctrl+H	<u>Crosshair</u>
C	<u>Curve</u>
Ctrl+X	<u>Cut</u>
@	<u>Dimension</u>
G	<u>Display Grid</u>
E	<u>Erase</u>
Ctrl+E	<u>Erase Last</u>
Alt+F4	<u>Exit</u>
F	<u>Fillet</u>
Ctrl+W	<u>Fit to Window</u>
.	<u>Gravity</u>
,	<u>Gravity Move</u>
N	<u>Intersect-1</u>
Ctrl+N	<u>Intersect-1 Move</u>
V	<u>Line</u>
K	<u>Line Snap</u>
Ctrl+K	<u>Line Snap Move</u>
Ctrl+F9	<u>Load Selection</u>
F2	<u>Midpoint</u>
Ctrl+F2	<u>Midpoint Move</u>
Y	<u>New</u>
Ctrl+F12	<u>Open</u>
Ctrl+O	<u>Original Size</u>
=	<u>Parallel-1</u>
Ctrl+V	<u>Paste</u>
;	<u>Point Polar</u>
'	<u>Point Relative</u>
Ctrl+S	<u>Point Select Mode</u>
Ins	<u>Point Set</u>
:	<u>Point XY</u>
Shift+S	<u>Preset Point Mode</u>
Ctrl+Shift+F12	<u>Print</u>
(<u>Quarter Circle</u>
Ctrl+Z	<u>Redo</u>
Ctrl+R	<u>Regenerate</u>
F3	<u>Repeat</u>
Shift+F12	<u>Save</u>
F12	<u>Save As</u>
S	<u>Select Mode</u>
)	<u>Semi-Circle</u>
Shift+Alt+A	<u>Show Attribute</u>
H	<u>Show Color Table</u>
Ctrl+Alt+B	<u>Show Coordinate Bar</u>

Shift+Alt+T	<u>Show Hot Toolbox</u>
Ctrl+I	<u>Show Info Box</u>
/	<u>Show Line Style Box</u>
Shift+Alt+M	<u>Show Main Toolbox</u>
Shift+Alt+R	<u>Show Rulers</u>
Shift+Alt+N	<u>Show Snap Toolbox</u>
Shift+Alt+B	<u>Show Status Bar</u>
J	<u>Show Polar Coordinates</u>
Ctrl+G	<u>Snap Grid</u>
&	<u>Stop Recording Macro</u>
T	<u>Text</u>
F7	<u>Text Options</u>
B	<u>Trim Two Lines</u>
Ctrl+B	<u>Trim Double Lines</u>
Ctrl+Z	<u>Undo</u>
!	<u>Unerase</u>
U	<u>Units</u>
Z	<u>Zoom</u>
+	<u>Zoom In</u>
-	<u>Zoom Out</u>
Ctrl+P	<u>Zoom Previous</u>

See Also: Function Keys, DOS Keystroke Commands

LandScape Page Orientation

Choosing LandScape page orientation box rotates your drawing 90 degrees, and prints it sideways on the page.

See Also: [Changing Page Orientation](#), [Portrait Page Orientation](#), [Printing](#), [Printer Setup command](#)

Left-Justified Text

After the Text command is selected, choose the Left alignment option in the Orientation box in the Command Line.

Set a point for the lower left corner of the text.

Set a point for the lower right corner of the text.

See Also: [Centered Text](#), [Right-justified Text](#), [Aligning Text](#), [Text Alignment](#), [Text Options command](#), [Text Options](#), [Changing Text Options](#)

Loading a Color Table

Choose the Load Color Table command from the Options menu.
Choose the color table name from the File Name box.
Choose the OK button.

See Also: Load Color Table command, Getting Color Table

Loading Objects

Choose the Load Selection command from the File menu.

Choose the drawing name from the File Name box.

Choose the OK button.

Set a point for the lower left corner of the object.

Set a point for the lower right corner of the object.

See Also: [Load Selection command](#), [Load Symbol command](#), [Merge command](#), [Loading Symbols](#), [Inserting an Entity](#)

Loading Symbols

Choose the Load Symbol command from the File menu.

Choose the symbol name from the File Name box.

Choose the OK button.

Set a point for the lower left corner of the symbol.

Set a point for the lower right corner of the symbol.

See Also: [Load Symbol command](#), [Load Selection command](#), [Merge command](#), [Loading Objects](#), [Inserting an Entity](#)

Marking Points

DesignCAD offers four ways to mark points in a drawing. These visible points designate reference points such as center of gravity and midpoint indications. A single point defines each point mark.

Marking points

Choose the Point Mark command.

To mark the point with a cross, choose the cross box in the Command Line.

To mark the point with a cross inside a box, choose the box and cross box in the Command Line.

To mark the point with a cross inside a circle, choose the circle and cross box in the Command Line.

To mark the point with a cross inside a circle and a box, choose the box, circle, and cross box in the Command Line.

To change the size of the point mark, type that number in the Point mark size box.

To draw the point mark the same size as a point mark in the drawing, choose the Same as box. Then choose a point mark in the drawing.

Set a point on the point to be marked.

See Also: [Point Mark command](#)

Maximizing Drawings

You can enlarge any drawing window to fill the entire DesignCAD window. When you maximize a drawing window, other open drawing windows remain open, even though they are hidden.

Maximizing a drawing window

Choose the Maximize button in the upper-right corner of the window you want to expand.

See Also: [Maximize command](#), [Maximizing Windows](#)

Maximizing Windows

You can enlarge any drawing window to fill the entire DesignCAD window. When you maximize a drawing window, other open drawing windows remain open, even though they are hidden.

Maximizing a drawing window

Choose the Maximize button in the upper-right corner of the window you want to expand.

See Also: [Maximize command](#), [Maximizing Drawings](#)

Menus

Using commands

[Choosing a Menu Command](#)

[Closing a Menu](#)

[Interrupting an Action](#)

[Command Menu Bar](#)

[Repeating an Action](#)

Using dialog boxes

[Closing a Dialog Box](#)

[Selecting an Item in a List in a Dialog Box](#)

[Setting Options in a Dialog Box](#)

[Typing and Editing in Dialog Boxes](#)

[Typing Measurements in a Dialog Box](#)

Selecting an Item in a List in a Dialog Box

Related dialog box options are often grouped into a scrollable list to save space.

To select an item

Point to an item and click the mouse button.

-or-

Press **Alt** and the under lined letter in the list name. Then press the up and down arrows to select an item.

Note: You can also press a letter to go straight to the first item in the list beginning with that letter.

To select an item

To display the list, click the down arrow.

-or-

Press **Alt** and the down arrow key.
Select an item.

Note: Some boxes include a text area in which you can type a response instead of selection from a list.

To hide the list

Click the down arrow again.

-or-

Press **Alt** and the up arrow.



Use the controls on this box to increase or decrease the measurement one increment at a time.

To select an item

Click the up and down arrows in the dialog box.

-or-

Press the up and down arrows on the keyboard.

Note: This box includes a text area in which you can type a measurement instead of selecting from the list.

See Also: [Using Dialog Boxes](#), [Setting Options in a Dialog Box](#)

Typing and Editing in Dialog Boxes

Some dialog boxes contain text boxes in which you can type a response.

To replace text in a text box

Select the text you want to replace by:

Dragging the mouse pointer over the text you want to replace.

-or-

Holding down the **Alt** key and pressing the underlined letter in the text box.

Type your response. DesignCAD deletes the selected text as soon as you start typing.

See Also: [Using Dialog Boxes](#), [Selecting Options](#)

Typing Measurements in a Dialog Box

DesignCAD accepts measurement for certain items.

To type measurements in a dialog box

Move to the text box in which you want to type a measurement.

Type a number.

See Also: [Using Dialog Boxes](#), [Typing and Editing in Dialog Boxes](#)

Merging Drawings

DesignCAD allows you to merge more than one drawing into a single drawing. DesignCAD opens the first drawing, then merges the second drawing on top of it. The Drawing Units of the second drawing are matched to those of the drawing already on the screen.

Merging drawings

- Choose Merge from the File menu.
- Choose the name of the drawing in the File Name box.
- Choose the OK button.
- Set a point for the lower left corner of the drawing.
- Set a point for the lower right corner of the drawing.
- Set a point for the upper left corner of the drawing.
- Set a point for the upper right corner of the drawing.

See Also: [Merge command](#), [Combining Drawings](#)

Mirroring Objects

[Horizontally Mirroring Objects](#)

[Vertically Mirroring Objects](#)

[Mirroring Objects Based on Custom Mirror](#)

[Mirroring Objects Based on Custom Normal](#)

Horizontally Mirroring Objects

Select the object to be mirrored.

Choose the Mirror command.

Choose the Mirror Horizontally button in the Command Line.

Choose the Make Copy box to have a copy of the entity drawn.

Set a point for the location of the imaginary mirror.

See Also: [Vertically Mirroring Objects](#)

Vertically Mirroring Objects

Select the object to be mirrored.

Choose the Mirror command.

Choose the Mirror Vertically button in the Command Line.

Choose the Make Copy box to have a copy of the entity drawn.

Set a point for the location of the imaginary mirror.

See Also: [Horizontally Mirroring Objects](#)

Mirroring Objects Based on Custom Mirror

Select the entity to be mirrored.

Choose the Mirror command.

Choose the Custom Mirror button in the Command Line.

Choose the Make Copy box in the Command Line to have a copy of the entity drawn.

Set a point for the primary handle of the imaginary mirror.

Set another point for the secondary handle of the imaginary mirror.

See Also: [Vertically Mirroring Objects](#), [Horizontally Mirroring Objects](#)

Mirroring Objects Based on Custom Normal

Select the entity to be mirrored.

Choose the Mirror command.

Choose the Custom Normal button in the Command Line.

Choose the Make Copy box in the Command Line to have a copy of the entity drawn.

Set a point for the primary handle of the "normal."

Set point for the secondary handle of the "normal."

See Also: [Vertically Mirroring Objects](#), [Horizontally Mirroring Objects](#)

Mirror Normal

The mirror "normal" is an imaginary line that is perpendicular to the mirror.

Mouse Shortcuts

The following table provides a review of basic mouse techniques you will need to know to work in DesignCAD. For more information and practice using these skills, see the documentation that came with your mouse.

To	Do This
Point	Position the mouse pointer on or next to something.
Click	Position the pointer then quickly press and release the left mouse button.
Double-click	Position the pointer then quickly press and release the left mouse button twice.
Drag	Position the pointer. Press and hold the left mouse button as you move the mouse to the desired position. Then release the button.

Middle mouse button

Sometimes when you select a DesignCAD command you will need to press **Enter**. If you have a three-button mouse, you can press the middle button for the **Enter** key.

Setting points

You can select a command before setting the points required for the command. For example, you can draw a line by selecting the Line command. Then set the points for the line.

See Also: [Selecting Commands with a Mouse](#)

Moving Objects

Activate the Select Mode.

Select the entity to be moved.

Move the pointer to the area inside the highlighting box.

Press and hold down the left mouse button. Drag the pointer in the desired direction.

Move the selection box to the new location.

Release the mouse button.

-or-

Press **Alt+Ins**.

Use the arrow keys to move the selection box to the new location.

Press **Ins** to set a point.

See Also: [Moving Objects with Handles](#), [Moving Objects Vertically](#), [Moving Objects Horizontally](#)

Moving Objects with Handles

Moving an entity to an exact location

Activate the Point Select Mode.

Select the entity to be moved.

Press and hold down the left mouse button. Drag the pointer across the entity.

Set a point for the reference point.

Move the pointer to the new location.

Press the right mouse button to set a gravity point on the second entity.

-or-

Move the cursor to the upper corner of the entity.

Press **Alt+Ins** to activate the selection box.

Use the arrow keys to drag the selection box across the entity.

Press **Ins** to set a point.

Set a point for the reference point.

Use the arrow keys to move the pointer to the new location.

Press . to set a gravity point on the second entity.

Moving several objects to an exact location

Activate the Select Mode.

Select the first entity.

Press **Shift** while selecting other objects.

Click the left mouse button inside the selection box.

-or-

Press **Ins** to set a point inside the selection box.

Set a point for the handle of the objects.

Set a point for the new location of the handle.

See Also: [Moving Objects](#)

Moving Windows

You sometimes need to move a window to see another window behind it. You cannot drag a window completely off the screen; at least some of the title bar must remain visible. If the DesignCAD window is maximized, it must be restored before you can move it.

Moving a window

Point to the title bar of the window.

Drag it to the new position.

-or-

Press **Alt+** to choose the drawing Control-menu box.

Press **M** to choose the Move command.

Press the arrow keys to position the outline of the window.

When the window is where you want it, press **Enter**.

See Also: [Switching Windows](#)

Naming Drawings

The Save command in DesignCAD saves drawings with any acceptable DOS filename. That is, the filename can be from one to eight characters long, followed by a period and a three-character filename extension. You can use any characters except spaces and the following characters: * ? , ; [] + = \ / : | < >. You cannot use a period except to separate the filename extension.

Filename extensions distinguish between types of files. For example, DesignCAD binary drawing files have the extension .DW2, DesignCAD ASCII drawing files have the extension .DC2, and backup files have the extension .BAK.

If you do not add a filename extension to drawing filenames, DesignCAD adds .DW2 by default.

DesignCAD assumes specific filename extension. If you use other extensions for your files, type the extension every time you type the filename.

DesignCAD uses the following filename extensions:

Extension	Meaning
.DW2	Binary file
.DC2	ASCII file
.BAK	Backup file
.EXE	Program file
.DAT	Data file
.SYS	Program file
.MNU	Menu file
.PMT	Prompt file
.B2D	Drawing Configuration file

See Also: [Saving a New, Unnamed Drawing](#)

New Drawings

After you close a drawing in DesignCAD, you can create a new empty drawing.

Creating a new drawing

Choose the New command from the File menu.

DesignCAD gives the new drawing a temporary name, such as Untitled-View1. You give the drawing a new name when you save it.

See Also: [Your First DesignCAD Drawing](#), [Creating a New Drawing](#), [Saving a New, Unnamed Drawing](#)

Opening Drawings

Before you can work on an existing drawing, it must be open on the screen.

Opening an existing drawing

Choose the Open command from the File menu.

In the File Name box, type the filename of the drawing you want to open.

-or-

Double-click the name in the File Name box or click the arrows in the scroll bar to choose the filename.
Choose the OK button.

If the drawing you want is not listed in the File Name box, choose a new drive or directory in the Open dialog box.

Listing a drawing in the File Name box

Choose the Open command from the File menu.

If the file is in a different sub directory, choose the directory in the Directories box.

If the file is on a different drive, choose the drive letter in the Drives box.

Choose the OK button.

Note: If the Directories box does not display the directory that contains the file, you may have to move higher in the directory hierarchy to find the file. For example, if you have two sub directories under drive C and you want to list the contents of another sub directory, you must first double-click the C:\ directory and then double-click the second directory.

The File menu lists the four most-recently opened drawings.

Opening a recent drawing

Choose the file from the File menu.

See Also: [Creating a New Drawing](#), [Opening a File Created by Another Application](#), [Opening an Existing Drawing](#)

Opening Views

Choose the New Window command from the Window menu.

Drawing commands effect all views of the drawing.

Making a drawing view active

Choose the name of the drawing view you want to make active from the Window menu.

-or-

Click inside the view you want to make active if there is more than one window open.

See Also: [New Window command](#), [Changing the View](#)

Page Orientation

The page orientation refers to how a drawing will be printed on the page. DesignCAD can print a drawing in LandScape or portrait mode.

Setting print orientation

Choose the Print command.

In the Orientation Box, choose the Portrait or LandScape box.

Choose the OK button.

See Also: [Changing Page Orientation](#), [Printing](#), [Printer Setup command](#), [Portrait Page Orientation](#), [LandScape Page Orientation](#)

Page Size

When you start a drawing, DesignCAD assumes you will print on paper that is 8.5 by 11 inches in portrait orientation. However, you can change the paper size loaded into your printer within the Printer Setup options box.

Setting page size

Choose the Printer Setup command in the File menu.

Choose the Setup button.

Choose the Options button.

In the Paper box, choose the size of the paper you want to use.

Choose the OK button.

See Also: [Setting the Paper Size](#), [Printing](#), [Printer Setup command](#)

Setting the Paper Size

DesignCAD prints drawings on paper determined by the size set with the Print command.

Setting the paper size

Choose the Print command

In the Paper Size box, enter the paper margins.

Choose the OK button.

See Also: [Printing](#), [Printer Setup command](#)

Panning Drawings

Choose the Pan command.
Press and hold the left mouse button.
Move the pointer in the desired direction.
Release the mouse button.

-or-

Choose the Pan command.
Press **Alt+Ins**.
Use the arrow keys to move the cursor in the desired direction.
Set a point.

See Also: [Pan command](#)

Portrait Page Orientation

Portrait page orientation forces a drawing to be printed with the drawing width the same direction as the printer width.

See Also: [Changing Page Orientation](#), [Page Orientation](#), [Printing](#), [LandScape Page Orientation](#), [Printer Setup command](#)

Printing

DesignCAD allows you to produce extremely high resolution drawings on ordinary dot matrix and laser printers or plotters.

Once you choose a printer and the appropriate Print Setup options, printing is simple just choose the Print command from the File menu.

The printer you use affects how DesignCAD displays and prints your drawing. For instructions on installing your printer and connecting it to your computer, see your Windows documentation and your printer manual.

With the Print command, you can print an entire drawing or just selected objects. The Print command allows you to change the number of dots-per-inch used to print a drawing, the orientation of the image printed on the paper, and the light intensity used to print the drawing.

See Also: [Printing a Drawing](#), [Printing Several Copies](#), [Printing Part of a Drawing](#)

Printing Part of a Drawing

Select the part of the drawing to be printed.
Choose Print from the File menu.
In the Print box, select Selection.
Choose the OK button.

Shortcut

Keys: **F5**

See Also: [Print command](#)

Plotting Drawings

The plotter you use affects how DesignCAD displays and plots your drawing. For instructions on installing your plotter and connecting it to your computer, see your Windows documentation and your plotter manual.

DesignCAD allows you to produce extremely high resolution drawings on ordinary plotters. Once you choose a plotter and the appropriate Printer Setup options, plotting is simple, just choose the Print command from the File menu.

If you have installed a plotter, first select it from the list of available plotters and printers in the Printer Setup command. Then choose the Print command to send your drawing to the plotter instead of the printer.

See Also: [Printing](#), [Printer Setup](#), [DesignCAD Print Options](#)

Repairing Lines

Choose the Regenerate command from the View menu.

See Also: [Regenerate command](#)

Repeating Commands

Before the command is selected, double-click the command icon in the Toolbox.

-or-

After the command is selected, press **F3** to choose the Repeat command.

See Also: [Repeating an Action](#), [Repeat command](#)

Retrieving Files

Choose the Open command from the File menu.

In the File Name box, type the filename of the drawing you want to open.

Choose the OK button.

See Also: [Opening Drawings](#), [Open command](#), [Loading Files](#)

Right-Justified Text

After the Text command is selected, choose the Right alignment option in the Orientation box in the Command Line.

Set a point for the lower right corner of the text.

Set a point for the lower left corner of the text.

See Also: [Left-justified Text](#), [Centered Text](#), [Text Alignment](#), [Aligning Text](#), [Text Options command](#), [Text Options](#), [Changing Text Options](#)

Rotating Objects

Rotating an object with the keyboard

Select the objects to be rotated.

Choose the Select Rotate command from the Edit menu.

Type the rotation angle in the Rotate angle box in the Command Line.

Press **Enter**.

Rotating an object with the mouse

Activate the Select Mode.

Select the object to be rotated.

Double-click a selection node on the highlighting box activate the rotate mode.

Press and hold down the mouse button on a corner node of the highlighting box.

Move the pointer in the desired direction.

Release the mouse button.

See Also: [Select Rotate command](#)

Saving Color Table

Customize the color table with the Set Color command.
Choose the Save Color Table command from the Options menu.
Type a name for the color table in the File Name box.
Choose the OK button.

See Also: Set Color command, Save Color Table command, Customizing Color Table

Saving Drawings

Choose the Save command from the File menu.
In the File Name box, type a filename.
Choose the OK button.

See Also: [Closing a Drawing](#), [Saving a New, Unnamed Drawing](#), [Saving an Existing Drawing](#), [Save command](#), [Save As command](#), [Saving Drawing to Drive or Directory](#)

Saving an Existing Drawing

The Save command records any changes made to an active drawing. The drawing name and location remain the same. You should save drawings frequently as you work.

To save an existing drawing

Choose the Save command from the File menu.

Shortcut

Keys: **F10**

See Also: [Closing a Drawing](#), [Saving a New, Unnamed Drawing](#), [Saving an Existing Drawing](#), [Save command](#), [Save As command](#)

Saving Layers

Choose the layer to be saved in the Layer box in the Coordinate Bar.
Choose the Save Layer command from the Options menu.
Type the file name in the File Name box.
Choose the OK button.

See Also: [Save Layer command](#), [Editing Layers](#), [Layer Setup command](#)

Segmenting Entities

Choose the Segment command from the Edit menu.

Type the number of sections in the Number of sections box in the Command Line.

Set a point on the entity to be segmented.

See Also: [Segment command](#), [Editing Entities](#)

Selecting Objects

Selecting an Entity

Selecting an Entity with Selection Box

Selecting Several Entities

Selecting All Entities

Selecting Last Entity Selected

Selecting Certain Entities

"Unselecting" Entities

Selecting an Entity

Activate the Select Mode.
Move the pointer near the entity.
Set a point.

See Also: [Using the Mouse](#), [Using a Digitizer](#), [Using the Keyboard](#)

Selecting an Entity with Selection Box

Activate the Select Mode.

Move the pointer to the corner of the entity.

Press and hold down mouse button.

Drag the selection window until the entity is enclosed.

Release the mouse button.

-or-

Use the arrow keys to move the pointer to the corner of the entity.

With the Num Lock off, press **Alt+Ins** to activate the selection box.

Use the arrow keys to "drag" the selection box.

Press **Ins** to set a point.

See Also: [Using the Mouse](#), [Using a Digitizer](#), [Using the Keyboard](#)

Selecting Several Entities

Activate the Select Mode.
Select the first entity.
Press **Shift** while selecting other entities.

Selecting several entities in same area

Activate the Select Mode.
Move the pointer to the corner of the first entity.
Press and hold down the left mouse button.
Drag the mouse until the selection window includes all the entities to be selected.
Release the mouse button.

-or-

Use the arrow keys to move the pointer to the first entity.
Press **Ins** to select the entity.
Press **Alt+Ins** to activate the selection box.
Use the arrow keys to move the pointer until the selection box encloses all entities to be selected.
Press **Ins** to set a point and select the entities.

See Also: [Using the Mouse](#), [Using a Digitizer](#), [Using the Keyboard](#)

Selecting All Entities

Choose the Select All command from the Edit menu.

See Also: [Using the Mouse](#), [Using a Digitizer](#), [Using the Keyboard](#)

Selecting Last Entity Selected

Choose the Select Previous command from the Edit menu.

See Also: [Using the Mouse](#), [Using a Digitizer](#), [Using the Keyboard](#)

Selecting Certain Entities with the Keyboard

Choose the Entity Select command from the Edit menu.
Choose the check box next to the entity types you want to select.
Choose the OK button.

See Also: [Using the Mouse](#), [Using a Digitizer](#), [Using the Keyboard](#)

Selecting Layers

Choose the layer to be selected in the Layer box in the Coordinate Bar.
Choose the Select Layer command in the Options menu.

See Also: [Select Layer command](#), [Editing Layers](#), [Layer Setup command](#)

Setting Points

The basis to using all DesignCAD drawing commands is the location of points. Points determine the diameter of a circle and the radius of an arc. Some commands use control points to draw a curve.

DesignCAD offers a variety of point commands. With DesignCAD, point commands can be selected from the Command Menu, the Snap Toolbox, the digitizer menu, or issued with shortcut keys.

Using Points with DesignCAD Commands

[Setting Regular Points](#)

[Setting Gravity Points](#)

[Snapping to Lines](#)

[Entering Coordinates for Points](#)

Setting the Snap Grid

The snap grid is an imaginary grid placed over your drawing. Each point set with the Point Set command is located exactly on one of the grid points. The cursor movement follows the snap grid.

Enabling the snap grid

Choose the Snap Grid option from the Options menu.

Setting the snap grid size

Choose the Grid Options command from the Options menu.

Type the size in the Snap grid size box.

Choose the Enable snap grid box to enable the snap grid.

Choose the OK button.

See Also: [Snap Grid option](#), [Display Grid option](#)

Setup

The DesignCAD Setup program installs the DesignCAD files, including the DesignCAD program and on-line Help. The Setup program automatically creates the DesignCAD 2-D program group, the DesignCAD 2-D Setup icon, and the DesignCAD 2-D application icon.

Setting up DesignCAD on a hard disk

If Windows is not already running, type **WIN** at the DOS prompt.

-or-

If Windows is running, close any open applications.

Insert the DesignCAD Installation disk (disk 1) into the appropriate drive.

Choose Run from the File menu.

Type **A:INSTALL** or **B:INSTALL**.

Press **Enter**.

To accept the path that Install proposes, choose the Continue button.

-or-

To choose your own directory, type a new path in the Install To box, then choose the Continue button.

If Install detects that the directory in the path doesn't exist, do one of the following:

To create the directory, choose the Yes button.

To edit the path, choose the No button, edit the path in the Install To box, then choose the Continue button.

If Install detects a previous version of DesignCAD in the directory you specified, do one of the following:

To overwrite the directory, choose the Continue button.

To keep the old version, choose the Change Directory button, then type a different path in the Install To box.

When Install asks if you want to update the startup file AUTOEXEC.DCM, do one of the following:

To update the file, choose the Update button.

When Setup is complete, choose the OK button.

See Also: [Installing Symbols](#)

Shortening Lines

Select the line to be extended.
Choose the Extend command from the Edit menu.
Set a point on the endpoint to be shortened.
Move the pointer to the new location for the endpoint.
Set a point for the new endpoint.
-or-
Activate the Point Select Mode.
Press **Shift** and select the endpoint to be shortened.
Move the pointer in the desired direction.
Set a point for the new endpoint.

See Also: [Extend command](#), [Editing Lines](#)

Showing Coordinate Bar

Choose the Coordinate Bar option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Coordinate Bar](#), [Customizing DesignCAD](#)

Showing Info Box

Choose the Info Box option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Info Box](#), [Customizing DesignCAD](#)

Showing Main Toolbox

Choose the Main Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Main Toolbox](#), [Customizing DesignCAD](#)

Showing Snap Toolbox

Choose the Snap Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Snap Toolbox](#), [Customizing DesignCAD](#)

Showing Status Bar

Choose the Status Bar option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Status Bar](#), [Customizing DesignCAD](#)

Smoothing Lines

Select the line to be smoothed.
Choose the Smooth command

See Also: [Editing Lines](#)

Starting DesignCAD

You can start DesignCAD using either the mouse or the keyboard. If Windows is not already running, type **WIN** at the command prompt.

Starting DesignCAD

Double-click the DesignCAD icon.

-or-

Press **Ctrl+Tab** simultaneously until the DesignCAD group is highlighted.

Use the arrow keys to choose the DesignCAD icon.

Press **Enter**.

See Also: [Closing a Drawing and Exiting DesignCAD](#)

Starting the Program

You can start DesignCAD using either the mouse or the keyboard. If Windows is not already running, type **WIN** at the command prompt.

The DesignCAD icon



Starting DesignCAD with a mouse

Double-click the DesignCAD icon.

Starting DesignCAD with the keyboard

Press **Ctrl+Tab** simultaneously until the DesignCAD group is highlighted.

Use the arrow keys to choose the DesignCAD icon.

Press **Enter**.

See Also: [Opening DesignCAD files](#), [Before You Begin Your Work with DesignCAD](#), [Beginning Work on a Drawing](#)

Opening DesignCAD Files

You can start DesignCAD from either Windows or the command prompt.

To start DesignCAD from the Windows Program Manager

Switch to the Program Manager window.

Open the group window that contains the DesignCAD icon.

Double-click the DesignCAD icon.

-or-

Use the arrow keys to select the DesignCAD icon. Then press **Enter**.

To start DesignCAD from the Windows File menu

Choose Run from the Program Manager file.

If DesignCAD is in your path, type **WDC**.

If DesignCAD is not in your path, type the path for DesignCAD -- for example: **C:\WDC\WDC.EXE**.

Choose the OK button.

To start DesignCAD from the command prompt

At the command prompt, type **WIN WDC**.

Press **Enter**.

Tip: To open an existing file automatically when you start DesignCAD from the command prompt, type **WIN WDC filename**.

Straightening Lines

Straightening a curve

Select the curve to be changed.
Choose the Curve To Line command.

Straightening a line

Select the line to be straightened.
Choose the Select Ortho command from the Edit menu.

See Also: [Editing Lines](#)

Stretching Objects

Activate the Select Mode.

Select the object to be stretched.

Select the Stretch command from the Edit menu.

Set a point to the right of the endpoint to be stretched.

Set a point in the opposite corner of the object to be stretched.

Set a point on the object to be stretched.

Set a point for the new location of the endpoint of that object.

See Also: [Editing Entities](#)

Stretching a Line

Activate the Select Mode.

Select the line to be stretched.

Select the Stretch command from the Edit menu.

Set a point to the right of the endpoint to be stretched.

Set a point in the opposite corner of the entity to be stretched.

Set a point on the entity to be stretched.

Set a point for the new location of the endpoint of that line.

See Also: [Editing Lines](#)

Switching Windows

Switching to a Different Drawing Window

Moving Windows

Opening Views

Text Alignment

Left-aligning text

After the Text command is selected, choose the Left alignment option in the Orientation box in the Command Line.

Set a point for the lower left corner of the text.

Set a point for the lower right corner of the text.

Centering text

After the Text command is selected, choose the Center alignment option in the Orientation box in the Command Line.

Set a point for the center of the text.

Set a point for the lower right corner of the text.

Right-aligning text

After the Text command is selected, choose the Right alignment option in the Orientation box in the Command Line.

Set a point for the lower right corner of the text.

Set a point for the lower left corner of the text.

See Also: [Aligning Text](#), [Text Options command](#), [Text Options](#), [Changing Text Options](#)

Tiling Windows

Choose the Tile Horizontal command from the Window menu.

-or-

Choose the Tile Vertical command from the Window menu.

See Also: [Arranging Views](#), [Moving Windows](#), [Cascading Windows](#), [Opening Views](#), [Manipulating the Views](#), [Windows](#)

Trimming Lines

[Trimming One Line](#)

[Trimming Two Lines](#)

[Trimming Between a Line](#)

[Trimming Four Lines](#)

Trimming One Line

Choose the Trim One Line command.

To trim the shorter end, choose the Trim shorter end box in the Command Line.

Set a point on the line to be trimmed.

Set a point on the line intersecting that line.

-or-

Choose the Trim One Line command.

To trim one line, disable the Trim shorter end box in the Command Line.

Set a point on the line to be trimmed.

Set a point on the line intersecting that line.

See Also: [Trim one line command](#)

Trimming Two Lines

Choose the Trim Two Lines command.

To trim the shorter end, choose the Trim shorter end box in the Command Line.

Set a point on the first line.

Set a point on the second line.

-or-

Choose the Trim Two Lines command.

To trim one line, disable the Trim shorter end box in the Command Line.

Set a point on the first line to be trimmed.

Set a point on the line intersecting that line.

See Also: [Trim Two Lines command](#)

Trimming Between a Line

Choose the Trim Between Lines command.
Set a point on the line to be trimmed.
Set a point on the first intersecting line.
Set a point on the second intersecting line.

See Also: [Trim Between Lines command](#)

Trimming Four Lines

Choose the Trim Double Lines command.

To trim the lines within a specified area, choose the Select trim area box in the Command Line.

-or-

To trim certain lines within an area, choose the Select trim lines and area box in the Command Line.

Then select each of the four lines to be trimmed.

Set a point in the corner of the lines to be trimmed.

Set a point in the opposite corner of the lines to be trimmed.

See Also: [Editing Lines](#), [Trim Double Lines command](#)

Undoing Commands

Choose the Undo command from the Edit menu.

See Also: [Redoing Commands](#)

Unerasing Commands

If you erase a command by mistake

Choose the Unerase command from the Edit menu.

See Also: [Erasing Entities](#), [Undoing Commands](#)

Units of Measurement

The units of measurement for a drawing are called Drawing Units. Drawing Units can be inches, centimeters, feet, miles, or any unit of measurement that makes sense for the drawing.

See Also: [Units command](#)

"Unselecting" Entities

If you select an entity by mistake, press the **Shift** key and select the entity again.

-or-

Click outside the highlighting box.

-or-

Set a point outside the highlighting box.

See Also: [Selecting Objects](#)

Using Symbols

[Installing Symbols](#)

[Loading Symbols](#)

[Manipulating Symbols](#)

[Creating Symbols](#)

Viewing Drawings

[Opening a New Window](#)

[Manipulating the Views](#)

Filling Wide Lines

The [Show Line Style Box option](#) allows you to set the width of a line. These wide lines are "hollow," unless they are filled in with the Fill Wide Lines option.

Choose the Fill Wide Lines option in the Options menu.

See Also: [Changing Line Characteristics](#), [Show Line Style Box option](#)

WIN.INI File

WIN.INI is the Windows settings file which contains information about your Windows applications, including the default locations for the DesignCAD files, utilities, fonts, and WDC.INI.

Windows

Arranging all open drawing windows

Closing a window

Expanding a window to maximum size

Opening multiple views of a drawing

Restoring a drawing window to its former size

Sizing a window

Arranging All Open Drawing Windows

If you open several drawing windows, or views, the screen may be cluttered and some windows may be hidden.

Arranging all open drawing windows

Choose Cascade from the Window menu.

See Also: [Arranging Windows and Icons](#), [Windows](#)

Closing a Window

When you finish working on a drawing, you can close the drawing window. DesignCAD prompts you to save changes before it closes the drawing. When you quit DesignCAD, all open drawing windows are closed.

Closing a drawing window

Choose the Close command from the Edit menu.

-or-

Choose the Close command from the drawing Control-menu.

To close all open windows

Choose the Close All command from the Window menu.

See Also: [Opening a New Window](#), [Windows](#)

Expanding a Window to Maximum Size

You can enlarge any drawing window to fill the entire drawing window. When you enlarge, or "maximize," a drawing window, other open drawing windows remain open, even though they are hidden.

Expanding the DesignCAD window to maximum size

Choose the Maximize command from the application Control-menu.

See Also: [Restoring a Drawing Window to Its Former Size](#)

Opening Multiple Views of a Drawing

The New Window command opens several views, or windows, of the current drawing. These views can be "zoomed" and "panned" independently.

Opening multiple views

Choose the New Window command from the Window menu.

Drawing commands effect all views of the drawing.

See Also: [Opening a New Window](#)

Restoring a Drawing Window to Its Former Size

If you have enlarged a [drawing window](#) or an [application window](#) to its maximum size, you must restore it to its former size before you can move or size it.

Restoring a drawing window to its former size

Choose the Restore command from the drawing Control-menu.

See Also: [Expanding a Window to Maximum Size](#), [Restore command](#)

Sizing a Window

You can make a window smaller to view more than one window at a time, or make it larger to see more of a drawing's contents. You cannot move or size a window that has been enlarged to its maximum size.

Sizing a drawing window

If the window is enlarged to its maximum size, restore it to its original size by choosing Restore from the drawing Control-menu.

Choose the Size command from the Control menu.

-or-

Press **Ctrl+F8**.



When the mouse pointer appears as a  , use the arrow keys to select the border you want to move.



When the mouse pointer appears as a  , use the arrow keys to move the borders.

Note: To return the window to its former size, press **Esc**.

When the window is the size you want, press **Enter**.

Sizing an application window

If the window is enlarged to its maximum size, restore it to its original size by choosing Restore from the application Control-menu.

Choose Size from the Control menu.

-or-

Press **Ctrl+F8**.



When the mouse pointer appears as a  , use the arrow keys to select the border you want to move.



When the mouse pointer appears as a  , use the arrow keys to move the borders.

Note: To return the window to its former size, press **Esc**.

When the window is the size you want, press **Enter**.

See Also: [Maximizing Windows](#), [Minimizing Windows](#)

Zooming a Drawing

Choose the Zoom command from the View menu.

Type the zoom factor on the Command Line.

To permanently change the zoom factor of the drawing, click the Static Zoom box on the Command Line.

Set a point for the zoom origin.

Set a point for the zoom destination.

Zooming a drawing with window

Choose the Zoom Window command in the Toolbox.

Set two points for the zoom window.

Zooming in a drawing

Choose the Zoom In command in the Toolbox.

Set a point for the zoom origin.

That point is moved to the center of the screen. The drawing is zoomed according to the zoom factor set in the Options command.

Zooming out a drawing

Choose the Zoom Out command in the Toolbox.

Set a point for the zoom origin.

That point is moved to the center of the screen. The drawing is zoomed according to the zoom factor set in the Options command.

See Also: [Zoom command](#), [Zoom In command](#), [Zoom Out command](#)

Mathematical Calculations and Equations

By inserting operators between numbers in the Calculator dialog box, you can perform the following calculations:

45*23
(6+23)*4
SQRT(9)
SIN(45)

The expression can contain mathematical functions and the following operators:

+ Addition
- Subtraction
* Multiplication
/ Division
^ Raises a number to a power

Commands

Canceling

[Closing a Dialog Box](#)

[Closing a Menu](#)

[Interrupting an Action](#)

[Reversing an Action](#)

Choosing

[Choosing a Menu Command](#)

[Repeating an Action](#)

Macros

[Using Macros](#)

Shortcuts

[Shortcut Keys](#)

[Creating a Hot Toolbox](#)

Closing a Dialog Box

Click the Cancel button.

-or-

Press **Esc**.

See Also: [Using Dialog Boxes](#)

Closing a Menu

Click outside the menu.

-or-

Press **Esc**.

Note: After you press **Esc** to close a menu, the menu name is still selected. To return to your drawing, press **Esc** again.

See Also: [Canceling a Menu](#)

Interrupting an Action

Press **Esc**.

If DesignCAD does not stop the process, press **Esc** again.

See Also: [Canceling a Command](#)

Reversing an Action

Choose the Undo command from the Edit menu.

Note: You must choose Undo before performing any other action.

See Also: [Undoing an Action](#), [Undoing Commands](#)

Choosing a Menu Command

Point to the menu name.

To display the menu, click the left mouse button.

Point to a command name.

Click the command name with the left mouse button.

-or-

Press **Alt** and the underlined letter in the menu name.

To choose a command, press the underlined letter in the command name.

A command name followed by an ellipsis (...) indicates that a dialog box exists so you can set the options you want.

See Also: [Setting Options in a Dialog Box.](#)

Setting Options in a Dialog Box

Point to the option and click the left mouse button.

-or-

Hold down the **Alt** key and press the underlined letter in the option name.

To carry out the options you set

Choose the OK button.

-or-

Press **Enter**.

To cancel changes to options settings

Choose the Cancel button.

-or-

Press **Esc**.

See Also: [Using Dialog Boxes](#)

Repeating an Action

Before the command is selected

Double-click the command in the Toolbox.

After the command is selected

Press **F3**.

See Also: [Repeating Commands](#)

Using Macros

Creating

[Getting Help on Creating Macros](#)

[Recording a Macro](#)

[Creating Macros that Run Automatically](#)

[Creating User-Defined Macros](#)

Running

[Running a Macro](#)

Reference information

[BasicCAD Programming Language](#)

Getting Help on Creating Macros

For help on creating, changing, and running macros, see [Recording a Macro](#).

For a list of all BasicCAD statements and functions, including descriptions, syntax, and examples, see [Using BasicCAD](#).

See Also: [Creating Macros](#)

Recording a Macro

Choose the Record Macro command from the File menu.
Type the name for the macro in the Record Macro Name box.
Choose the OK button.
Set a point for the startup cursor position.
Perform the actions you want to record.
Choose the Stop Recording Macro command.

See Also: [Getting Help on Creating Macros](#), [Recording Macro Options](#)

Creating Macros that Run Automatically

Choose the Options command from the Options.

Choose the Macro category.

Choose the Startup Macro box.

Choose the startup macro name.

Choose the OK button.

See Also: [Creating Macros](#)

Running a Macro

Choose the Execute Macro command from the File menu.
Select the macro you want to run in the Macro Name box.
Choose the OK button.
Set a point for the startup cursor position.

See Also: [Creating Macros](#), [Using Macros](#)

Creating Drawings

[Your First DesignCAD Drawing](#)

[Creating a New Drawing](#)

[Saving a New, Unnamed Drawing](#)

Creating a New Drawing

Choose New from the File menu.

Note: DesignCAD gives the new drawing a temporary name, such as DesignCAD 2-D (Untitled View1), until you save it with a unique filename.

See Also: [New Drawings](#), [Your First DesignCAD Drawing](#), [New command](#), [Creating Drawings](#)

Saving a New, Unnamed Drawing

Choose Save from the File menu.

-or-

Choose Save As from the File menu.

To save the drawing in the current drive and directory, type a name of up to eight characters in the File Name box.

Choose the OK button.

Until you give a new drawing a unique filename, DesignCAD temporarily names new drawings.

Note: DesignCAD automatically adds a .DW2 extension unless you specify a different extension.

See Also: [Saving Your Drawings](#), [Saving Drawings](#), [Saving Drawing to Drive or Directory](#)

Your First DesignCAD Drawing

[Before You Begin Your Work with DesignCAD](#)

[Starting DesignCAD](#)

[Beginning Work on a Drawing](#)

[Working with Drawings](#)

[Saving Your Work](#)

[Closing a Drawing and Exiting DesignCAD](#)

Keyboard, Mouse and Digitizer

Keyboard

Controlling the Cursor

Function Keys

Keyboard Techniques

Selecting Commands with the Keyboard

Selecting Objects with the Keyboard

Shortcut Keys

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Using Dialog Boxes

Mouse

Mouse Techniques

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Digitizer

Digitizer Techniques

Selecting Commands with a Digitizer

Selecting Entities with a Digitizer

Using a Digitizer

Function Keys

You can use the function keys on your keyboard to perform many tasks faster.

Selecting Commands with the Keyboard

Pressing the **Alt** key and the underlined letter of each menu accesses the Command Menu. When you press **Enter**, the menu will "pull-down." Pressing the underlined letter of the desired command selects commands from the menus.

To choose a command from a menu, you can press a shortcut key. In DesignCAD, shortcut keys are listed after the command names in the Command Menu.

Choosing the Open command

Press **Alt+F** to open the File menu.

Press **O** to choose the Open command.

-or-

Press **Ctrl+F12** to choose the Open command.

When you open a menu the first command is highlighted. Use the arrow keys on the keyboard to move the highlight bar and choose a command.

Secondary menus

To choose the Load Selection command, choose the word "Load" from the Edit menu. Then move the highlight bar to **Load Selection** and press the **Enter** key. Because an arrow follows the word "Load," a secondary menu with more commands will "pop up" on the screen.

See Also: [Using the Keyboard, Keyboard Techniques](#)

Controlling the Cursor

The arrow keys or the number keys on the numeric keypad move the cursor, or pointer. If the **Num Lock** is on, the cursor moves in small steps when you press the number keys. The same is true if you press the **Shift** key and the arrow keys simultaneously. If the **Num Lock** is off, the cursor moves in larger steps.

See Also: [Using Arrow Keys](#), [Setting the Cursor Step Size](#)

Selecting Objects with the Keyboard

Selecting an Entity with Keyboard

"Unselecting" an Entity with Keyboard

Selecting Several Entities with Keyboard

Selecting All Entities with Keyboard

Selecting Last Entity Selected with Keyboard

Selecting Certain Entities with Keyboard

Selecting an Entity with Keyboard

Press **S** to activate the Select Mode.
Use the arrow keys to move the pointer to the entity.
Set a point on the entity to select it.

"Dragging" the selection box

Press **S** to activate the Select Mode.
Use the arrow keys to move the pointer to the corner of the entity.
With the Num Lock off, press **Alt+Ins** to activate the selection box.
Use the arrow keys to "drag" the selection box.
Press **Ins** to set a point.

See Also: [Using the Keyboard](#), [Controlling the Cursor](#), [Using Arrow Keys](#)

Unselecting an Entity with Keyboard

If you select an entity by mistake, press the **Shift** key and select the entity again.

-or-

Set a point outside the highlighting box.

See Also: [Using the Keyboard](#), [Controlling the Cursor](#), [Using Arrow Keys](#)

Selecting Several Entities with Keyboard

Selecting several entities one at a time

Press **S** to activate the Select Mode.

Use the arrow keys to move the pointer to the first entity.

Set a point on the first entity to be selected.

Move the pointer to the next entity to be selected.

Press **Shift+Ins** to set a point on the next entity to be selected.

Selecting several entities in the same area with Enclose Mode

Press **S** to activate the Select Mode.

Use the arrow keys to move the pointer to the first entity.

Press **Alt+Ins** to activate the selection box.

Use the arrow keys to move the pointer until the selection box encloses all entities to be selected.

Press **Ins** to set a point and select the entities.

Selecting several entities in same area with Touch Mode

Press **S** to activate the Select Mode.

Use the arrow keys to move the pointer to the first entity.

Press **Alt+Ins** to activate the selection box.

Use the arrow keys to move the pointer until the selection box touches all entities to be selected.

Press **Ctrl+Ins** to select the entities.

See Also: [Using the Keyboard](#), [Controlling the Cursor](#), [Using Arrow Keys](#)

Selecting All Entities with Keyboard

Press **Alt+E** to open the Edit menu.

Press **A** to choose the Select All command.

See Also: [Using the Keyboard](#), [Controlling the Cursor](#), [Using Arrow Keys](#)

Selecting Last Entity Selected with Keyboard

Press **Alt+E** to open the Edit menu.

Press **v** to choose the Select Previous command.

See Also: [Using the Keyboard](#), [Controlling the Cursor](#), [Using Arrow Keys](#)

Using Dialog Boxes



The OK, Cancel, and Close command buttons appear in many dialog boxes. You can choose one of these buttons when you finish setting options in a dialog box.

OK button

The OK button closes the dialog box and completes the command using the selected options. For example, choosing the OK button in the Open dialog box opens the drawing. To choose the OK button, click the button or press **Enter** when it is selected.

Cancel button

Choosing the Cancel button discards the options you have selected, closes the dialog box, and returns you to your drawing. To choose the Cancel button, press the **Esc** key.

Close button

When DesignCAD completes an action that cannot be canceled, the Cancel button changes to the Close button.

The Close button closes the dialog box without reversing any completed changes. You can also use the Close command on the Control menu in the upper-left corner of the dialog box.

The **Tab**, **Space bar**, and arrow keys move the selection box in Command Line and Dialog boxes.

See Also: [Closing a Dialog Box](#), [Typing and Editing in Dialog Boxes](#), [Selecting Options in a Dialog Box](#), [Selecting an Item in a List in a Dialog Box](#)

Using Arrow Keys

If you are not using a mouse, you can select menus by pressing **Alt** and the arrow keys. Once a menu is pulled down, you can choose the command you want by pressing the arrow keys. You can choose a command from the menu by pressing **Enter**.

See Also: [Controlling the Cursor](#)

Mouse Techniques

The following table provides a review of basic mouse techniques you will need to know to work in DesignCAD. For more information and practice using these skills, see the documentation that came with your mouse.

To	Do This
Point	Position the mouse pointer on or next to something.
Click	Position the pointer then quickly press and release the left mouse button.
Double-click	Position the pointer then quickly press and release the left mouse button twice.
Drag	Position the pointer. Press and hold the left mouse button as you move the mouse to the desired position. Then release the button.

Middle mouse button

Sometimes when you select a DesignCAD command you will need to press **Enter**. If you have a three-button mouse, you can press the middle button for the **Enter** key.

Setting points

The Preset Point Mode allows you to set points before selecting a command. If the Preset Point Mode is not active, you must select a command before setting the points required for the command. For example, you can draw a line by selecting the Line command. Then set the points for the line.

See Also: [Mouse Shortcuts](#)

Selecting Entities with a Mouse

Selecting Entity

"Unselecting" an Entity

Selecting Several Entities

Selecting All Entities

Selecting Last Entity Selected

Selecting Certain Entities

Selecting an Entity with the Mouse

Activate the Select Mode in the Toolbox.
Move the pointer near the entity.
Click the left mouse button.

"Dragging" the selection box

Activate the Select Mode in the Toolbox.
Move the pointer to the corner of the entity.
Press and hold down mouse button.
Drag the selection window until the entity is enclosed.
Release the mouse button.

See Also: [Using the Mouse](#), [Mouse Techniques](#)

Unselecting an Entity with the Mouse

If you select an entity by mistake, press the **Shift** key and select the entity again.

-or-

Click outside the highlighting box.

See Also: [Using the Mouse](#), [Mouse Techniques](#)

Selecting Several Entities with the Mouse

Selecting several entities one at a time

Activate the Select Mode in the Toolbox.
Select the first entity.
Press **Shift** while selecting other entities.

Selecting several entities in same area with Enclose mode

Activate the Select Mode in Toolbox.
Move the pointer to the corner of the first entity.
Press and hold down the left mouse button.
Drag the mouse until the selection window includes all the entities to be selected.
Release the mouse button.

Selecting several entities in same area with Touch mode

Activate the Select Mode in Toolbox.
Move the pointer to the corner of the first entity.
Press and hold down **Ctrl** and the left mouse button.
Drag the mouse until the selection window includes all the entities to be selected.
Release the mouse button.

See Also: [Using the Mouse](#), [Mouse Techniques](#)

Selecting All Entities with Mouse

Choose the Select All command from the Edit menu.

See Also: [Using the Mouse](#), [Mouse Techniques](#)

Selecting Last Entity Selected with Mouse

Choose the Select Previous command from the Edit menu.

See Also: [Using the Mouse](#), [Mouse Techniques](#)

Selecting Certain Entities with Mouse

Choose Entity Select from the Edit menu.

Click the check box next to the entity types you want to select.

Click the OK button.

See Also: [Using the Mouse](#), [Mouse Techniques](#)

Moving Vertically and Horizontally

The mouse can be forced to move horizontally by pressing **Ctrl+Alt**. Pressing **Shift+Alt** forces the mouse to be moved vertically.

See Also: [Using the Mouse](#)

Digitizer Techniques

The digitizer moves the cursor, or pointer, sets points, and chooses DesignCAD commands.

To	Do This
Point	Position the pointer on or next to something.
Click	Position the pointer then quickly press and release the button.
Double-click	Position the pointer then quickly press and release the button twice.
Drag	Position the pointer. Press and hold the button as you move the cursor to the desired position. Then release the button.

Setting points

The Preset Point Mode allows you to set points before selecting a command. If the Preset Point Mode is not active, you must select a command before setting the points required for the command. For example, you can draw a line by selecting the Line command. Then set the points for the line.

See Also: [Using a Digitizer](#)

Selecting Entities with a Digitizer

Selecting an Entity with Digitizer

"Unselecting" an Entity with Digitizer

Selecting Several Entities with Digitizer

Selecting an Entity with Digitizer

Activate the Select Mode in the Toolbox.
Move the pointer near the entity.
Click the first digitizer button.

"Dragging" the selection box

Activate the Select Mode in the Toolbox.
Move the pointer to the corner of the entity.
Press and hold down the digitizer button.
Drag the selection window until the entity is enclosed.
Release the digitizer button.

See Also: [Using a Digitizer](#), [Digitizer Techniques](#)

Unselecting an Entity with Digitizer

If you select an entity by mistake, press the **Shift** key and select the entity again.

-or-

Click outside the highlighting box.

See Also: [Using a Digitizer](#), [Digitizer Techniques](#)

Selecting Several Entities with Digitizer

Selecting several entities at a time

Activate the Select Mode in the Toolbox.
Select the first entity.
Press **Shift** while selecting other entities.

Selecting several entities in same area with Enclose mode

Activate the Select Mode in Toolbox.
Move the pointer to the corner of the first entity.
Press and hold down the first digitizer button.
Drag the digitizer until the selection window includes all the entities to be selected.
Release the digitizer button.

Selecting several entities in same area with Touch mode

Activate the Select Mode in Toolbox.
Move the pointer to the corner of the first entity.
Press and hold down **Ctrl** and the first digitizer button.
Drag the digitizer until the selection window includes all the entities to be selected.
Release the digitizer button.

See Also: [Using a Digitizer](#), [Digitizer Techniques](#)

Selecting Commands with a Digitizer

If you have a digitizer, you can choose commands from the Command Menu. To choose a drawing command, move the digitizer cursor, or stylus, to that menu and press the first button on the cursor.

Selecting Open command

Choose File menu.

Choose Open command.

-or-

Choose Open command on Command Line.

See Also: [Using a Digitizer](#), [Digitizer Techniques](#)

Using a Digitizer

DesignCAD supports many different digitizers, also called graphics tablets and digitizing tablets.

The digitizer moves the cursor, or pointer, sets points, and enters DesignCAD commands. To move the pointer, merely move the stylus or cursor around the digitizer. The DesignCAD pointer will move around the screen in the same way.

Your digitizer may have either a stylus, a pen-like device; or a cursor, also called a puck. Depressing a stylus onto the digitizer sets a point. Pressing the button or barrel switch on a stylus sets a gravity point.

See Also: [Digitizer Techniques](#), [Selecting Commands with a Digitizer](#), [Selecting Entities with a Digitizer](#)

Selecting Commands with a Mouse

If you have a mouse, you can move the pointer to a menu in the Command Menu.

You can use the mouse or the arrow keys to move the highlight bar up and down the menus to choose a command.

You can use the mouse to choose a command from the Command Menu.

Selecting a command

Point to a menu, then click the left mouse button.

Point to a command in the menu, then click the left mouse button.

-or-

Point to a command in the Toolbox, then click the left mouse button.

-or-

Point to a command in the Command Line, then click the left mouse button.

See Also: [Mouse Shortcuts](#), [Using a Mouse](#), [Mouse Techniques](#)

BasicCAD Programming Language

The following is an alphabetic list of all BasicCAD statements and functions.

-A-

ANYKEY Statement
Assignment Statement

-C-

CHAIN Statement
CLEAR Statement
CLOSE Statement
CLS Statement
COLOR Statement

-D-

DIM Statement
DO WHILE Statement

-E-

END Statement
ENTITY Statement
EXIT DO Statement
EXIT FOR Statement

-F-

FOR ... NEXT Statement

-G-

GET ATTR Statement
GET Statement
GET XY Statement
GOSUB Statement
GOTO Statement

-I-

IF (Group) Statement
IF (single line) Statement
INPUT Statement
INPUT# Statement

-L-

LABELS Statement
LAYER Statement
LOCATE Statement
LOOP Statement

-M-

MESSAGE Statement

-O-

ON ERROR Statement

OPEN Statement

-P-

POINTVAL Statement

PRECISION Statement

PRINT # Statement

PRINT Statement

PUT Statement

PUTATTR Statement

-R-

REGEN Statement

RESUME Statement

RETURN Statement

RUN Statement

-S-

SET POINT Statement

STOP Statement

-T-

TAB Statement

-W-

WCLOSE Statement

WINDOW Statement

Built-In Functions:

SYS Function

SYS\$ Function

DesignCAD Commands in Program:

ENCRYPT Utility

Assignment Statement

Use the Assignment statement to assign a value to a numeric variable.

Syntax:

variable = expression

stringvar = expressionlist

Variable can be any numeric variable name. Expression can be any valid numeric expression. Stringvar is any valid string variable name. Expressionlist is one or more string or numeric expressions separated by commas. Numeric expressions can be assigned to a string - they are evaluated and converted to ASCII format. Using more than one expression allows you to concatenate strings.

ANYKEY Statement

Use the ANYKEY statement to wait for a single key press. This statement waits for and reads a keystroke from the keyboard. This statement can be used with or without a variable. If a variable is used, the value of the key pressed is assigned to that variable. String variables or numeric variables can be used. If a numeric variable is used, the ASCII code for the keystroke is assigned to the variable.

This statement is useful if you want to pause for a keypress after printing to the bottom of the screen or a text window.

Syntax:

ANYKEY {*variable*}

CHAIN Statement

Use the CHAIN statement to transfer control to another BasicCAD program. This statement is used to run another BasicCAD program, leaving all the variables from the current program intact when the other program is executed. A string variable or constant can be used for the program name.

The CHAIN statement causes the new program (*programname*) to replace the current program in memory. To return to the original program, you must execute another CHAIN or RUN statement from the new program. The execution of a program called by the CHAIN statement always begins at the first line in the called program.

The RUN statement is similar to the CHAIN statement, but the RUN statement clears all variables before executing the new program.

Syntax:

CHAIN *programname*

See Also: [RUN Statement](#)

CLEAR Statement

Use the CLEAR statement to erase all variables in the program. This statement is used to erase and de-allocate all variables in the program. It can be used to free memory or to re-initialize variables.

Syntax:
CLEAR

CLOSE Statement

Use the CLOSE statement to close a disk file. This statement is used with the OPEN statement. If a disk file has been opened for input or output, it should be closed after being used.

A disk file left open when the program ends is automatically closed. However, the CLOSE statement must be used to close an open file before that file number is used again.

Syntax:

CLOSE {*filenumber*}

Filenumber is the number of the file to be closed. If it is omitted, all open files will be closed.

CLS Statement

Use the CLS statement to clear a text window. This statement clears all the text inside the text window. If a text window is not open, then this statement has no effect.

Syntax:

CLS

See Also: [WINDOW Statement](#)

COLOR Statement

Use the COLOR statement to set the text color in a text window. This statement is used to set the color of the text in a text window. The COLOR statement affects only the text displayed with PRINT statements after the COLOR statement is executed. The text color remains set at the specified color until the next COLOR statement is executed.

Syntax:

COLOR *foreground* {,*background*}

Foreground is the foreground color for the text. Background is the background color. The colors can range from 0 to 255.

DIM Statement

Use the DIM statement to specify the size of one or more arrays. The DIM statement is used to allocate storage for arrays. An array is a single variable with several elements, addressed with a subscript in parentheses such as A(20).

An array must be dimensioned before it is used. Any numeric expression can be used for the array size. String arrays are allowed. All arrays are one-dimensional - that is, an array can have only one subscript.

An array cannot be re-dimensioned. A DIM statement must be executed only once. A variable array must be dimensioned before the variable is used.

Syntax:

DIM *variable (expression) {,variable(expression)...}*

DO WHILE Statement

Use the DO WHILE statement with a LOOP statement to execute a series of instructions in a loop.

The DO WHILE statement is used to start a loop that will be executed as long as the logical expression *logexpression* is true. When a LOOP statement is encountered, program execution is transferred back to the DO WHILE statement and the logical expression is checked again. When the expression is false, execution continues after the LOOP statement.

The logical expression consists of one or more relational expressions separated by AND or OR. A logical expression can also be preceded by NOT.

A relational expression consists of two numeric or string expressions separated by one of the following relational operators: <, <=, =, >=, >, <>.

When using a DO loop, be sure the logical expression will eventually change to False, or the loop will never end. (Any BasicCAD program can be terminated by pressing the **Esc** key, however.

DO WHILE loops can be nested up to eight levels deep.

Syntax:

DO WHILE *logexpression*

.

.

.

LOOP

LOOP Statement

Use the LOOP statement with a DO WHILE statement to execute a series of instructions in a loop.

The DO WHILE statement is used to start a loop that will be executed as long as the logical expression *logexpression* is true. When a LOOP statement is encountered, program execution is transferred back to the DO WHILE statement and the logical expression is checked again. When the expression is false, execution continues after the LOOP statement.

The logical expression consists of one or more relational expressions separated by AND or OR. A logical expression can also be preceded by NOT.

A relational expression consists of two numeric or string expressions separated by one of the following relational operators: <, <=, =, >=, >, <>.

When using a DO loop, be sure the logical expression will eventually change to False, or the loop will never end. (Any BasicCAD program can be terminated by pressing the **Esc** key, however.

DO WHILE loops can be nested up to eight levels deep.

Syntax:

DO WHILE *logexpression*

.

.

.

LOOP

END Statement

Use the END statement to terminate the program and return to DesignCAD. The END statement can be placed anywhere in the program. An END statement at the end of the program is optional. This statement does the same thing as the STOP statement.

Syntax:
END

See Also: [STOP Statement](#)

ENTITY Statement

Use the ENTITY statement to access an entity in the drawing by entity number. The ENTITY statement is used to read an entity into the SYS function variables 90-99, and set points on the screen for each point of the entity. This makes it possible to process all entities in the drawing.

Syntax:

ENTITY *expression*

EXIT DO Statement

Use the EXIT DO statement to exit a DO loop prematurely. This statement can be used to exit a DO loop from anywhere within the loop. This makes it easy to exit a DO loop without using a label and a GOTO statement.

Syntax:
EXIT DO

EXIT FOR Statement

Use the EXIT FOR statement to exit a FOR NEXT loop prematurely. This statement can be used to exit a FOR-NEXT loop from anywhere within the loop. This makes it easy to exit a FOR-NEXT loop without using a label and a GOTO statement.

Syntax:
EXIT FOR

FOR ... NEXT Statement

Use the FOR...NEXT statement to execute a section of the program a certain number of times.

The program statements after the FOR statement and before the NEXT statement are called the loop. Each time the loop is executed, the counter is incremented. If it is greater than the test value, the loop is exited and the program branches to the line following the NEXT statement. (If the STEP *expression* is negative, then the loop is exited when the counter is less than then test value.)

FOR-NEXT loops can be nested, that is, one FOR-NEXT loop can be placed inside another. The FOR-NEXT loops can be nested up to 8 levels deep.

Syntax:

FOR *variable* = *expression* TO *expression* {STEP *expression*}

The *variable* is used as the counter. The first *expression* is the initial value of the counter variable. The second *expression*, after TO, is the test or final value of the counter. The optional STEP *expression* can be used to specify the amount that the counter is incremented each iteration.

GET Statement

Use the GET statement to read a record from a random access file. The record length is specified in the OPEN statement.

When the GET statement is executed, the designated record (*recordnumber*) of the file will be read into the string variable (*stringvar*). The MKS\$ and the CVS functions can be used to convert numeric values to and from four-byte strings file input and output.

Syntax:

GET *file*, *recordnumber*, *stringvar*

GETATTR Statement

Use the GETATTR statement to get entity type, group status, and layer of an entity.

Syntax:

GETATTR *entity*, *type* {*,select*, *layer*, *group*, *color*}

Entity is the entity number - 1 for the first entity in the drawing, 2 for the second, etc. *Type* is the entity type, as shown:

1 = Line
2 = Oval
3 = Text
4 = Curve
5 = Arc
11 = Bezier Curve
15 = Attribute
16 = Circle, Circular Arc
17 = Hatch
21 = New Layer
22 = Text Arc
23 = Layer Names
24 = Arrow
26 = Symbol
31 = Plane
32 = Grid
33 = Grid Line
70 = Point Mark
74 = Dimension, Angle
75 = Dimension
76 = Dimension, Diameter/Radius
77 = Dimension, Arc
78 = Dimension, Radius Progressive
79 = Dimension, Progressive
80 = Dimension, Chamfer
81 = Dimension, Coordinate
90 = Bitmap Image

Select is 0 if the entity is not selected, or 1 if the entity is currently selected. *Layer* is the layer number of the entity. *Group* is the group ID number if the entity is part of a Group, or 0 otherwise. *Color* is the color of the entity.

Type, *select*, *layer*, *group*, and *color* must be BasicCAD variables, not expressions, since they will be assigned values.

GETXY Statement

Use the GETXY statement to get the X and Y coordinates of the current cursor position. This statement can be used to assign the X and Y coordinates of the current cursor position to two variables. It is very useful in getting the "current" position to provide a reference location for the rest of the program.

Syntax:

GETXY *variable variable {variable}*

GOSUB Statement

Use the GOSUB statement to call a subroutine. This statement is used to transfer program execution to another statement out of the normal sequence of execution, and resume execution after the GOSUB statement when a RETURN statement is encountered - it calls a subroutine. A RETURN statement must be used to return from a subroutine called by a GOSUB statement.

Syntax:

```
GOSUB label  
RETURN
```

Label can be any valid BasicCAD label.

RETURN Statement

Use the RETURN statement to return from a subroutine called by a GOSUB statement.

Syntax:

GOSUB *label*

RETURN

Label can be any valid BasicCAD label.

GOTO Statement

Use the GOTO statement to branch to another statement. This statement is used to transfer program execution to another statement out of the normal sequence of execution.

Syntax:

GOTO *label*

Label can be any valid BasicCAD label.

IF (single line) Statement

Use the IF statement to execute a BasicCAD statement under certain conditions. The logical expression consists of one or more relational expressions separated by AND or OR. A logical expression can also be preceded by NOT.

A relational expression consists of two numeric or string expressions separated by one of the following relational operators: <, <=, =, >=, >, <>.

Syntax:

IF *logexpression* THEN *statement*

Logexpression is a logical expression that can be answered true or false. If the expression is true, then the *statement* is executed; otherwise, it is not.

IF (Group) Statement

Use the IF statement to execute a set of BasicCAD statements under certain conditions. The ELSE section is optional, but the END IF is required.

The logical expression consists of one or more relational expressions separated by AND or OR. A logical expression can also be preceded by NOT.

A relational expression consists of two numeric or string expressions separated by one of the following relational operators: <, <=, =, >=, >, <>.

With the Group IF statement, the IF statement line must end with the word THEN (except for comments). In contrast, the single-statement IF must have the conditionally executed statement following the word THEN on the same line.

Syntax:

```
IF logexpression THEN  
  statements  
.  
.  
{ELSE  
  statements  
.  
}  
END IF
```

Logexpression is a logical expression that can be answered true or false. If the expression is true, then the first set of statements is executed, otherwise the second set is executed.

INPUT Statement

Use the INPUT statement to read a value from the keyboard and assign it to a variable. This statement is used to ask the user for input and assign the entered value to a variable.

If a single string variable is used, the entire string input from the keyboard is assigned to the variable.

If numeric variables are used, the numbers input from the keyboard are assigned to the corresponding variables.

If more than one variable is used with the INPUT statement, any strings input should be enclosed in quotes (" ").

Values read with the INPUT statement can be any valid BasicCAD expressions - they do not have to be simple constants. For example, the user can enter SQRT(2) or 45/2 when a number is requested.

Syntax:

INPUT *string*, *variable list*

INPUT *variable list*

String is an optional message to be displayed for the user. *Variable list* is one or more variables, separated by commas, to which the input values are to be assigned.

INPUT# Statement

Use the INPUT # statement to read a line from a disk file and assign it to a variable. This statement is used to read a line from a disk file and assign it to a numeric variable or to a string variable. The INPUT statement reads an entire line from the disk file.

When this statement is executed, a line from the file is read and a value is assigned to each variable. If the line from the file has fewer values than there are variables, the leftover variables are not modified.

Strings values in the file should be enclosed in quotes if there are multiple values on a line. If an entire line from the file is to be read into a single string variable, use a single string variable with the INPUT # statement.

Values read can be any valid BasicCAD expressions - they do not have to be simple constants.

The file must have been opened using the OPEN statement before the INPUT # statement is executed. The BasicCAD function (*file*) can be used to determine whether the end-of-file has been reached.

Syntax:

INPUT # *file*, *variablelist*

File is the file number that was used in the OPEN statement. *Variablelist* is a set of one or more variables separated by commas. The variables can be string or numeric variables.

LAYER Statement

Use the LAYER statement to turn drawing layers on or off, and to set the current layer. This statement can be used to set a layer to be visible/invisible or editable/uneditable. It can also be used to set the current layer. The following values can be used with the Layer command:

- 0 -- Invisible and not editable
- 2 -- Visible but not editable
- 4 -- Editable but invisible
- 6 -- Visible and editable
- 14 -- Set as current layer

The first *expression* is the layer number, and the second expression must be one of the above values. The LAYER function can be used to get the current status of a layer. See the BasicCAD function descriptions in this manual.

The REGENERATE statement should be used to regenerate the drawing if the visibility of layers has been changed. Otherwise, entities from invisible layers may remain on the screen, and entities from visible layers may not appear on the screen.

Syntax:

LAYER (*expression*) = *expression*

LOCATE Statement

Use the LOCATE to position the cursor in a text window. This statement positions the cursor in the text window that was opened by the WINDOW statement. The next PRINT or INPUT statement will begin at the specified row and column.

This statement has no effect if a text window is not open.

Syntax:

LOCATE *row, column*

See Also: [WINDOW Statement](#)

MESSAGE Statement

Use the MESSAGE statement to output data to the screen. This statement is used to output numeric and/or string expressions to the DesignCAD screen. It is similar to the PRINT statement, except that a dialog box is opened for each message statement, and the program pauses until the user presses the OK button. This command uses the Windows Message Box function.

If more than one line is to be output, chr\$(13) can be included in the output expressions.

Syntax:

MESSAGE *{list of expressions}*

ON ERROR Statement

Use the ON ERROR statement to set up an error handling routine. After the ON ERROR has been executed, any BasicCAD error will cause the execution to be transferred to the specified label. Program execution will continue there until a RESUME statement is encountered.

This statement is used to trap errors in a program. The ERR function can be used to determine the error code.

Syntax:

ON ERROR GOTO *label*

See Also: [RESUME Statement](#)

OPEN Statement

Use the OPEN statement to open a file for input or output for the INPUT #, PRINT #, GET, or PUT statements.

A file must be opened before it is accessed by the INPUT # or PRINT # statements. To open a file for sequential input, use the "I" parameter before the file name. To open a file for sequential output, use "O". (This is the letter O, not the number zero.) You can use "A" to append data to a file - this is like "O," but if the file exists, data will be output to the end of the file.

To open a file for random access (for GET and PUT), use "R." If random access is specified, then the record length must also be specified. This value represents the number of bytes that will be read or written with the GET and PUT statements.

Syntax:

OPEN "A", *filenumber*, *filename*

OPEN "I", *filenumber*, *filename*

OPEN "O", *filenumber*, *filename*

OPEN "R", *filenumber*, *filename*, *recordlength*

Printer output can be specified for the file name in order to output to the printer. However, LPT1 can be used only for output, with OPEN "O".

The *filenumber* can be from one to four. Up to four files can be opened at one time. *Filename* can be any valid DOS file name, including the path.

POINTVAL Statement

Use the POINTVAL statement to assign the coordinates of one of the points that has been set to two variables. The two *variables* will be assigned the X and Y coordinates of a point that has been set in DesignCAD. A third variable can be used for the Z coordinate, for 3-D applications. The *expression* determines which point will be assigned to the variables. The number of points currently set can be determined in the system function SYS(1). An error will occur if *expression* is greater than the number of points set, SYS(1).

Syntax:

POINTVAL *variable variable {variable} expression*

PRECISION Statement

Use the PRECISION statement to set the precision for PRINT statements and for numeric-to-string conversion. This statement determines the number of digits to the right of the decimal point to be used in PRINT statements and in numeric-to-string conversions.

For example, a precision of zero can be used to print or assign only whole numbers. A precision of four can be used to print numbers to the nearest .0001 .

The PRECISION statement affects only the conversion of an expression - it does not affect the value of a numeric variable.

The precision remains the same until it is changed again by the PRECISION statement.

Syntax:

PRECISION *expression*

PRINT Statement

Use the PRINT statement to output data to the screen. This statement is used to output numeric and/or string expressions to the DesignCAD screen. It is identical to the PRINT # statement, except that the data is output to the screen rather than to disk.

Only one line of output can be displayed at a time, unless a text window is open. If more than one line is to be output, the print statements can be separated by ANYKEY statements. This requires the user to press a key before the next line is displayed.

If a text window is open, the PRINT statement will be displayed in the window. The LOCATE and TAB statements can be used to position the output for the window.

The PRINT statement can be terminated with a semicolon (";") to leave the cursor at the end of the line of a text window. The next PRINT statement will begin at that location.

The MESSAGE statement is similar to the PRINT statement, but the MESSAGE statement opens a dialog box for the message, and the user must press the OK button to continue after the message is displayed.

Syntax:

PRINT {*list of expressions*}

See Also: [WINDOW Statement](#)

PRINT# Statement

Use the PRINT # statement to output data to a disk file. This statement is used to output numeric and/or string expressions to a disk file. It is identical to the PRINT statement, except the data is output to disk rather than the screen.

The file must have been opened using the OPEN statement before this statement is executed. Numeric expressions are output in ASCII format. A carriage-return and line-feed are output after each PRINT # statement.

The PRINT # statement can be used to output to a printer by opening the file LPT1 with the OPEN statement. If no expressions are used with a PRINT # statement, then a blank line is output.

Syntax:

PRINT # *file*, {*list of expressions*}

File is the file number that was used in the OPEN statement.

PUT Statement

Use the PUT statement to output a record to a random access file. The record length is specified in the OPEN statement.

When the PUT statement is executed, the string (*stringvar*) will be written to the file at the designated record (*recordnumber*).

If the string to be output is less than the record length, it will be padded with undefined characters. If the string is longer than the record length, it will be truncated.

The MKS\$ and the CVS functions can be used to convert numeric values to and from four-byte strings file input and output.

Syntax:

PUT *file*, *recordnumber*, *stringexpression*

PUTATTR Statement

Use the PUTATTR statement to set the entity type, group status, and layer of an entity. The specified entity will be assigned the specified attributes.

Syntax:

PUTATTR *entity, type, {select, layer, group, color}*

Entity is the entity number - 1 for the first entity in the drawing, 2 for the second, etc.

Type is the entity type, as shown:

1 = Line
2 = Oval
3 = Text
4 = Curve
5 = Arc
11 = Bezier Curve
15 = Attribute
16 = Circle, Circular Arc
17 = Hatch
21 = New Layer
22 = Text Arc
23 = Layer Names
24 = Arrow
26 = Symbol
31 = Plane
32 = Grid
33 = Grid Line
70 = Point Mark
74 = Dimension, Angle
75 = Dimension
76 = Dimension, Diameter/Radius
77 = Dimension, Arc
78 = Dimension, Radius Progressive
79 = Dimension, Progressive
80 = Dimension, Chamfer
81 = Dimension, Coordinate
90 = Bitmap Image

Select is zero if the entity is not to be selected, and one if the entity is to be selected. *Layer* is the layer number to be assigned to the entity. *Group* is the group ID number if the entity is to become part of that Group, or zero if it is not to be part of a Group. *Color* is the color for the entity.

REGEN Statement

Use the REGEN statement to regenerate the drawing on the screen. This statement performs the same function as the DesignCAD Zoom command with a zoom factor of one, and no points set.

Drawing regeneration may be necessary after changing layers with the BasicCAD LAYER statement, or after assigning values to the SYS function. In addition, it is sometimes useful to regenerate the drawing after parts of the drawing have been erased.

Syntax:
REGEN

RESUME Statement

Use the RESUME statement to resume program execution after an ON ERROR statement. This statement is used to continue program execution after an ON ERROR unit has activated by a BasicCAD error.

Syntax:

RESUME *{label}*

Label can be used to specify the location at which program execution will resume. If the label is omitted, the program execution will resume at the statement following the statement that caused the error.

See Also: [ON ERROR Statement](#)

RUN Statement

Use the RUN statement to transfer control to another BasicCAD program. This statement can be used to run another BasicCAD program or a COM or EXE program from within your BasicCAD program.

Running another BasicCAD program with the Run statement

When this statement is used to run another BasicCAD program, all the variables from the current program are cleared when the other program is executed. A string variable or constant can be used for the program name.

The RUN statement causes the new program (*programname*) to replace the current program in memory. To return to the original program, you must execute another RUN or CHAIN statement from the new program. The execution of a program called by the RUN statement always begins at the first line in the called program.

If the file extension of the BasicCAD program is omitted, .BSC will be used. To use an encrypted BasicCAD program, specify the extension .BSX .

The CHAIN statement is similar to the RUN statement, but the CHAIN statement leaves all variables intact when the new program is executed.

Running a COM or EXE program with the Run statement

To run a program from within BasicCAD, just specify the program name with the Run statement. For example,

```
run "edit.com"
```

will run the DOS editor EDIT. The BasicCAD program will continue to run while EDIT.COM is being executed.

With the Run statement, you can run both DOS and Windows applications.

Syntax:

```
RUN programname
```

SETPOINT Statement

Use the SET POINT statement to require the user to set a number of points. This statement displays the message in *string*, and allows the user to set a number of points.

Expression is the number of points to be set. The user can press **Enter** or **Esc** before all the points are set. The system function SYS(1) is the current number of points set - it can be checked to determine if enough points were set.

Syntax:

SETPOINT *string expression*

STOP Statement

Use the STOP statement to terminate the program and return to DesignCAD. The STOP statement can be placed anywhere in the program. A STOP statement at the end of the program is optional. This statement does the same thing as the END statement.

Syntax:
STOP

See Also: [END Statement](#)

TAB Statement

Use the TAB statement to move the cursor in a text window to a certain column. The next PRINT statement will begin at that column.

If the new column is less than the current column, the cursor will move to the next line. The average character width of the current font is used to calculate character columns, since in Windows different characters may have different widths.

Syntax:

TAB *column*

WCLOSE Statement

Use the WCLOSE statement to close the text window or dialog box. The drawing behind the text window will be replaced.

A text window that is opened by the WINDOW command remains on the screen until it is closed or until the program terminates.

Syntax:
WCLOSE

See Also: [WINDOW Statement](#)

WINDOW Statement

Use the WINDOW statement to open a window or dialog box for subsequent PRINT statements. This statement opens a text window, or dialog box on the screen.

As long as the text window is open, all PRINT statements are displayed in the window. The text window remains open until a WCLOSE statement is executed or until the program terminates.

The CLS, LOCATE, and TAB statements can be used to clear the screen and position the cursor in the text window. The COLOR statement can be used to set the text color inside the window.

Only one text window can be opened at a time.

Syntax:

WINDOW *nrows*, *ncols*

Nrows and *ncols* are the number of rows and columns for the text window - they determine the window size.

LABELS Statement

Use the LABELS statement to provide a reference to locations in the program for GOTO statements.

Syntax:

labels:

Labels can be up to seven characters long, may consist of letters and numbers, and must begin with a letter. A BasicCAD statement can optionally follow a label on a line.

SYS Function

The SYS function returns the value of a DesignCAD System Variable. The variable returned is determined by the value of the argument. This function is described below.

The SYS function represents many different DesignCAD system variables. SYS(1), for example, is the number of points set, and SYS(3) is the current layer. A list of the available SYS variables follows.

- 1 - Number of points set
- 3 - Current layer
- 4 - Current line type
- 5 - Line type scale
- 6 - Current line width
- 7 - Current precision
- 9 - Number of entities in the drawing

- 10 - Units of measurement for display - 1=inches, 2.54=cm.
- 11 - Units per inch on output.
- 12 - Default text size
- 13 - Default text angle
- 14 - Display grid type
- 15 - Display grid enable/disable
- 17 - Snap grid on or off
- 19 - Display grid size
- 20 - Snap grid size
- 21 - Attribute display enable/disable
- 22 - Save parameters with drawing, enable/disable
- 23 - Mathematical or geographical angles
- 24 - Fill wide lines enable/disable
- 25 - Sound on/error only/off
- 26 - Manipulate current layer only
- 27 - Status line format
- 29 - Point type
- 30 - Large cursor step size
- 31 - Small cursor step size

- 32 - Drawing unit size
- 34 - Returns 1 if entities are selected, 0 otherwise. If SYS(34) is set to 0, any selected entities are deselected.
- 35 - Number of sides in the rubber band polygon
- 36 - Silent mode (1 for silent, 0 for normal)
- 37 - Cursor step consistent with Screen or Drawing
- 38 - Text mirror disable
- 39 - Point mark type

- 40 - Crosshair enable/disable
- 41 - Rubber band line enable/disable

- 60 - Tick mark segments
- 61 - Tick mark segment division
- 62 - Text horizontal scale

- Functions 90-99 are values for an entity just selected with the Entity statement.
- 90 - Entity type
- 91 - Entity line type

92 - Entity color
93 - Entity layer
94 - Group Number (Solid number in DesignCAD 3-D)
96 - 128 or greater if entity is selected.
97 - Line type scale.
98 - Line thickness
99 - Number of points

101 - Dimension type
103 - Tolerance enable/disable
104 - Arrowhead type
106 - Dimension precision

120 - Minimum X value in the drawing.
121 - Minimum Y value in the drawing.
122 - Maximum X value in the drawing.
123 - Maximum Y value in the drawing.

134 - Printer top margin
135 - Printer bottom margin
136 - Printer left margin
137 - Printer right margin

150 - Angle mode - 0=degrees, 1=grads, 2=radians, 3=degrees-minutes-seconds, 4=bearings.
151 - Distance mode - 1=normal, 2=fractions, 3=feet-inches.
152 - Scale drawings on retrieval and copy option

181 - Number of available BasicCAD symbols.

190 - Handle 1 X value
191 - Handle 1 Y value
192 - Handle 2 X value
193 - Handle 2 Y value

300 - Current drawing color, Red value.
301 - Current drawing color, Green value.
302 - Current drawing color, Blue value.
303 - Hatch pattern color, Red value.
304 - Hatch pattern color, Green value.
305 - Hatch pattern color, Blue value.
306 - Rubber band color, Red value.
307 - Rubber band color, Green value.
308 - Rubber band color, Blue value.
309 - Grid color, Red value.
310 - Grid color, Green value.
311 - Grid color, Blue value.

312 - Point color, Red value.
313 - Point color, Green value.
314 - Point color, Blue value.
315 - Highlight color, Red value.
316 - Highlight color, Green value.
317 - Highlight color, Blue value.
318 - Dimension color, Red value.
319 - Dimension color, Green value.
320 - Dimension color, Blue value.

321 - Background color, Red value.
322 - Background color, Green value.
323 - Background color, Blue value.
324 - Foreground color, Red value.
325 - Foreground color, Green value.
326 - Foreground color, Blue value.
327 - Entity point color, Red value.
328 - Entity point color, Green value.
329 - Entity point color, Green value.
330 - Background color.

401 - Angular dimension prefix
402 - Linear dimension prefix
403 - Radius dimension prefix
404 - Diameter dimension prefix
405 - Chamfer dimension prefix
406 - Coordinate dimension prefix

407 - Angular dimension suffix
408 - Linear dimension suffix
409 - Radius dimension suffix
410 - Diameter dimension suffix
411 - Chamfer dimension suffix
412 - Coordinate dimension suffix

413 - Dimension layer
415 - Linear text location
416 - Linear text position
417 - Angular text position
418 - Linear text direction
419 - Angular text direction
421 - Extended text type
424 - Linear text type
425 - progressive text direction
428 - angular dimension type
433 - Angular dimension precision
437 - Line connecting inward arrows
438 - Dimension text size
439 - dimension text horizontal scale
440 - dimension text slant
441 - dimension tolerance text size
442 - overshoot of Dimension extension lines
443 - gap of Dimension extension lines
444 - length of Dimension extension lines
445 - arrow size
446 - balloon size
447 - Dimension line offset for baseline dimension
450 - Dimension arc prefix
451 - Dimension arc suffix
452 - Radius dimension text position
453 - Dimension text position
454 - Dimension text position
455 - Chamfer dimension text position
456 - Radius dimension text direction
457 - Dimension text direction
458 - Linear dimension text direction

- 459 - Chamfer dimension text direction
- 460 - Dimension arrowhead type
- 462 - Dimension arrowhead scale

To use the SYS function in a BasicCAD program, use it like you would any other function.

The SYS function is different from the other BasicCAD functions in that you can assign a value to a SYS variable. In other words, you can use the SYS function on the left side of the equal sign in an assignment statement.

SYS variables nine (the number of entities in the drawing) and 90-93 (entity characteristics) cannot be modified by assigning a value to the SYS function - they are read-only variables.

When some of the SYS variables are modified, the DesignCAD drawing screen should be regenerated with the REGEN statement or the DesignCAD Zoom command. BasicCAD does NOT check for valid SYS assignments or screen regeneration - if you change the SYS variables, your BasicCAD program is responsible for all validity checking.

The SYS variables 90-93 can be used together with the BasicCAD ENTITY statement to read the entity type, color, line type, and layer of the entity referred to by the last ENTITY statement.

SYS\$ Function

The SYS\$ function returns the value of a DesignCAD System Variable. The variable returned is determined by the value of the argument.

The SYS\$ function represents several different DesignCAD system variables, in string format. SYS\$, for example, is the current path name. A list of the available SYS\$ functions follows:

- 1 - Entity text string. This is the text from a Text or Attribute entity that was "loaded" with an ENTITY statement.
- 2 - Current drawing name.
- 5 - Current drive.
- 6 - Current path name.
- 7 - Current time.
- 8 - Current date.
- 9 - Command line parameter used with DesignCAD.
- 20 - DesignCAD drawing path name
- 21 - DesignCAD symbol path name
- 22 - DesignCAD bitmap path name
- 23 - DesignCAD macro path name
- 24 - DesignCAD BasicCAD path name
- 25 - DesignCAD view path name
- 26 - DesignCAD color path name
- 27 - DesignCAD DXF path name
- 28 - DesignCAD IGES path name
- 29 - DesignCAD HPGL path name
- 30 - DesignCAD XY file path name
- 31 - DesignCAD text path name
- 33 - DesignCAD WPG path name
- 34 - DesignCAD LST path name

To use the SYS\$ function in a BasicCAD program, use it like you would any other string function.

ENCRYPT Utility

ENCRYPT - BasicCAD Program Encryption Utility

If you write a BasicCAD program for distribution, it is possible that you do not want to distribute the source code for your program. Since BasicCAD programs are interpreted and do not have a separate executable format, the ENCRYPT utility has been provided to allow you to distribute your programs without distributing the source code.

BasicCAD programs have a file extension of .BSC, and encrypted BasicCAD programs have a file extension of .BSX. Either form can be read and executed by DesignCAD. To encrypt a program, enter **ENCRYPT *program*** where *program* is the name of your BasicCAD program (with a .BSC extension). An encrypted copy of your program will be created with an extension of .BSX. The encrypted version can be executed, but since it is encrypted it cannot be listed, printed, or edited.

The Encrypt program should be run from the MS-DOS window, since it is a DOS application.

Encrypting a BasicCAD program makes no noticeable degradation in performance, as it does in other CAD languages.

Note: The ENCRYPT program only encrypts programs - it does NOT decrypt, or un-encrypt. **BE SURE TO KEEP A COPY OF YOUR SOURCE CODE!**

Also, remember that NO ENCRYPTION METHOD IS UNBREAKABLE, and this one is no exception. American Small Business Computers makes no guarantee as to the security of the encryption methods used by the ENCRYPT program.

Error Messages

-A-

An error has occurred in your application.
Application Error: WDC caused a General Protection Fault.

-C-

Current layer is always visible!

-E-

Error reading file.
Error writing file.

-F-

File was not found.

-I-

Insufficient memory.
Invisible layer cannot be edited!

-N-

No attribute entities for this drawing.
No entity was selected.
Not a DesignCAD color palette format.
Not a DesignCAD file format.

-P-

Please select entity to be modified first.
Please select line to be modified first.
Printer is out of paper or is not connected to your computer.
Printer out of paper.

-R-

Replace existing file?
Replace existing Macro?

-S-

Save current drawing?

-T-

The filename is not valid.

-U-

Unknown item.

An error has occurred in your application.

A problem has occurred in your application.

To correct this error

If you suspect insufficient memory, free some memory.

Application Error: WDC caused a General Protection Fault.

An application error has occurred.

To correct this error

You must restart DesignCAD for Windows.

Current layer is always visible!

The current layer cannot be set to be "invisible."

To correct this error

Choose another layer to be the current layer.
Disable the Visible box.

Invisible layer cannot be edited!

A layer must be visible in order to be editable.

To correct this error

Choose the layer number.

Enable the Visible box.

Enable the Editable box.

No attribute entities for this drawing.

There must be attribute entities in the drawing for the Material List command to compile a materials list.

To correct this error

Add attributes to the drawing with the Attribute command.

Choose the Material List command.

Error reading file.

An error occurred reading the file.

To correct this error

Check the filename and path.
Open the file again.

Error writing file.

An error occurred writing the file.

To correct this error

Check the filename and path.
Save the file again.

File was not found.

The specified file was not found on the specified drive and/or directory.

To correct this error

Check the drive, directory, and file name of the file.

Insufficient memory.

There is not enough memory to perform the task.

To correct this error

Quit any other applications you are running.
Close any windows you do not need.

No entity was selected.

There were not any entities selected when a command was issued. Some commands cannot be performed unless entities are selected.

To correct this error

Select the entities.

Select the command again.

Not a DesignCAD color palette format.

The file specified in the Load Color Table dialog box was not a color palette format.

To correct this error

Re-check the file extension.

Choose a file with the .PAL extension in the Color Table dialog box.

Not a DesignCAD file format.

The file specified was not a DesignCAD file.

To correct this error

Double-check the drawing file extension.
Choose a file with the .DW2 or .DC2 extension.

Please select entity to be modified first.

An entity was not selected before an editing command was chosen.

To correct this error

Choose the entity to be changed.

Choose the command.

Please select line to be modified first.

A line was not selected before an editing command was chosen.

To correct this error

Choose the line to be changed.

Choose the command.

Printer is out of paper or is not connected to your computer.

DesignCAD is trying to print a drawing and the printer is either out of paper or not connected.

To correct this error

Check the paper tray for the printer.

-or-

Check the printer connection.

Replace existing file?

The specified filename already exists.

To correct this error

Choose Yes to have the file saved with the current filename.

-or-

Choose No to return to the dialog.

Enter a new name for the file.

Replace existing Macro?

The specified macro name already exists.

To correct this error

Choose Yes to have the macro saved with the same macro name.

-or-

Choose No to return to the dialog box.

Enter a new name for the macro.

Save current drawing?

The Close or Exit command have been selected before the drawing has been saved.

To correct this error

Choose Yes to save the drawing.

-or-

Choose No to close the drawing without saving it.

-or-

Choose Cancel to return to the drawing without saving or closing it.

The filename is not valid.

An invalid filename has been entered for a drawing.

To correct this error

Check the filename and extension.

Unknown item.

DesignCAD has discovered an unknown item.

To correct this error

Repeat the command.

Printer out of paper.

DesignCAD is trying to print a drawing and the printer is either out of paper.

To correct this error

Check the paper tray for the printer.

Line Types

There are 13 line types available with DesignCAD. These are:

Solid

Dashed

Hidden

Center

Phantom

Dotted

Dashdot

Border

Divide

Custom-1

Custom-2

Custom-3

Custom-4



See Also: [Changing Line Characteristics](#)

Showing and Hiding Information

With DesignCAD, you can display information such as Toolboxes, the Command Line, and an Info Box. This information can also be "hidden" or removed temporarily from the screen so that the drawing area does not become cluttered.

[Showing Main Toolbox](#)

[Showing Snap Toolbox](#)

[Showing Status Bar](#)

[Showing Symbol Toolbox](#)

[Showing Coordinate Bar](#)

[Showing Rulers](#)

[Showing Info Box](#)

[Hiding Main Toolbox](#)

[Hiding Snap Toolbox](#)

[Hiding Status Bar](#)

[Hiding Symbol Toolbox](#)

[Hiding Coordinate Bar](#)

[Hiding Rulers](#)

[Hiding Info Box](#)

Position of Dimension Arrows

Dimension arrows can be drawn:



Inside the dimension extension lines



Outside the dimension extension lines

Setting the position of dimension arrows

After a Dimension command is selected, choose the [Dimension Line button](#) in the Command Line.

Choose the arrow button next to the Position box to choose the arrow position.

Choose the Same as button to have the dimension drawn in the same position as other dimensions.

Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set the points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Arrowhead Type

The dimension arrowhead type is chosen with the Dimension Line button in the Command Line after a Dimension command is selected. You can change any dimension arrowhead by selecting the dimension, then selecting the [Show Info Box option](#). Next, choose the new arrowhead type in the arrowhead type box.

There are 12 dimension arrowhead types available. These are:

None
Normal
Slash
Filled Circle
Filled (Short)
Filled (Long)
Hollow Circle
Hollow (Short)
Hollow (Long)
Normal (60°)
Filled (60°)
Hollow (60°)



Setting the arrowhead type

After a Dimension command is selected, choose the [Dimension Line button](#) in the Command Line. Choose the arrow button next to the Arrowhead Type box to choose the arrowhead type. Choose the Same as button to have the arrowhead drawn with the same type as other dimensions. Then set a point on the existing dimension in the drawing. Choose the Default button to reset the dimension options. Set the points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Arrowhead Size

The dimension arrowhead size is set in Drawing Units in the Size box of the Dimension Line button.

Setting the arrowhead size

After the Dimension command is selected, choose the Dimension Line button in the Command Line.

Choose the Arrowhead Size box.

Type the size of the arrowhead in Drawing Units.

Choose the Same as button to have the arrowhead drawn in same size as other dimensions. Then set a point on the existing dimension in the drawing.

Set the points for the dimension.

See Also: Dimension Options, Changing Dimension Options

Overshoot Length

The overshoot length is set in the Overshoot box with the [Extension Line button](#). This determines the distance the extension line extends past the dimension line.

Setting the overshoot length

After a Dimension command is selected, choose the Extension Line button in the Command Line.

Choose the Overshoot box.

Type the [Overshoot](#) length.

Choose the Same as button to have the overshoot drawn with the same length as other dimensions.

Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set the points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Fixed Length

When the Fixed Length button is selected with the [Extension Line button](#), you can set a distance for the fixed length. This determines the distance the extension line extends from the dimension line. The extension line is always drawn a certain distance from the dimension line.

Setting a fixed length

After a Dimension command is selected, choose the Extension Line button in the Command Line.

Choose the Fixed Length check box.

Type the Fixed Length in Drawing Units.

Choose the Same as button to have the extension line drawn with the same fixed length as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set the points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Variable Length

When the Variable Length button is selected with the [Extension Line button](#), you can set a distance for the [gap size](#). This determines the distance between the extension line and the object being dimensioned.

Setting a variable length

After a Dimension command is selected, choose the Extension Line button in the Command Line.

Choose the Variable Length check box.

Type the Gap Size in Drawing Units.

Choose the Same as button to have the extension line drawn with the same gap size as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Text Size

The dimension text size is entered in Drawing Units in the Size box of the Dimension Text button.

Setting the dimension text size

After a Dimension command is selected, choose the Dimension Text button in the Command Line.

Choose the Text Size box.

Choose the Same as button to have the dimension text drawn the same size as other dimensions.

Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: Dimension Options, Changing Dimension Options

Dimension Prefix

The dimension prefix is entered in the Prefix box of the [Dimension Prefix, Suffix button](#).

Setting the dimension prefix

After a Dimension command is selected, choose the Dimension Prefix, Suffix button in the Command Line.

Choose the Prefix box.

Type a new prefix, or choose one from the box.

Choose the Same as button to have the dimension text drawn with the same prefix as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Suffix

The dimension suffix is entered in the Suffix box of the [Dimension Prefix, Suffix button](#).

Setting the dimension suffix

After a Dimension command is selected, choose the Dimension Prefix, Suffix button in the Command Line.

Choose the Suffix box.

Type a new suffix, or choose one from the box.

Choose the Same as button to have the dimension text drawn with the same suffix as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Postive or Negative Dimension Tolerance

With DesignCAD you can display positive or negative tolerance numbers with each dimension. This option is selected with the Dimension Tolerance button.

Setting the positive or negative dimension tolerance

After a Dimension command is selected, choose the Dimension Tolerance button in the Command Line.

Choose the Positive or Negative Tolerance box.

Type the tolerance measurement.

Choose the Same as button to have the dimension drawn with the same tolerance as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Postive and Negative Dimension Tolerance

With DesignCAD you can display both positive and negative tolerance numbers with each dimension. This option is selected with the Dimension Tolerance button.

Setting the positive and negative dimension tolerance

After a Dimension command is selected, choose the Dimension Tolerance button in the Command Line.

Choose the Positive and Negative Tolerance box.

Type the positive tolerance measurement in the first box.

Type the negative tolerance measurement in the second box.

Choose the Same as button to have the dimension drawn with the same tolerance as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

No Dimension Tolerance

With DesignCAD you can choose not to display any tolerance numbers with a dimension. This option is selected with the Dimension Tolerance button.

Setting no dimension tolerance

After a Dimension command is selected, choose the Dimension Tolerance button in the Command Line.

Choose the None box.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Saving a Vector Dimension

The Save as Vector form box in the Command Line saves the dimension as a vector entity.

Saving a vector dimension

After a Dimension command is selected, choose the Save as vector form box.

See Also: Dimension Options, Changing Dimension Options

Symbol File

The symbol file is a .DW2 file containing information about a symbol. If the symbol file is modified, the change will be reflected in all the drawings using that symbol. This file must be present when the symbol is loaded. The symbol file is loaded when a symbol is retrieved with the Symbol Load command.

Entering Coordinates for Points

You can enter the exact coordinates of a point using several different DesignCAD commands.

The Coordinate Bar at the top of the screen displays the X, Y, Delta X, and Delta Y coordinates of the cursor. The Coordinate Bar can also be used to enter coordinates.

Setting X and Y coordinates

Choose the Point XY command in the Point menu.
Type the X coordinate for the cursor in the X coordinate box.
Type the Y coordinate for the cursor in the Y coordinate box.
Choose the OK button.

Setting relative X and Y coordinates

Choose the Point Relative command in the Point menu.
Set a point for the reference point.
-or-
Press **Enter** to use the last point as the reference point.
Type the horizontal distance from the current cursor location in the Delta X coordinate box.
Type the vertical distance from the current cursor location in the Delta Y coordinate box.
Choose the OK button.

Setting polar X and Y coordinates

Choose the Point Polar command in the Point menu.
Set a point for the reference point.
-or-
Press **Enter** to use the last point as the reference point.
Type the distance from the current cursor location in the Distance coordinate box.
Type the angle from the current cursor location in the Angle box.
Choose the OK button.

See Also: [Setting Points](#), [Choosing Point Commands](#)

Minimizing Windows

Clicking the Minimize button at the right end of the title bar in an application window shrinks the application window to an icon. This is the same as choosing Minimize from the application Control-menu.

These icons can be moved around the screen with the Arrange Icons command. This command straightens any icons on the screen and places them along the bottom of the screen.

See Also: Maximize button, Windows

Moving Points

Activate the Point Select Mode.
Select the entity to be changed.
Set a point on the point to be moved.
Set a point for the new location of the first point.

See Also: [Point Move command](#), [Deleting Points](#)

Undoing an Action

Canceling an action that is in progress
Press **Esc**.

See Also: [Undoing Commands](#)

Canceling a Menu

Click outside the menu.

-or-

Press **Esc** to cancel the menu.

To return to your drawing, press **Esc** again.

See Also: [Closing a Menu](#)

Erasing the Screen

Choose the New command from the File menu.

When asked if you want to save the drawing, choose the Yes button to save the drawing.

Choose the No button to clear the screen without saving the drawing.

See Also: [Clearing the Screen](#)

Using the Mouse

[Clicking the Mouse](#)

[Dragging the Mouse](#)

[Selecting Commands with a Mouse](#)

[Selecting Entities with the Mouse](#)

[Mouse Techniques](#)

Using Menus

The Command Menu lists all DesignCAD commands. You can choose commands from the menus using the mouse, keyboard, or digitizer.

Selecting Commands with a Mouse

Selecting Commands with the Keyboard

Selecting Commands with a Digitizer

If an ellipsis (...) follows a command name, a dialog box appears so you can set the options for the command.

You can close a menu without choosing a command.

Canceling a menu

Click outside the menu.

-or-

Press **Esc** to cancel the menu.

To return to your drawing, press **Esc** again.

Editing Layers

[Deleting a Layer](#)

[Saving Layers](#)

[Selecting Layers](#)

Getting Color Table

Choose the Load Color Table command.
Type the name of the color table you want.
Choose the OK button.

See Also: Set Color command, Load Color Table command, Loading Color Table

Loading Files

[Loading Color Table](#)

[Loading Symbols](#)

[Loading and Selecting Objects](#)

[Merging Drawings](#)

[Importing Drawings](#)

Text Arc command

The Text Arc command draws text in an arc.

Text

The text is entered in the Text box in the Command Line.

Size

The size of the text is entered in the Size box in the Command Line.

Font

Use the Font box in the Command Line to change the font.

Angle

The angle of the text is entered in the Angle box in the Command Line. The angle of the text is also determined by the location of the points set with the Text command.

Orientation

Text can be left-justified, centered, or right-justified by selecting the orientation in the Orientation box in the Command Line.

Bold

Choose this button to have the text drawn in bold.

Italic

Choose this button to have the text drawn in italic.

Same as

Text can be drawn with the same options as other text in the drawing by selecting the Same as button in the Command Line.

After all the options are set, points are set for the lower-left, center, and lower-right points of the text string.

Shortcut



Mouse:

See Also: [Drawing Text in an Arc](#)

Backward Arc

A backward arc is drawn with earlier versions of DesignCAD.

Toolbox

A Toolbox contains icons that issue DesignCAD commands. These icons can quickly be selected with the mouse, digitizer, or keyboard.

Area command

Use the Area command to calculate the area of part of the drawing.

Dialog box options

Show Result in Acres

The area of an entity can be calculated in acres when the Show Result in Acres box is selected in the dialog box.

Precision

The Precision box determines the number of decimal places the measurement will display.

Select Lines

When the Select Lines box is selected in the Command Line, lines can be selected in the drawing.

See Also: [Adding Area to Drawing](#), [Calculating Area](#)

Calculating Area

Calculating the area

Choose Area from the Dimension menu.
Set points around the area to be measured.
Press **Enter**.

Calculating the area in acres

Choose Area from the Dimension menu.
Set points around the area to be measured.
Choose the Acres box to have the area measured in acres.
Press **Enter**.

Calculating the area of an entity

Choose Area from the Dimension menu.
Choose the Select Lines box in the Command Line.
Set a point on the entity to be measured.
Press **Enter**.

See Also: [Adding Area Drawing](#), [Area command](#)

Info Box

The Info Box appears when you double-click an entity or choose the Show Info Box option from the View menu.

The Info Box displays information such as color, line type, angle, height, text style and font for the selected entity. This information can be changed by choosing the appropriate box in the Info Box.

The information displayed in the Info Box depends on the type of entity selected.

Displaying Attributes

Choose the Show Attribute option in the Options menu.

-or-

Choose the Attribute command from the Dimension menu.

Choose the Show attribute box in the Command Line.

See Also: [Show Attribute option](#)

Bookmark

A bookmark is a marked location in the Help file that you have named for reference purposes.

BasicCAD

BasicCAD is a powerful programming language that comes with DesignCAD 2-D for Windows. With BasicCAD, you can write a program and execute it as you would any other DesignCAD command.

BasicCAD is similar to standard Quick Basic. It provides the capability to execute DesignCAD commands as well as Basic statements. This gives BasicCAD the flexibility of Basic with the power of DesignCAD.

Command Prompt

The command prompt, or DOS prompt, appears when you are in MS-DOS.

Your prompt probably looks like this:

C:\>

You must enter **WIN** at this prompt to start Windows.

Digitizer

A digitizer, or graphics tablet, is used to move the cursor, set points, and enter commands. With a graphics tablet, you can "trace" a drawing from paper onto the screen.

Note

A note, or annotation, is a comment added to the help file. Annotations appear in a separate pane.

Annotation

A annotation, or note, is a comment added to the help file. Annotations appear in a separate pane.

Snap Grid

The snap grid is an imaginary grid placed over your drawing. Each point set with the Point Set command is located exactly on one of the grid points. The cursor movement follows the snap grid.

Regenerate

To regenerate means to quickly refresh the screen.

The Regenerate command is a quick way to refresh the screen and repair any lines that look as if they have been partially erased.

Resolution

The resolution of a drawing determines the total number of dots per inch printed with each drawing.

With DesignCAD, you can print drawings with different resolutions. Depending on your printer, the printer setup dialog box allows you to choose between 300 d.p.i., 150 dpi, or 75 dpi.

Text Style

Text style refers to how the text looks in the drawing. By choosing the appropriate buttons in the Text command or Info Box, text can be drawn bold or italic.

Zoom Static

Zoom static permanently changes the "zoom," or size of the drawing.

Mirror command

The Mirror command creates a mirrored image of selected objects. Points are set for the location and orientation of an imaginary mirror. The mirror can be drawn horizontally, vertically, at a custom angle, or "normal" to a line perpendicular to the mirror.

Mirror Vertically

Choose this button in the Command Line to have the object mirrored vertically.

Mirror Horizontally

Choose this button in the Command Line to have the object mirrored horizontally.

Custom Mirror

Choose this button in the Command Line to have the object mirrored based on a mirror defined by two points.

Custom Normal

Choose this button in the Command Line to have the object mirrored based on an imaginary line that is perpendicular to the mirror.

Make Copy

Choose this box to have a copy of the object drawn when the it is mirrored.

Shortcut



Mouse:

See Also: [Mirroring Objects](#)

Align Drawing command

The Align Drawing command changes the size and location of the drawing based on four points. The first point set with the Align Drawing command is moved to the location of the second point. The third point is moved to the location of the fourth point. The Align Drawing command scales and rotates the drawing so that the first and third points fit into the second and fourth points. This command changes the actual size and position of the drawing.

See Also: [Aligning a Drawing](#)

View Read command

The View Read command reads the view configuration from a disk file that was previously saved with the View Save command.

DesignCAD can read view files in either binary or ASCII format. Binary view files are saved with the extension ***.B2D**. ASCII view files are saved with the extension ***.A2D**.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Opening Drawing Views](#)

View Save command

The View Save command saves the current view configuration to disk so that it can be read with the View Read command.

DesignCAD allows you to open several views of a drawing, edit the objects, and save the view configurations as a single file. You can then open the view configurations with DesignCAD.

DesignCAD can save view files in either binary or ASCII format. Binary view files are saved with the extension ***.B2D**. ASCII view files are saved with the extension ***.A2D**.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Saving Drawing Views](#)

Point Line Distance command

The Point Line Distance command sets the distance for the [Point Line Snap command](#).

Original

The Original box shows the original distance.

New

Enter the new distance for the Point Line Snap command in the New box.

See Also: [Setting Points Along a Line](#)

Point Line Snap command

The Point Line Snap command sets a point a certain distance from a point on a line in the drawing. The first point is erased. If the line is not straight, the distance along the line will be measured. If the line bends between the first point and the new point, the straight-line distance between the points will be shorter than the distance specified.

The distance is set with the [Point Line Distance command](#).

Shortcut



Mouse:

See Also: [Setting Points Along a Line](#)

Run command

The Run command executes programs created with the BasicCAD programming language.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Running BasicCAD Programs](#)

Preset Point Mode command

In DesignCAD, points can be set before or after the command is selected. Normally, the command is selected and then the points are set. However, the Preset Point Mode allows you to set the points, then select the command.

Shortcuts



Mouse:

Keys: **Shift+S**

See Also: [Setting Points First](#)

Save Bitmap command

The Save Bitmap command saves a bitmap to disk. This command saves screen images in .BMP file format. These bitmaps are compatible with many Paint and Desktop Publishing packages.

When a bitmap is saved, only the colors on the screen are saved. The drawing itself is not saved.

Two points can be set for the lower left and upper right corners of the drawing. If no points are set, any toolboxes displayed on the screen will be saved with the bitmap.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters. DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Saving Bitmaps](#), [Reading Bitmaps](#)

Load Bitmap command

The Load Bitmap command reads a bitmap from disk and displays it on the screen. This command reads bitmaps generated with the Save Bitmap command and most .BMP format files. The bitmaps being read must be compatible with the graphics adapter being used.

Dialog Box Options

File Name

Type a new filename to save a drawing with a different name or in a different file format.

Note: You cannot save a drawing in a different file format with the same name. Use the current name, or select a name in the list to save a drawing with an existing filename.

Note: A filename can contain up to eight characters and an extension of up to three characters.

DesignCAD adds the extension you specify in the Save File As Type box.

Drives

Select the drive in which you want to store the drawing.

Directories

Select the directory in which you want to store the drawing.

Save File As Type

DesignCAD converts drawings to many other file formats. What appears in the list depends on which converters you install with DesignCAD.

See Also: [Saving Bitmaps](#), [Reading Bitmaps](#)

Bitmaps

A bitmap, or screen image, is a copy of the screen created with the Save Bitmap command and imported with the Load Bitmap command command.

A bitmap is only for viewing purposes and cannot be edited or manipulated as a regular drawing.

It is possible to "trace over" a bitmap with DesignCAD drawing commands.

The bitmap being read must be compatible with the graphics adapter being used.

Vector Conversion command

The Vector Conversion command converts DesignCAD entities to vector entities.

See Also: [Converting Entities](#)

Converting Entities

Select the entity to be converted.
Choose Vector Conversion from the Edit menu.

See Also: [Vector Conversion command](#)

Material List command

The Material List command counts all the attributes in a drawing and displays a list.

Attributes are drawn with the [Attribute command](#).

The list can be saved to a disk file so that it can be used with other software, such as database systems.

See Also: [Drawing Attributes](#), [Listing Attributes](#)

Show Rulers option

Use the Show Rulers option displays the rulers at the top and left edge of the drawing area.

The vertical and horizontal rulers help you align entities and keep track of where you are in the drawing.

Shortcut

Keys: **Shift+Alt+R**

See Also: [Showing Rulers](#), [Hiding Rulers](#)

Showing Rulers

Choose the Rulers option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Rulers](#), [Customizing DesignCAD](#)

Hiding Rulers

Choose the Rulers option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Rulers](#), [Customizing DesignCAD](#)

Zoom Window command

Use the Zoom Window command to zoom into an area in the drawing based on two points set for a zoom window.

Shortcut



Mouse:

See Also: [Zooming a Drawing](#)

Calculator command

Use the Calculator command to perform calculations on the screen.

The expression entered in the Calculator command can contain a formula or expression.

Dialog box options

Text

The expression is entered in this box.

Compute

Choose this button to have the mathematical expression calculated.

See Also: [Calculating Information](#)

Calculating Information

Choose Calculator from the Info secondary menu in the Dimension menu.
Type the calculation.
Choose the Compute button.
Choose the OFF button to end the command

See Also: [Inserting calculations](#), [Calculator command](#)

Inserting Calculations into Drawing

In the Calculator dialog box, highlight the text.
Press **Ctrl+C** to copy the text to the Clipboard.
Choose the OK button.
Choose the Text command.
Choose the Text box in the Command Line.
Press **Ctrl+V** to paste the Clipboard contents.
Set the points for the text.
The calculation will be inserted into the drawing.

See Also: [Calculating information](#), [Calculator command](#)

Creating User-Defined Macros

DesignCAD allows you to assign the macro buttons in the Command Line to macros. These macros can be selected by choosing the macro button or by pressing the keystroke associated with that button.

Assigning Buttons to Macros

Choose the Options command.

Choose the Macro category.

Use the options under Modify Macro Settings to define the macro buttons.

Button to Change Select the buttons, icons, and macros for the macro buttons.

Which Select the macro button to change.

Icon Select the icon to use for the defined macro button.

Available Macros Select the macro to define for the button from the list of available macros.

Keystroke Shows the keystroke used for the defined macro.

Change Select this button to change the button definitions.

Startup macro Select the macro file to be used as the startup macro.

Choose the OK button.

See Also: [Creating Macros](#)

Digitizer Menu Create command

Use the Digitizer Menu Create command to create a digitizer menu. The menu created with this command does not appear on the screen. The menu contains several selection boxes. These selection boxes are spaces where commands are selected.

See Also: [Creating a Digitizer Menu](#)

Digitizer Menu Load command

Use the Digitizer Menu Load command to load the "data" portion of the digitizer menu. The "paper" portion of the menu should be attached to the digitizer before the digitizer is loaded.

A maximum of 10 digitizer menus can be loaded at once.

Note: You cannot draw inside the area containing the digitizer menu, unless you disable the digitizer menu first.

See Also: [Loading a Digitizer Menu](#)

Digitizer Menu Disable All command

Use the Digitizer Menu Disable All command to disable all active digitizers. So that the entire digitizer drawing area can be used.

See Also: [Disabling Digitizer Menus](#)

Digitizer Menu Enable All command

Use the Digitizer Menu Enable All command to enable all disabled menus.

See Also: [Enabling Digitizer Menus](#)

Digitizer Menu Remove All command

Use the Digitizer Menu Remove All command to remove all active and inactive digitizer menus so that the entire digitizer drawing area can be used.

See Also: [Removing Digitizer Menus](#)

Symbol Toolbox

A Symbol Toolbox is a Toolbox containing frequently-used symbols. A Symbol Toolbox is created within the Options command.

Show Symbol Toolbox option

Use the Show Symbol Toolbox command to show or hide the [Symbol Toolbox](#).

Shortcut:

Key **Shift+Alt+Y**

See Also: [Showing Symbol Toolbox](#), [Hiding Symbol Toolbox](#)

Hiding Symbol Toolbox

Choose the Symbol Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Symbol Toolbox](#), [Customizing DesignCAD](#)

Showing Symbol Toolbox

Choose the Symbol Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Hiding Symbol Toolbox](#), [Customizing DesignCAD](#)

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Loading a Digitizer Menu

Attach the paper template onto the digitizer.

Select the Digitizer Menu Load command from the Digitizer Menu secondary menu in the File menu.

Choose the digitizer menu name in the File Name box.

Choose the OK button.

Set a point in the lower left corner of the digitizer menu.

A rubber band box shows how the digitizer menu will be loaded.

Set a point in the upper right corner of the digitizer menu.

Note: You cannot draw inside the area containing the digitizer menu, unless you disable the digitizer menu first.

See Also: [Digitizer Menu Load command](#)

Creating a Digitizer Menu

To create the digitizer template using DesignCAD commands, remember these tips:

Draw a box around the menu and around each command box. Both the menu and the command boxes must be rectangular in shape.

Inside each selection box, draw text or a picture to be associated with that command box.

Print this digitizer menu to scale so that it may be placed on the digitizer.

Load the file containing the digitizer template.

Choose the Digitizer Menu Create command from the Digitizer Menu secondary menu in the File menu.

Enter the name of the digitizer in the File Name box.

Choose the OK button.

Set a point in the lower left corner of the digitizer template.

A rubber band box shows how the menu is drawn.

Set a point in the upper right corner of the digitizer template.

Set a point in the lower left corner of the command box.

A rubber band box shows how the command box is drawn.

Set a point in the upper right corner of the command box.

Enter the command name in the Command box.

-or-

Choose the Use Keystroke box.

Choose the Press a Key Now button.

Press the shortcut key for the command.

-then-

Choose the OK button.

Set two more points for the next command box.

-or-

Press **Enter** or click the middle mouse button to end the command.

Note: Use the DCMACRO.SYS file to find the correct command names to enter in the Command box.

See Also: [Digitizer Menu Create command](#)

Disabling a Digitizer Menu

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Disabling All Digitizer Menus

After a digitizer menu has been loaded, choose the Digitizer Menu Disable All command from the Digitizer Menu secondary menu in the File Menu.

See Also: [Digitizer Menu Disable All command](#)

Disabling One Digitizer Menu

Choose the Digitizer Menu category in the Options command.

Choose the Disable button next to the digitizer menu to disable the digitizer menu.

See Also: [Enabling One Digitizer Menu](#)

Enabling Digitizer Menus

[Enabling All Digitizer Menus](#)

[Enabling One Digitizer Menu](#)

Enabling All Digitizer Menus

After a digitizer menu has been disabled, choose the Digitizer Menu Enable All command from the Digitizer Menu secondary menu in the File menu.

See Also: [Digitizer Menu Enable All command](#)

Enabling One Digitizer Menu

Choose the Digitizer Menu category in the Options command.

Choose the Enable button next to the digitizer menu to enable the digitizer menu.

See Also: [Disabling One Digitizer Menu](#)

Removing Digitizer Menus

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Removing All Digitizer Menus

After a digitizer menu has been loaded, choose the Digitizer Menu Remove All command from the Digitizer Menu secondary menu in the File menu.

See Also: [Digitizer Menu Remove All command](#)

Removing One Digitizer Menu

Choose the Digitizer Menu category in the Options command.

Clear the check mark next to the digitizer menu name to remove the digitizer menu.

See Also: [Loading a Digitizer Menu](#)

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Selecting Command First

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Using Symbol Toolbox

A Symbol ToolboxSymbolToolbox can be created using the Options command. This Toolbox can contain any DesignCAD symbols you have loaded. These symbols can then be loaded into a drawing by simply choosing them from the Toolbox.

Using Symbol Toolbox

Choose the symbol in the Symbol Toolbox.

Set a point for the primary handle.

Set a point for the secondary handle.

The symbol will be loaded according to the two points set.

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Choose Options from the Options menu.

Choose the Symbol Toolbox category.

Type the correct path for the drive and directory of your symbol files in the Directory box.

Choose the symbol you want to add in the Available Symbols box.

Choose the Add button.

The symbol will be added to the Current Content box.

Choose the OK button.

Note: Use the Delete button to delete symbols from the Symbol Toolbox.

See Also: [Using Symbol Toolbox](#)

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Saving the Configuration

DesignCAD automatically saves the system configuration for each drawing to disk when you save the drawing. When a configuration is saved, it becomes the default configuration for DesignCAD. This configuration is loaded whenever DesignCAD is run.

The configuration information includes DesignCAD options and view information. Information such as the Drawing Unit size, precision, text size and font, and dimension options are all saved with the drawing.

This option can be disabled.

See Also: [Customizing the Screen](#)

DOS DesignCAD Compatible Shortcut Keys

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DOS Keystroke Commands

Using Keystroke Commands

DesignCAD Windows allows you to use shortcut keys common to Windows applications or DOS applications.

The DOS Keystrokes option allows you to use DOS keystrokes instead.

Enabling DOS keystrokes

Choose the DOS Keystrokes option from the Options menu.

See Also: [Customizing DesignCAD](#), [DOS Keystrokes command](#), [DOS Keystroke Commands](#)

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Using a Mouse

The mouse controls a pointer on the screen. You move the pointer by sliding the mouse over a flat surface in the direction you want the pointer to move. You don't press the mouse button when you move the mouse. If you run out of room to move the mouse, lift it and then put it down. The pointer doesn't move while the mouse is in the air.

With DesignCAD, a mouse moves the cursor, or pointer, sets points, draws lines, and chooses commands from the Command Menu. In addition, a mouse can choose commands from the Main Toolbox, the Snap Toolbox, or a Hot Toolbox.

A mouse usually contains two or three buttons. The buttons on the mouse are defined as:

Left Button:	Point Set command
Right Button:	Gravity command
Middle Button:	Enter key

"Pointing" the mouse

Moving the mouse to place the pointer over an item is called pointing.

"Clicking" the mouse

Pointing to an item on your screen and then quickly pressing and releasing the mouse button is called clicking. You choose entities on the screen by clicking. Double-clicking, pointing to an item and quickly pressing the mouse button twice, is a convenient shortcut for many of the tasks you will perform in DesignCAD.

"Dragging" the mouse

DesignCAD allows you to move and copy commands by choosing entities on the screen and "dragging" the mouse. To do this, select the entities. Then hold down the left mouse button and move the mouse so that the highlighting box is moved across the. Once the highlighting box is in the location you want, release the left mouse button.

Moving the mouse

The mouse can be forced to move horizontally by pressing **Ctrl+Alt**. Pressing **Shift+Alt** forces the mouse to be moved vertically.

Special Mouse Operations

The mouse may have two or three buttons. In DesignCAD, these buttons are used to manipulate and edit objects.

In Point Select Mode:

Right Mouse Button

Clicking - Adds a vertex, or corner, to a line.

Double-Clicking - Adds point to a line.

Left Mouse Button

Clicking - Snaps to nearest point.

Double-Clicking - Deletes nearest point.

In Select Mode:

Right Mouse Button

Clicking - Selects entity.

Double-Clicking - Opens Info Box for selected object.

Left Mouse Button

Clicking - Snaps to nearest point.

Double-Clicking - Snaps to nearest point and enables selection box for scaling.

See Also: [Mouse Shortcuts](#), [Selecting Commands with the Mouse](#), [Mouse Techniques](#)

Using the Keyboard

The keyboard can be used to issue most DesignCAD commands. Points are set, the cursor, or pointer, is moved, and options are entered from the keyboard.

Keys are often used in combinations or sequences. For example, **Ctrl+F1** keystroke means to hold down the **Control** key while pressing **F1**.

Shortcut Keys can be invoked from the keyboard by pressing a key or set of keys.

See Also: [Controlling the Cursor](#), [Selecting Commands with the Keyboard](#), [Selecting Objects with the Keyboard](#)

Selecting Commands

To choose a command from a menu, you can use the mouse or press a shortcut key. In DesignCAD, shortcut keys are listed after the command names in the Command Menu.

[Selecting Commands with a Mouse](#)

[Selecting Commands with the Keyboard](#)

[Selecting Commands with a Digitizer](#)

See Also: [Commands](#), [Shortcut Keys](#)

Opening an Existing Drawing

Before you can work on an existing drawing, it must be open on the screen.

Opening an existing drawing

Choose Open from the File menu. DesignCAD displays the Open dialog box.

In the File Name box, type the filename of the drawing you want to open.

-or-

Double-click the name in the File Name box or use the arrows in the scroll bar to choose the filename. Choose the OK button.

If the drawing you want is not listed in the File Name box, choose a new drive or directory in the Open dialog box.

Choosing the Set Handles box in the Open dialog box allows you to set handles for the drawing. If the drawing was saved with handles, you can set up to two points for the location of the pre-defined handles. If the drawing was not saved with handles, you can set up to four points for the location of the lower left, lower right, upper left, and upper right corners respectively.

The File menu lists the four most-recently opened drawings.

Opening a recent drawing

Choose the file from the File menu.

Tip: You can see a list of all the files with a specific extension in a drive or directory by typing an asterisk (*), a period (.), and the extension in the File Name box.

See Also: [Opening DesignCAD Files](#), [Opening a Drawing](#), [Opening Drawings](#), [Opening a File Created by Another Application](#)

Setting Drawing Handles

DesignCAD allows you to set drawing handles when you save or open a drawing. Up to two points can be set on the screen to determine the lower left and lower right corners of the drawing.

If the Set Handles command is not selected, and no points are set with the Open command, the drawing handles will be set at the lower left and lower right corners of the drawing.

Saving a drawing with handles

Choose the Set Handles command from the Options menu.

Set up to two points for the drawing handles.

Press **Enter** if only one point is set.

Choose the Save command from the File menu.

Type the filename from the File Name box.

Choose the OK button.

Opening a drawing with handles

Choose the Open command from the File menu.

Select the filename from the File Name box.

Choose the Set Handles box.

Choose the OK button.

Set up to two points for the drawing handles.

Press **Enter** if only one point is set.

See Also: [Copying Objects with Handles](#), [Moving Objects with Handles](#)

Speeding Up Basic Tasks

If you have a mouse, you can use the commands in the Toolboxes to perform many basic tasks quickly, such as drawing a line, setting a point, drawing a circle, and drawing polygons. For example, you can quickly choose the command to draw a polygon by clicking the polygon button.

If you forget which button accomplishes a particular task, you can view a brief description in the Status Bar at the lower-left corner of the screen. Point to a command in the Main Toolbox and hold down the left mouse button. To avoid activating the button, continue to press the mouse button. Drag the mouse pointer away from the button on the Main Toolbox, then release the mouse button.

See Also: [Shortcut Keys](#), [Mouse Shortcuts](#)

Saving Your Drawings

When you are working in DesignCAD, you are actually working on a copy of your drawing temporarily stored in the computer's memory. To save your work for future use, you must give the drawing a name and store it on a disk, either a hard disk or a floppy disk.

Saving a drawing

Choose the Save command from the File menu. If you are saving the drawing for the first time, DesignCAD displays the Save As dialog box.

DesignCAD saves the drawing in the current directory and automatically adds a .DW2 extension to the filename unless you specify another extension.

Choose the OK button.

You can choose a new drive or directory in the Save dialog box to save the drawing in a different drive or directory.

See Also: [Saving a Drawing](#), [Saving Drawings](#)

Closing a Drawing

If a drawing has changed, DesignCAD asks if you want to save the changes before closing the drawing or quitting DesignCAD. If you choose the "Yes" button, but haven't named the drawing, DesignCAD displays the Save As dialog box.

Closing a drawing

Double-click the drawing Control-menu box in the upper-left corner of the drawing window.

-or-

Choose the Close command from the File menu.

See Also: [Clearing the Screen](#)

Choosing Commands

You can choose commands from the menus or press the shortcut keys assigned to the most commonly used commands. The Command Menu lists the shortcut key to the right of the command name.

If an ellipsis (...) follows a command name, a dialog box appears so you can set the options for the command.

You can close a menu without choosing a command.

Canceling a menu

Click outside the menu.

-or-

Press **Esc** to cancel the menu.

To return to your drawing, press **Esc** again.

See Also: Selecting Commands with a Mouse, Selecting Commands with the Keyboard, Selecting Commands with a Digitizer

Selecting Options

The OK, Cancel, and Close command buttons appear in many dialog boxes. You can choose one of these buttons when you finish setting options in a dialog box.

OK button

The OK button closes the dialog box and completes the command using the selected options. For example, choosing the OK button in the Open dialog box opens the drawing. To choose the OK button, you can either click the button or press **Enter**.

Cancel and Close buttons

Choosing the Cancel button discards the options you have selected, closes the dialog box, and returns you to your drawing. When DesignCAD completes an action that cannot be canceled, the Cancel button changes to the Close button.

The Close button closes the dialog box without reversing any completed changes. To choose the Cancel button, you can click the button or press the **Esc** key. You can also use the Close command on the Control menu in the upper-left corner of the dialog box.

See Also: [Using Dialog Boxes](#), [Typing and Editing in Dialog Boxes](#)

Selecting a Menu

If you have a mouse, you can move the pointer to the menu in the Command Menu. You can also press **Alt** and move to the menu with the arrow keys.

When you press the left mouse button or **Enter**, the menu will "pull-down."

You can use the mouse or the arrow keys to move the highlight bar up and down the menu to choose a command. You can also press the highlighted letter of the command you want to choose.

Secondary menus

To choose the Load Selection command, choose the word "Load" from the Edit menu. Then move the highlight bar to **Load Selection** and press the left mouse button or the **Enter** key. Because an arrow follows the word "Load," a secondary menu with more commands will "pop up" on the screen.

Using arrow keys

If you are not using a mouse, you can select menus by pressing **Alt** and the arrow keys. Once a menu is pulled down, you can choose the command you want by pressing the arrow keys. You can choose a command from the menu by pressing **Enter**.

See Also: [Using Menus](#), [Selecting Commands with a Mouse](#), [Selecting Commands with the Keyboard](#), [Selecting Commands with a Digitizer](#)

Switching to a Different Drawing Window

With DesignCAD, you can have more than one view of a drawing open at a time. You can open several drawing views, depending on how much memory is available on your computer. A separate drawing window is opened for each view. The Window menu lists the views sequentially.

The window you are currently working in, the window containing the cursor, is called the active window. The view you are working in is called the active view. Drawing commands affect each window. To work in a different view, you must make that drawing window active.

Making a drawing window active

From the Window menu, choose the name of the drawing view you want to make active.

Making the next open window active

If the window you want to work in is visible, click anywhere within its borders.

-or-

Press **Alt** to choose the drawing Control-menu box.

Press **Enter** to open the drawing Control-menu box.

Choose the Next command.

-or-

Press **Ctrl+F6** to choose the Next command.

See Also: [Windows](#), [Moving and Sizing Windows](#), [Opening a New Window](#)

Moving and Sizing Windows

[Arranging Windows and Icons](#)

[Moving Windows](#)

[Maximizing Windows](#)

[Minimizing Windows](#)

[Restoring a Drawing Window to its Former Size](#)

[Closing a Window](#)

Arranging Windows and Icons

Cascading windows

Choose the Cascade command from the Window menu.

Tiling windows

Choose the Tile Horizontal command from the Window menu.

-or-

Choose the Tile Vertical command from the Window menu.

Arranging Icons

Different views of the drawing window become icons when they are minimized. You can move these icons around the screen.

Choose the Arrange Icons command from the Window menu.






See Also: [Tiling Windows](#), [Cascading Windows](#), [Arranging Icons](#)

Cursor Pointer Shapes

Pointing to different parts of the screen, causes the pointer shape to change, allowing you to perform different tasks. Some commands also change the pointer shape.

If the pointer assumes a shape you do not want to use, press **Esc** to restore the pointer to its usual shape.

The following table lists the common pointer shapes.

Shape	Significance
	The Select Mode is active, or the pointer is in the menus, scroll bars, Toolbox, or drawing area. You can choose a menu and command, click a button, set points, or move an object.
	The Point Select Mode is active. You can select an entity, show the points of that entity, and move or delete points.
	The Preset Point Mode is active. You can set points before selecting a command.
	DesignCAD is performing a task that will take a few seconds.
	The pointer is in a selection box. You can change the size of an object vertically.



You can change the size of an object horizontally.

You can change the size of an object diagonally.

You can change the diagonal width or height of an open window.

You can move entities to new positions, or "pan" the drawing across the screen with the Pan command.

Text command is selected. You can move the cursor and the text box to the location for the text and set a point.



Shift+F1 has been pressed to select the Help Index command. You can select a command to find specific information on the command.

When you are setting points for a command, this cursor appears.

The Select Scale or Select Zoom command is active and the cursor is on either the top or bottom center nodes of the selection box.

The Select Scale or Select Zoom command is active and the cursor is on either the left or right center nodes of the selection box.

The Point Select Mode is active. A line can be bent with this cursor.

The Point Select Mode is active and the **Ctrl** and **Shift** keys are being pressed. A point can be removed from the drawing with this cursor.



The Zoom Out command is selected. A point is set for the zoom origin. That point is moved to the center of the screen.



The Zoom command is selected. Set a point for the zoom origin and a point for the zoom destination.

The Show Hot Toolbox option has been selected and an icon from the Toolbox is being dragged to the Hot Toolbox.

The Same as button has been chosen in the Command Line or a command box. You can select an entity in the drawing.

See Also: [Using a Mouse](#)

Loading and Selecting Objects

Choose the Load Selection command from the File menu.

Choose the drawing name from the File Name box.

Choose the OK button.

Set a point for the lower left corner of the drawing. This point sets the location of the drawing. If only one point is set, the drawing will be drawn the size and angle at which it was saved.

Set a point for the lower right corner of the drawing. This point determines the size and angle of the drawing.














Set a point for the upper left corner of the drawing. This point determines the slant of the drawing.

Set a point for the upper right corner of the drawing. The drawing will be loaded and the objects will be selected.

See Also: [Load Selection command](#), [Loading Objects](#), [Selecting Objects](#)

Choosing Point Commands

DesignCAD offers several ways to choose the various point commands.

Button	Command	Shortcut Keys
	<u>Center of Gravity</u>	
	<u>Center of Gravity Move</u>	
	<u>Gravity</u> .	
	<u>Gravity Move</u> ,	
	<u>Intersect-1</u> N	
	<u>Intersect-1 Move</u>	Ctrl+N
	<u>Intersect-2</u>	
	<u>Intersect-2 Move</u>	
	<u>Line Snap</u> K	
	<u>Line Snap Move</u>	Ctrl+K
	<u>Midpoint</u> F2	
	<u>Midpoint Move</u>	Ctrl+F2
	<u>Point Line Snap</u>	
	<u>Point Polar</u> ;	
	<u>Point Relative</u> '	
	<u>Point Set</u> Ins	
	<u>Point XY</u> :	

Coordinate Bar

Pressing **F6** chooses the X coordinate box in the Coordinate Bar at the top of the screen. The Coordinate Bar allows you to set a specific coordinate for the X, Y, Delta X, and Delta Y cursor locations or polar coordinates.

See Also: [Setting Points with DesignCAD Commands](#)

Setting Points with Commands

You can select a DesignCAD drawing command before setting points. With most commands, a "rubber band line" shows you how the entity will be drawn as the points are being set. With most commands, after all the points are set, you must press **Enter** or click the middle mouse button to end the command.

By activating the Preset Point Mode first, you can set the points for a command and then select the command.

When you zoom, move, or rotate an object, the points and lines may not lie exactly at the correct position on the screen. The pointer, however, remains exactly on top of the dots or pixels on the screen. A point set on top of another point using the Point Set command may be a fraction of a dot off, although it will appear to be correct. The difference is seen when the drawing is zoomed. The Gravity command prevents this error from occurring.

See Also: [Setting Points with DesignCAD Commands](#), [Selecting Command First](#), [Setting Points First](#)

Opening a New Screen

Choose the New command from the File menu.

When asked if you want to save the drawing, choose the Yes button to save the drawing.

Choose the No button to clear the screen without saving the drawing.

-or-

Press **Esc**.

Choose the Cancel button to return to the drawing screen.

See Also: [Clearing the Screen](#)

Setting a New Origin

The Zoom, Rotate, and Pan commands may change the location of the origin on the screen, but it remains at the same point in the drawing. The Origin command, the Zoom Static button, or the load commands can change the origin of the drawing.

Setting the origin

Set a point at the location for the new origin.

Choose the Origin command from the View menu.

Until it is changed again, the origin will be at that point.

See Also: [Changing the Origin](#)

Printing a Drawing

It is a good idea to save your drawing before you print. That way, if a printer error or other problem occurs, you will not lose any of the work you have done since you last saved your drawing.

Printing a drawing

- Display the drawing you want to print in the active window.
- Choose the Print command from the File menu.
- Change any print options in the Print Options dialog box.
- Choose the Print button.

Canceling printing

- Press **Esc**.

See Also: [Printing Part of a Drawing](#), [Printing Several Copies](#), [Printing](#), [Printer Setup](#), [DesignCAD Print Options](#)

Printing Selected Objects

DesignCAD allows you to print only those objects selected in the drawing with the Print command.

Printing part of a drawing

Select the objects you want to print.

Choose the Print command.

If the Print Selection box is not selected, choose it.

Choose the Print button.

See Also: [Selecting Objects](#), [Printing](#), [Printer Setup](#), [DesignCAD Print Options](#)

Printer Setup

Because various printers have different printing capabilities, the dialog box that appears when you choose the Printer Setup command depends on the printer.

Most printers can accommodate different paper sizes, orientations, and paper sources. You can set the default paper size, orientation, and paper source for your drawing in this dialog box.

Selecting printer setup options

- Choose the Printer Setup command from the File menu.

- Choose the printer you want to use in the Printer box, if more than one is available.

- Choose the Setup button.

- Select the appropriate options.

- Choose the OK button.

See Also: [Printing](#), [Printer Setup command](#), [DesignCAD Print Options](#)

Setting the Print Quality

With DesignCAD, you can print drawings with different resolutions. Depending on your printer, the printer setup dialog box allows you to choose between 300 d.p.i., 150 d.p.i., or 75 d.p.i.. This resolution determines the total number of dots-per-inch printed with each drawing.

Setting the dots per inch for a drawing

Display the drawing you want to print in the active window.

Choose the Printer Setup command from the File menu.

Choose the Setup button.

Choose the dots per inch in the Resolution box.

Choose the OK button.

See Also: [Printing](#), [Printer Setup](#)

Changing Page Orientation

DesignCAD offers you a choice between "portrait" (vertical) and "landscape" (horizontal) page orientations. You can change the orientation of a drawing within the Print dialog box.

Setting print orientation

Choose the Print command from the File menu.

Choose Portrait in the Orientation box to have your drawing printed with the drawing width the same direction as the printer width.

Choose Landscape in the Orientation box to have your drawing rotated 90 degrees, or printed sideways on the page.

Choose the Print button.

See Also: [Printing](#), [Printer Setup](#)

Changing Paper Size

Depending on the printer you have installed, you can choose different paper sizes to print a DesignCAD drawing. You can change the paper size loaded into your printer within the Options dialog box.

Setting paper size

Choose the Print command from the File menu.

In the Paper box, choose the size of the paper you want to use. The options available will depend on the printer you selected.

Choose the source of the paper you want to use. The options available will depend on the printer you selected.

Choose the Print button.

See Also: [Printing](#), [Printer Setup](#)

Setting Dithering

Depending on the printer you have selected, you may have four options related to dithering in the Options dialog box: None, Coarse, Fine, and Line Art.

None This option turns off dithering. DesignCAD prints graphics in black and white with no gray shading.

Coarse Select this option if you have specified a resolution value of 300 dots per inch or more in the Print dialog box.

Fine Select this option if you have specified a resolution value of 200 dots per inch or less in the Print dialog box.

Line Art Select this option if the drawing you want to print includes well-defined borders between black, white, and gray shadings. Do not use this option if you are printing scanned images that include continuous variations in intensity and hue.

Setting dithering

- Choose the Printer Setup command in the File menu.
- Choose the Setup button.
- Choose the Options button.
- Choose option in the Dithering dialog box.
- Choose the OK button.

See Also: [Printing](#), [Printer Setup](#)

Setting Intensity

You can control the intensity of a drawing depending on the printer you choose. The Intensity option decreases or increases the darkness of your drawings. Moving the icon to the left, or pressing the left arrow key, sets the intensity to darker. Moving the icon to the right, or pressing the right arrow key, sets the intensity to lighter. Setting the icon in the middle sets the intensity to normal.

Controlling intensity

- Choose the Printer Setup command.
- Choose the Setup button.
- Choose the Options button.
- Choose the Intensity Control box.
- Move the page control in the Intensity Control dialog box.
- Choose the OK button.

See Also: [Printing](#), [Printer Setup](#)

Selecting TrueType Fonts

Depending on the printer you choose with the Printer Setup command, [TrueType fonts](#) can be printed as graphics. These fonts require less printer memory and may speed up printing time.

Printing TrueType as graphics

- Choose the Printer Setup command.
- Choose the Setup button.
- Choose the Options button.
- Choose the Print TrueType as Graphics check box.
- Choose the OK button.

See Also: [Printing](#), [Printer Setup](#)

Using the Help Window

The Help window has buttons you can use to move around the Help system easily. If a particular feature is not available, the button associated with it is dimmed.

By choosing a button at the top of the Help window, you can display a list of Help topics, search for a list of topics that include a key word, and move forward or backward through the Help topics.

The Help window marks some words with a dotted or solid line and displays them in color if you have a color display. Words marked with a solid line are "jump terms." When you click a jump term, you immediately move to a Help topic associated with that word. Clicking a word or phrase marked with a dotted line displays a definition of that term.

Getting help about Help

Choose the How to Use Help command from the Help menu.

Choose the information you want to see.

See Also: [Arranging Windows and Icons](#), [Getting Help as You Work](#), [Finding Information in Help](#), [Help Instructions](#)

Finding Information in Help

There are three ways to find the information you need in Help:

- Use the Search feature in Help.
- Get context-sensitive Help as you work.
- Use the Help Index.

Getting help in a dialog box

With the dialog box displayed, press **F1**.

Getting help on a command

Press **Shift+F1**.
Choose a command in the Command Menu.

Getting help on a key combination

Press **Shift+F1**.
Press a key combination.

Getting a definition

In a Help topic, choose a term or phrase with a dotted line.
-or-
Press **Tab** to select a term with a dotted line.
Press **Enter** to read the definition.

Minimizing Help

Click the Minimize button in the upper-right corner of the Help window.
-or-
Choose Minimize from the Control menu.

This procedure closes the Help window. However, the Help icon remains on the desktop. When you press **F1**, Help reappears. The last topic you viewed is displayed in the window.

Closing Help

Choose the Exit command from the Help window.

See Also: [Arranging Windows and Icons](#), [Getting Help as You Work](#), [Using the Help Window](#), [Help Instructions](#)

Marking Topics

A bookmark marks specific references in the Help window. With DesignCAD, you can place bookmarks at Help topics you use frequently. Once you have placed a bookmark at a topic, you can display that topic quickly by choosing the bookmark from the Bookmark menu in Help.

Placing a bookmark

Display the topic.

Choose the Define command from the Bookmark menu.

To use the title as the bookmark name, choose the OK button.

-or-

To use a different name, type a name for the bookmark. Then choose the OK button.

Selecting a marked topic

Open the Bookmark menu.

Choose the name of the bookmark you want to view.

Removing a bookmark

Choose the Define command from the Bookmark window.

Select the bookmark you want to remove.

Choose the Delete button.

Choose the OK button.

See Also: [Using the Help Window](#)

Adding Notes

You can add your own comments to Help topics. These comments can be up to 2,000 characters long, using any characters on the keyboard. When you add a comment, Help places a paper clip symbol to the left of the title of the topic to remind you that you have added text to the topic. DesignCAD does not print comments when the topic is printed.

Adding text to a Help topic

- Display the topic.
- Choose the Annotate command from the Help Edit menu.
- Type the text.
- Choose the Save button.

Viewing a comment

- Display the topic.
- Choose the paper clip symbol.
- Choose the Cancel button.

Removing a comment

- Display the topic.
- Choose the paper clip symbol.
- Choose the Delete button.

See Also: [Using the Help Window](#)

Setting Regular Points

The Point Set command sets a point at the cursor location. There are several ways to set a point with the Point Set command. You can press the **Ins** key on the keyboard. You can also set a point by pressing the left button on a mouse or the first button on a digitizer.

See Also: [Setting Points with DesignCAD Commands](#), [Choosing Point Commands](#)

Setting Gravity Points

Choose the Gravity command.
Move the pointer near the point.
Click the left mouse button.

-or-

Move the pointer near the point.
Set a point.

Moving to gravity without setting a point

Choose the Gravity Move command.
Move the pointer near the point.
Press **Enter**.

Finding the center of gravity

Choose the Center of Gravity command.
Move the pointer near the point.
Press **Enter**.

Finding center of gravity without setting a point

Choose the Center of Gravity Move command.
Move the pointer near the entity.
Press **Enter**.

See Also: [Setting Points with DesignCAD Commands](#), [Choosing Point Commands](#)

Snapping to Lines

Snapping to a line

Choose the Line Snap command.

Move the pointer near the line.

Click the left mouse button.

-or-

Set a point.

Snapping to line without setting a point

Choose the Line Snap Move command.

Move the pointer near the line.

Click the left mouse button.

-or-

Set a point.

See Also: [Setting Points with DesignCAD Commands](#), [Choosing Point Commands](#)

Selecting Points

The Point Select Mode allows you to display all the points making up an entity. After a line or entity is selected, each point making up that entity is highlighted with a small square, or select node. This command selects entities so that you can use other commands to move, add, or delete points.

Showing points

Activate the Point Select Mode.

Move the pointer near the entity.

Click the entity to select it.

-or-

Press **Ins** to select the entity.

See Also: [Choosing Point Commands](#), [Point Select Mode](#)

Moving and Copying Objects

Moving Objects

Moving Objects with Handles

Moving Objects Vertically

Moving Objects Horizontally

Copying Objects

Copying Objects with Handles

Drawing Multiple Copies

Copying Objects in Row

Copying Objects in Circle

Drawing Regular Arcs

Drawing Arc with Center and Radius

Drawing Arc with Beginning and End

Drawing Arc with Three Points

Drawing Arc with Center, Beginning, and End

Drawing Arc with Beginning, End, and Radius

Drawing Arc with Center and Radius

Choose the Arc-1 command.

To change the angle of the arc, type that number in the Angle box in the Command Line.

To save the arc as a vector entity, choose the Save as vector form box in the Command Line.

To have the arc drawn with the same angle as another arc, choose the Same as box. Then set a point on the arc in the drawing.

Set a point for the center of the arc.

Set a point for the beginning of the arc.

See Also: [Arc-1 command](#)

Drawing Arc with Beginning and End

Choose the Arc-2 command.

To change the radius of the arc, type that number in the Radius box in the Command Line.

To save the arc as a vector entity, choose the Save as vector form box in the Command Line.

To have the arc drawn with the same angle as another arc, choose the Same as box in the Command Line. Then set a point on the arc in the drawing.

Set a point for the beginning of the arc.

Set a point for the end of the arc.

See Also: [Arc-2 command](#)

Drawing Arc with Three Points

Choose the Arc-3 command.

To save the arc as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the beginning of the arc.

Set a point for the center of the arc.

Set a point for the end of the arc.

See Also: [Arc-3 command](#)

Drawing Arc with Center, Beginning, and End

Choose the Arc-4 command.

To save the arc as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the center of the arc.

Set a point for the beginning of the arc.

Set a point for the end of the arc.

See Also: [Arc-4 command](#)

Drawing Arc with Beginning, End, and Radius

Choose the Arc-5 command.

To save the arc as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the beginning of the arc.

Set a point for the end of the arc.

Set a point for the center, or radius, of the arc.

See Also: [Arc-5 command](#)

Drawing Elliptical Arcs

Choose the Elliptical Arc command.

To save the arc as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the center of the arc.

Set a point for the beginning of the arc.

Set a point for the end of the arc.

See Also: [Elliptical Arc command](#)

Drawing Part of a Circle

Drawing quarter circle

Choose the Quarter Circle command.

To save the quarter circle as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the beginning of the quarter circle.

Set a point for the end of the quarter circle.

Drawing semi-circle

Choose the Semi-Circle command.

To save the semi-circle as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the beginning of the semi-circle.

Set a point for the end of the semi-circle.

See Also: [Quarter Circle command](#), [Semi-Circle command](#)

Drawing Regular Curves

Choose the Curve command.

To save the curve as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the beginning of the curve.

Set one or more points for the center of the curve.

Set a point for the end of the curve.

Press **Enter**.

See Also: [Curve command](#)

Drawing Bezier Curves

Choose the Bezier Curve command.

Choose the Save as vector form box in the Command Line to save the curve as a vector entity.

Set a point for the beginning of the curve.

Set a point for the first control point.

Set a point for the second control point.

Set a point for the end of the curve.

Press **Enter** to end the command.

See Also: [Bezier Curve command](#)

Drawing Freehand Curves

Choose the Sketch command.

Press and hold down the left mouse button.

Move the pointer in a curve motion.

Release the mouse button.

-or-

Press **Alt+Ins** to start the Sketch command.

Use the **Alt** and arrow keys to move the cursor.

Press **Enter** or **Ins** to end the command.

See Also: [Sketch command](#)

Drawing Regular Circles

[Drawing Circle with Center and Radius](#)

[Drawing Circle with Diameter Points](#)

[Drawing Circle with Three Points](#)

[Drawing Circle with Center and Radius Length](#)

Drawing Circle with Center and Radius

Choose the Circle-1 command.

To save the circle as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the center of the circle.

Set a point for the radius of the circle.

See Also: [Circle-1 command](#)

Drawing Circle with Diameter Points

Choose the Circle-2 command.

To save the circle as a vector entity, choose the Save as vector form box in the Command Line.

Set a point for the endpoint of the diameter.

Set a point for the other endpoint of the diameter.

See Also: [Circle-2 command](#)

Drawing Circle with Three Points

Choose the Circle-3 command.

To save the circle as a vector entity, choose the Save as vector form box in the Command Line.
Set three points anywhere on the circle.

See Also: [Circle-3 command](#)

Drawing Circle with Center and Radius Length

Choose the Circle-4 command.

Type the length in the Radius box in the Command Line.

To save the circle as a vector entity, choose the Save as vector form box in the Command Line.

To have the radius length the same as another circle in the drawing, click the Same as box in the Command Line. Then set a point on the other circle in the drawing.

Set a point for the center of the arc.

See Also: [Circle-4 command](#)

Drawing Tangent Circles

[Drawing Circle Tangent to Two Lines](#)

[Drawing Circle Tangent to Three Lines](#)

Drawing Circle Tangent to Two Lines

Choose the Tangent Two Lines command.

Type the length of the radius in the Radius box in the Command Line.

To have the radius drawn the same length as another circle in the drawing, choose the Same as box.

Then set a point on the circle in the drawing.

Set a point on the first line.

Set a point on the second line.

See Also: [Tangent Two Lines command](#)

Drawing Circle Tangent to Three Lines

Choose the Tangent Three Lines command.

Set a point on the first line.

Set a point on the second line.

Set a point on the third line.

See Also: [Tangent Three Lines command](#)

Drawing Elliptical Circles

Choose the Ellipse command.

To save the arc as a vector entity, choose the Save as vector form box.

Set a point for the center of the ellipse.

Set a point for one axis of the ellipse.

Set a point through which the ellipse passes.

See Also: [Ellipse command](#)

Drawing Regular Lines

Drawing a line

Choose the Line command.
Set a point for the beginning of the line.
Set up to 198 points for the line.
Set a point for the end of the line.
Press **Enter**.

Drawing an ortho line

Choose the Ortho Line command.
Set a point for the beginning of the line.
Set up to 198 points for the line.
Set a point for the end of the line.
Press **Enter**.

Drawing an arrow

Choose the Arrow command.
To change the size of the arrowhead, type the size in the Arrowhead Size box in the Command Line.
To change the type of arrowhead, choose the arrowhead type in the Arrowhead Type box in the Command Line.
To have the arrowhead drawn the same as another arrow, choose the Same as box in the Command Line. Then choose another arrow entity in the drawing.
Set a point for the beginning of the arrow.
Set up to 198 points for the arrow.
Set a point for the end of the arrow.
Press **Enter**.

See Also: [Changing Line Characteristics](#)

Drawing Parallel Lines

Drawing parallel lines with two points

Choose the Parallel-1 command.

Set a point on the line to which the parallel line is to be drawn.

Set a point at the location of the parallel line.

Drawing a parallel line with distance

Choose the Parallel-2 command.

Type the distance the parallel is to be drawn from the line in the Distance box in the Command Line.

Set a point on the line to which the parallel line is to be drawn.

Set a point the direction the parallel line is to be drawn.

See Also: [Changing Line Characteristics](#)

Drawing Perpendicular Lines

Drawing a perpendicular line

Choose the Perpendicular To Line command.

Set a point for the beginning of the perpendicular line.

Set a point on the line the perpendicular line is to be drawn to.

Drawing a perpendicular line

Choose the Perpendicular From Line command.

Type the length of the perpendicular line in the Distance box in the Command Line.

Set a point on the line to which the perpendicular line is to be drawn.

Set a point the direction the perpendicular line is to be drawn.

See Also: [Changing Line Characteristics](#)

Drawing Tangent Lines

Drawing a tangent from a point

- Choose the Tangent To Circle command.
- Set a point for the beginning of the line.
- Set a point on or near the entity.

Drawing a tangent between circles

- Choose the Tangent Between Circles command.
- Set a point on or near the first circle.
- Set a point on or near the second circle.

Drawing a tangent from a circle

- Choose the Tangent From Circle command.
- Set a point on or near the entity.
- Set a point for the end of the line on or near the line tangent to the circle. The endpoint of the line will be even with the second point.

See Also: [Changing Line Characteristics](#)

Drawing Short Line Segments

Drawing tick marks

Choose the Tick Marks command.

Type the number of sections of tick marks in the Sections box in the Command Line.

Type the number of divisions of tick marks in the Divisions box in the Command Line.

Type the large tick mark size in the Large tick mark box in the Command Line.

Type the small tick mark size in the Small tick mark box in the Command Line.

Set a point on the entity.

Set a point to indicate which side of the entity the tick marks will be drawn.

Centering tick marks

Choose the Tick Marks command.

Type the number of sections of tick marks in the Sections box in the Command Line.

Type the number of divisions of tick marks in the Divisions box in the Command Line.

Type the large tick mark size in the Large tick mark box in the Command Line.

Type the small tick mark size in the Small tick mark box in the Command Line.

Set two points on the entity in the same location.

See Also: [Changing Line Characteristics](#)

Changing Line Characteristics

The Show Line Style Box option selects the line pattern, width, and scale for lines drawn in DesignCAD. There are 13 line types to choose from.

Changing line style

Choose the Show Line Style Box option.

Choose the Line Type box.

Choose the new line type.

To change the scale of the line, choose the Scale box and type the new scale or use the arrow buttons to change the scale in .10 Drawing Unit increments.

To change the width of the line, choose the Width box and type the new scale or use the arrow buttons to change the width in .10 Drawing Unit increments.

To have the line drawn with the same scale and width of another line in the drawing, choose the Same as button. Then set a point on the line in the drawing.

Changing line style of selected lines

Select the lines to be changed.

Choose the Show Line Style Box option.

Choose the line type from the line type menu.

To change the scale of the line, choose the Scale box and type the new scale or use the arrow buttons to change the scale in .10 Drawing Unit increments.

To change the width of the line, choose the Width box and type the new scale or use the arrow buttons to change the width in .10 Drawing Unit increments.

To have the line drawn with the same scale and width as another line in the drawing, choose the Same as button. Then set a point on the line in the drawing.

Choose the Apply box.

Removing line style menu

Choose the window Control-menu in the upper left corner of the menu.

-or-

Choose Show Line Style Box in the Options menu.

When the Fill Wide Lines option is selected wide lines are filled.

Filling wide lines

Choose the Fill Wide Lines option.

-or-

Choose the Show Line Style Box option.

Choose the Fill Wide Line box.

See Also: [Show Line Style Box option](#)

Drawing Simple Polygons

[Drawing a Polygon with Corner Points](#)

[Drawing a Polygon with Center and Corner](#)

Drawing a Box

Choose the Box command.
Set a point for the first corner of the box.
Set a point for the opposite corner of the box.

Drawing a diagonal box

Choose the Box command.
Choose the Align to any angle box in the Command Line.
Set a point for the first corner of the box.
Set a point for the opposite corner of the box.
Set a point for the opposite edge of the box.

See Also: [Box command](#)

Drawing Polygon with Corner Points

Choose the Polygon-1 command.

Type the number of sides in the Number of sides box in the Command Line.

To have the polygon drawn with the same number of sides as a polygon in the drawing, choose the Same as box in the Command Line. Then set a point on the polygon in the drawing.

Set a point for the corner of the polygon.

Set a point for the opposite corner of the first side of the polygon.

See Also: [Polygon-1 command](#)

Drawing Polygon with Center and Corner

Choose the Polygon-2 command.

Type the number of sides in the Number of sides box in the Command Line.

To have the polygon drawn with the same number of sides as a polygon in the drawing, choose the Same as box in the Command Line. Then set a point on the polygon in the drawing.

Set a point for the center of the polygon.

Set a point for the first corner of the polygon.

See Also: [Polygon-2 command](#)

"Unerasing" Objects

If you erase an entity by mistake

Choose the Unerase command from the Edit menu.

See Also: [Unerase command](#), [Erasing Objects](#)

Redoing Commands

After an Undo command, choose the Redo command from the Edit menu.

See Also: [Undoing Commands](#)

Cutting Objects

Select the entity.

Choose the Cut command from the Edit menu.

See Also: [Cut command](#)

Pasting Objects

Choose the Paste command from the Edit menu.

See Also: [Paste command](#)

Changing Lines

[Bending Lines](#)

[Deleting Points](#)

[Extending Lines](#)

[Moving Points](#)

[Stretching a Line](#)

Breaking Lines

Defining a section

- Choose the Cutoff command.
- Set a point in the corner of the section.
- Set a point in the opposite corner of the section.

Breaking a line

- Select the line to be broken.
- Choose the Break Line command.

Segmenting entities

- Choose Segment from the Edit menu.
- Type the number of sections in the No. of sections box in the Command Line.
- Set a point on the entity to be segmented.
- To see how the line was segmented, select the line.

See Also: [Breaking a Line](#), [Break Line command](#), [Editing Lines](#)

Drawing Regular Text

Drawing Text

Drawing Text in a Block

Centered Text

Left-justified Text

Right-justified Text

Drawing Text in an Arc

Drawing Text

Choose the Text command.

Choose the Text box in the Command Line and type the text.

Type the size of text in the Size box in the Command Line.

Type the angle of the text in the Angle box in the Command Line.

To draw the text left justified, choose Left in the Orientation box.

To have the text drawn in bold text, choose the Bold button.

To have the text drawn in italic text, choose the Italic button.

Set a point for the lower left corner of the text.

Set a point for the lower right corner of the text.

To have the text drawn with the same options as other text in the drawing, choose the Same as button.

Then select the text in the drawing.

See Also: [Text command](#)

Drawing Text in a Block

Choose the Text command.

Type the text size in the Size box in the Command Line.

Type the angle of the text in the Angle box in the Command Line.

Choose the Bold button in the Command Line to have the text drawn in bold text.

Choose the Italic button in the Command Line to have the text drawn in italic text.

Choose the Text Block button in the Command Line.

Type the text in the Text box.

Type the distance between lines in the Line box.

Choose the OK button.

Set a point for the upper left corner of the text.

Set a point for the angle of the text. Press the middle mouse button or **Enter** to have the text drawn without setting a point for the angle.

To have text drawn with the same options as other text in the drawing, choose the Same as button in the Command Line. Then set a point on the text in the drawing.

See Also: [Text command](#)

Drawing Attributes

The Attribute command assigns information to an object in the drawing. This information can later be extracted from the drawing file and used with other programs for applications such as [parts lists](#) and a [Bill of Materials lists](#).

Attributes will not be displayed on the screen unless the Show Attribute option is selected in the View menu. Whether the attribute is displayed on the screen or not, it will be copied, deleted, etc., as a normal entity.

Attributes can be extracted and used by the [Materials List command](#). This command creates a list of all attributes in a drawing and outputs this information to the printer or a disk file for use by other applications.

Adding attributes

Choose the Attribute command.

Choose the Attribute box in the Command Line and type the text.

If the show attributes options is selected, type the size of the attribute in the Size box.

Set a point for the lower left corner of the attribute.

Displaying attributes

Choose the Attribute command.

Choose the Show attributes box in the Command Line.

-or-

Choose Show Attribute from the View menu.

See Also: [Text Options](#), [Changing Text Options](#)

Adding Information to a Drawing

Drawing a Balloon around Information

Pulling out Information in a Drawing

Drawing a Balloon Around Information

Choose the Balloon command.

Choose the Text box in the Command Line and type the text for the balloon.

Type the size of the balloon, or circle, to be drawn around the text in the Balloon Size box in the Command Line.

Choose the Line box in the Command Line to set the arrowhead type, size, and line position.

To have the balloon drawn with the same options as another balloon in the drawing, choose the Same as button. Then set a point on the balloon in the drawing.

Set a point for the balloon arrowhead.

Set a point for the direction of the extension line.

Set a point for the location of the text information.

Press **Enter**.

See Also: [Balloon command](#), [Pullout command](#)

"Pulling Out" Information In a Drawing

Choose the Pullout command.

Choose the Text box in the Command Line and type the text for the pullout.

Choose the Line button in the Command Line to change the position, size, and type of the arrowhead.

Choose the Text button in the Command Line to change the format, precision, location, orientation, and size of the text.

To have the pullout drawn with the same options as another pullout in the drawing, click the Same as button. Then set a point on the balloon in the drawing.

Set a point for the pullout arrowhead.

Set a point for the direction of the extension line.

Set a point for the location of the text information.

Press **Enter**.

See Also: [Balloon command](#), [Pullout command](#)

Setting Font

The Text Options command sets the character font, style, size, angle, and spacing for text in the drawing. The font selected with this command determines how text drawn with the Attribute, Text, and Dimension commands will look. The size of the text is set with these various commands.

Choosing the font

Choose Text Options from the Options menu.

Choose the new font name in the Font box.

Choose a new style in the Font Style box.

Choose the text size in the Size box.

Choose the text angle in the Angle box.

Choose the spacing for text in the Text Block dialog box in the Spacing box.

Choose the OK button.

See Also: [Changing Text Font](#), [Changing the Text and Font](#)

Changing Text Layer and Color

Changing the layer of text

- Select the text to be changed.
- Choose Show Info Box from the View menu.
- Choose the Layer box in the Info Box.
- Choose the new layer.

Changing the text color

- Select the text to be changed.
- Choose Show Info Box from the View menu.
- Choose the Color box in the Info Box.
- Choose the new color in the Info Color box.
- Choose the OK button.

See Also: [Text Options](#)

Changing Text and Font

Changing the text

- Select the text to be changed.
- Choose Show Info Box from the View menu.
- Choose the Content box in the Info Box.
- Type the new text in the Text box.
- Choose the OK button.

Changing the font

- Select the text to be changed.
- Choose Show Info Box from the View menu.
- Choose the Font box in the Info Box.
- Choose the new font in the Font dialog box.
- Choose the OK button.

See Also: [Text Options](#)

Changing Height, Length, and Angle

Changing the text height

Select the text to be changed.
Choose Show Info Box from the View menu.
Click the Height box in the Info Box.
Type the new text size in the Height box.

Changing the text length

Select the text to be changed.
Choose Show Info Box from the View menu.
Click the Length box in the Info Box.
Type the new text size in the Length box.

Changing the text angle

Select the text to be changed.
Choose Show Info Box from the View menu.
Click the Angle box in the Info Box.
Type the new text angle in the Angle box.

See Also: [Text Options](#)

Changing Text Style

Select the text to be changed.
Choose Show Info Box from the View menu.
Choose the Bold button in the Style box.
Choose the Italic button in the Style box.

See Also: [Text Options](#)

Scaling Objects with Select Scale

Scaling Selected Objects

Changing Scale Center

Scaling Objects with Selection Box

Scaling One Side of Object

Scaling Objects with Constant Aspect Ratio

Scaling Three Sides of Object

Scaling Selected Objects

Changing the width of an object

Select the object to be scaled.

Choose Select Scale from the Edit menu.

Click the left or right select nodes.

Move the pointer in the desired direction.

Set a point.

-or-

Use the arrow keys to move the pointer to the left or right select nodes.

Press **Alt+Ins**.

Use the arrow keys to move the pointer.

Set a point.

Changing the height of an object

Select the object to be scaled.

Choose Select Scale from the Edit menu.

Click the top or bottom select nodes.

Move the pointer in the desired direction.

Set a point.

-or-

Use the arrow keys to move the pointer to the top or bottom select nodes.

Press **Alt+Ins**.

Use the arrow keys to move the pointer.

Set a point.

Changing diagonal width or height of object

Select the object to be scaled.

Choose Select Scale from the Edit menu.

Click the left or right select nodes.

Move the pointer in the desired direction.

Set a point.

-or-

Use the arrow keys to move the pointer to the select nodes.

Press **Alt+Ins**.

Use the arrow keys to move the pointer.

Set a point.

See Also: [Select Scale command](#)

Changing Scale Center

Select the object to be scaled.
Choose Select Scale from the Edit menu.
Set a point on the center point.
Move the pointer in the desired location.
Set a point at the new location of the center.

See Also: [Cursor Pointer Shapes](#), [Select Scale command](#)

Finding Angle of Points and Lines

[Calculating Angle of Two Points](#)

[Adding Angle of Objects to Drawing](#)

[Calculating Angle of Two Lines](#)

[Displaying Angle of Two Lines](#)

Finding Area

Calculating Area

Adding Area to Drawing

Performing Calculations

Calculating Information

Inserting Calculation Into Drawing

Adding Area to Drawing

In the Area dialog box, highlight the text.
Press **Ctrl+C** to copy the text to the Clipboard.
Choose the OK button.
Choose the Text command from the Toolbox.
Choose the Text box in the Command Line.
Press **Ctrl+V** to paste the Clipboard contents.
Set the points for the text.

See Also: [Area command](#), [Calculating Area](#)

Calculating Angle of Two Points

Choose the Angle, Distance-Two Points command.

Set two points for the angle and distance to be measured.

Choose the Format box in the Angle box to set the angle format.

Choose the Precision box in the Angle box to set the angle precision.

Choose the Format box in the Distance box to set the linear format.

Choose the Precision box in the Distance box to set the linear precision.

See Also: [Angle, Distance-Two Points command](#), [Adding Angle of Objects to Drawing](#)

Adding Angle of Objects to Drawing

In the Angle and Distance dialog box, highlight the text.
Press **Ctrl+C** to copy the text to the Clipboard.
Choose the OK button.
Choose the Text command.
Click the Text box in the Command Line.
Press **Ctrl+V** to paste the Clipboard contents.
Set the points for the text.

See Also: [Angle, Distance-Two Points command](#), [Calculating Angle of Two Points](#)

Calculating Angle of Two Lines

Choose the Angle Between Two Lines command.
Set a point on the first line of the angle to be measured.
Set a point on the second line.
Choose the Format box to choose the format.
Choose the Precision box to choose the precision.
Choose the OK button.

See Also: [Displaying Angle of Two Lines](#), [Angle Between Two Lines command](#)

Displaying Angle of Two Lines

In the Angle Between Two Lines dialog box, highlight the angle text.
Press **Ctrl+C** to copy the text to the Clipboard.
Choose the OK button.
Choose the Text command.
Choose the Text box in the Command Line.
Press **Ctrl+V** to paste the Clipboard contents.
Set the points for the text.

See Also: [Calculating Angle of Two Lines](#), [Angle Between Two Lines command](#)

Zooming Selected Objects

Zooming an object

Select the object to be zoomed.

Choose the Select Zoom command from the Edit menu.

Click a point on the highlighting box.

Move the pointer in the desired direction. A rubber band box shows how the object will be zoomed.

Click the left mouse button.

-or-

Move the cursor to a node on the highlighting box.

Press **Alt+Ins**.

Use the arrow keys to move the pointer to the desired location.

Set a point.

Moving the zoom center

Select the object to be zoomed.

Choose the Select Zoom command from the Edit menu.

Set a point on the center point mark in the highlighting box.

Move the pointer in the desired direction.

Click the left mouse button.

See Also: [Select Zoom command](#)

Understanding Dimension Basics

There are several options and characteristics that must be understood when using the dimension commands.

Extension line

Extension Line Overshoot

Gap

Dimension line

Terminator or arrowhead

Dimension text

Tolerancing information

Dimensioning Lines

[Drawing Horizontal Dimensions](#)

[Drawing Vertical Dimensions](#)

[Drawing Angle Dimensions](#)

[Drawing Chamfered Dimensions](#)

[Drawing Baseline Dimensions](#)

[Drawing Extended Dimensions](#)

[Drawing Progressive Dimensions](#)

Drawing Horizontal Dimensions

Choose the Dimension command.

Choose the Horizontal button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the beginning of the dimension.

Set a point for the end of the dimension.

Set a point for the location of the text information.

Drawing horizontal dimensions without extension lines

Choose the Dimension, distance only command.

Choose the Horizontal button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the beginning of the dimension.

Set a point for the end of the dimension.

Set a point for the location of the text information.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Drawing Vertical Dimensions

Choose the Dimension command.

Choose the Vertical button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the beginning of the dimension.

Set a point for the end of the dimension.

Set a point for the location of the text information.

Drawing vertical dimensions without extension lines

Choose the Dimension, distance only command

Choose the Vertical button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the beginning of the dimension.

Set a point for the end of the dimension.

Set a point for the location of the text information.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Drawing Angle Dimensions

Choose the Dimension Angle command.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the center, or vertex, of the angle to be measured.

Set a point for the beginning of the angle to be dimensioned.

Set a point for the end of the angle to be measured.

Set a point for the location of the text information.

Drawing dimensions aligned to any angle

Choose the Dimension command.

Click the Align to any angle button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the beginning of the dimension.

Set a point for the end of the dimension.

Set a point for the location of the text information.

Drawing angled dimensions without extension lines

Choose the Dimension, distance only command.

Choose the Align to any angle button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the beginning of the dimension.

Set a point for the end of the dimension.

Set a point for the location of the text information.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Drawing Chamfered Dimensions

Choose the Dimension Chamfer command.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the chamfered line.

Set a point for the extension line.

Set a point for the end of the extension line.

Set a point for the text location.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Dimensioning Coordinate Points

Choose the Coordinate Dimension command.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the origin, or reference point.

Set a point for the point to be measured.

Set a point for the text location.

Press **Enter**.

See Also: [Changing Dimension Options](#)

Dimensioning Circles

[Drawing Diameter Dimensions with Pre-Defined Extension Line](#)

[Drawing Diameter Dimensions Outside Circle](#)

[Drawing Diameter Dimensions Inside Circle](#)

[Drawing Radius Dimensions with Pre-Defined Extension Line](#)

[Drawing Radius Dimensions Outside Circle](#)

[Drawing Radius Dimensions Inside Circle](#)

Drawing Diameter Dimensions with Pre-Defined Extension Line

Choose the Diameter Dimension command.

To have the dimension drawn with a pre-defined extension line, choose the Pre-defined Extension button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the circle or arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Diameter Dimensions Inside Circle

Choose the Diameter Dimension command.

To have the dimension arrowheads drawn inside the circle, choose the Dimension Inside Circle button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the circle or arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Diameter Dimensions Outside Circle

Choose the Diameter Dimension command.

To have the dimension text and arrowheads drawn outside the circle, choose the Dimension Outside Circle button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the circle or arc to be dimensioned.

Set a point for the direction of the extension line.

Set a point for the location of the text information.

Press **Enter**.

See Also: [Changing Dimension Options](#)

Drawing Radius Dimensions with Pre-Defined Extension Line

Choose the Radius Dimension command.

To have the dimension drawn with a pre-defined extension line, choose the Pre-defined Extension button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the circle or arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Radius Dimensions Inside Circle

Choose the Radius Dimension command.

To have the dimension arrowheads drawn inside the circle, choose the Dimension Inside Circle button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the circle or arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Radius Dimensions Outside Circle

Choose the Radius Dimension command.

To have the dimension text and arrowheads drawn outside the circle, choose the Dimension Outside Circle button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the circle or arc to be dimensioned.

Set a point for the direction of the extension line.

Set a point for the location of the text information.

Press **Enter**.

See Also: [Changing Dimension Options](#)

Dimensioning Arcs

[Drawing Arc Dimensions](#)

[Drawing Diameter Dimensions with Pre-Defined Extension Line](#)

[Drawing Diameter Dimensions Outside Arc](#)

[Drawing Diameter Dimensions Inside Arc](#)

[Drawing Radius Dimensions with Pre-Defined Extension Line](#)

[Drawing Radius Dimensions Outside Arc](#)

[Drawing Radius Dimensions Inside Arc](#)

Drawing Arc Dimensions

Choose the Dimension Arc command.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the arc to be measured.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Diameter Dimensions Outside Arc

Choose the Diameter Dimension command.

To have the dimension text and arrowheads drawn outside the arc, choose the Dimension Outside Arc button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Diameter Dimensions Inside Arc

Choose the Diameter Dimension command.

To have the dimension arrowheads drawn inside the arc, choose the Dimension Inside Arc button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Radius Dimensions with Pre-Defined Extension Line

Choose the Radius Dimension command.

To have the dimension drawn with a pre-defined extension line, choose the Pre-defined Extension button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Radius Dimensions Inside Arc

Choose the Radius Dimension command.

To have the dimension arrowheads drawn inside the arc, choose the Dimension Inside Arc button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the arc to be dimensioned.

Set a point for the location of the text information.

See Also: [Changing Dimension Options](#)

Drawing Radius Dimensions Outside Arc

Choose the Radius Dimension command.

To have the dimension text and arrowheads drawn outside the arc, choose the Dimension Outside Arc button in the Command Line.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point on the arc to be dimensioned.

Set a point for the direction of the extension line.

Set a point for the location of the text information.

Press **Enter**.

See Also: [Changing Dimension Options](#)

Changing Dimension Options

[Angular Text Format](#)

[Angular Text Precision](#)

[Dimension Arrowhead Size](#)

[Dimension Arrowhead Type](#)

[Dimension Offset](#)

[Dimension Prefix](#)

[Dimension Suffix](#)

[Dimension Text Location](#)

[Dimension Text Orientation](#)

[Dimension Text Size](#)

[Dimension Tolerance Text Size](#)

[Fixed Length](#)

[Linear Text Format](#)

[Linear Text Precision](#)

[No Dimension Tolerance](#)

[Overshoot Length](#)

[Position of Dimension Arrows](#)

[Positive and Negative Dimension Tolerance](#)

[Positive or Negative Dimension Tolerance](#)

[Saving a Vector Dimension](#)

[Variable Length](#)

Drawing Baseline Dimensions

Choose the Baseline Dimension command.

Choose the Horizontal button to draw horizontal dimensions.

Choose the Vertical button to draw vertical dimensions.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the first measurement position.

Set a point for the second measurement position.

Set a point for the final measurement position.

Set a point for the location of the dimension text information.

Press **Enter**.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Drawing Extended Dimensions

Choose the Extended Dimension command.

Choose the Horizontal button to draw horizontal dimensions.

Choose the Vertical button to draw vertical dimensions.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the first measurement position.

Set a point for the second measurement position.

Set a point for the final measurement position.

Set a point for the location of the dimension text information.

Press **Enter**.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Drawing Progressive Dimensions

Drawing successive dimensions

Choose the Progress Dimension command.

Choose the Horizontal button to draw horizontal dimensions.

Choose the Vertical button to draw vertical dimensions.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the base point.

Set a point for the first measurement position.

Set a point for the final measurement position.

Set a point for the location of the dimension text information.

Press **Enter**.

Drawing successive dimensions

Choose the Progressive Radius Dimension.

To have the dimension drawn with the same options as another dimension in the drawing, choose the Same as button. Then set a point on the dimension in the drawing.

Set a point for the base point.

Set a point for the first measurement position.

Set a point for the final measurement position.

Set a point for the text location.

Press **Enter**.

See Also: [Dimensioning Lines](#), [Changing Dimension Options](#)

Changing Dimension Layer and Color

Changing the dimension layer

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Choose the Layer box in the Info Box.
- Choose the new layer number.

Changing the dimension color

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Choose the Color box in the Info Box.
- Choose the new color from the Info Color box.
- Choose the OK button.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Changing Dimension Position

Select the dimension to be changed.
Choose Show Info Box from the View menu.
Click the Position box in the Info Box.
Choose the new position.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Changing Arrowhead Type

Select the dimension to be changed.
Choose Show Info Box from the View menu.
Click the Arrow Type box in the Info Box.
Choose the new arrowhead.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Changing Dimension Location and Orientation

Changing the dimension location

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Click the Location box in the Info Box.
- Choose the new dimension location.

Changing the dimension orientation

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Click the Orientation box in the Info Box.
- Choose the new dimension orientation.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Changing Dimension Size and Tolerance

Changing the dimension size

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Click the Text Size box in the Info Box.
- Type the new text size.

Changing the dimension tolerance

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Click the Single Tolerance box or the Double Tolerance box in the Info Box.
- Type the new values for the tolerance.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Changing Dimension Prefix and Suffix

Changing the dimension prefix

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Choose the Prefix box in the Info Box.
- Type the new prefix.

Changing the dimension suffix

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Choose the Suffix box in the Info Box.
- Type the new suffix.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Changing Colors

Changing drawing color

Choose the Show Color Table option in the Toolbox.
Choose the color from the color table.

Changing color of selected objects

Select the object to be changed.
Choose the Show Color Table option in the Toolbox.
Choose the color.
Choose the Apply button.

Changing color of entity in Info Box

Select the entity to be changed.
Choose Show Info Box in the View menu.
Choose the Color button.
Choose the color.
Choose the OK button.

Setting color same as an entity

Choose the Show Color Table option in the Toolbox.
Choose the Same as box.
Select the entity with the color you want.
Select the entity to be changed.
Choose the Apply button.

Removing the color table

Choose the window Control-menu in the upper right corner of the menu.
-or-
Choose the Show Color Table option from the Options menu.

See Also: [Changing Entity Color](#)

Hatching a Selected Area

Choose the Hatch command from the Draw menu.

Type the size of the hatch pattern in the Hatch Scale box in the Command Line.

Type the angle of the hatch pattern in the Hatch Angle box in the Command Line.

Select the hatch pattern in the Hatch Pattern box in the Command Line.

To have the hatch pattern drawn with the same options as another hatch pattern in the drawing, choose the Same as button. Then set a point on the hatch pattern in the drawing.

Set points around the area to be hatched.

Press **Enter**.

See Also: [Hatch Patterns](#), [Selecting a Hatch Pattern](#)

Hatching a Line Entity

Choose the Hatch Line command from the Draw menu.

Select the line to be hatched.

Type the size of the hatch pattern in the Hatch Scale box in the Command Line.

Type the angle of the hatch pattern in the Hatch Angle box in the Command Line.

Select the hatch pattern in the Hatch Pattern box in the Command Line.

To have the hatch pattern drawn with the same options as another hatch pattern in the drawing, choose the Same as button. Then set a point on the hatch pattern in the drawing.

Press **Enter**.

See Also: [Hatch Patterns](#), [Selecting a Hatch Pattern](#)

Hatching an Enclosed Area

Choose the Hatch Fill command from the Draw menu.

Type the size of the hatch pattern in the Hatch Scale box in the Command Line.

Type the angle of the hatch pattern in the Hatch Angle box in the Command Line.

Select the hatch pattern in the Hatch Pattern box in the Command Line.

If the entity is selected before the command is chosen, you can click the "Selection Only" box to limit the hatch boundary to only those lines that are selected.

To have the hatch pattern drawn with the same options as another hatch pattern in the drawing, choose the Same as button. Then set a point on the hatch pattern in the drawing.

Set a point inside the area to be hatched.

See Also: [Hatch Patterns](#), [Selecting a Hatch Pattern](#)

Editing a Hatch Pattern

Erasing a pattern

Activate the Select Mode.

Move the pointer to the hatched area.

Select the hatch pattern.

-or-

Choose the Entity Select command from the Edit menu.

Choose Hatch.

Choose the OK button.

Choose the Erase command from the Edit menu.

Note: Choosing the Entity Select command then the Edit command erases all hatch patterns in the drawing.

See Also: [Hatch Patterns](#), [Selecting a Hatch Pattern](#)

Setting Layer Options

Choose the Layer Setup command from the Options menu.
Type the layer number in the Number box.
Type a name for the layer in the Name box.
Choose the Multilayer box to manipulate entities in the layers.
The Layer Status box displays information for the layers.
Choose the Commands button to choose one of the layer commands.
Choose the OK button.

Current layer

When you create a new drawing, the entities are added to the current layer. The current layer is shown in the Coordinate Bar at the top of the screen. The current layer defaults to Layer 1 unless otherwise specified.

Visible/Invisible layers

Each layer can be set to visible or invisible status. A visible layer is displayed on the screen. An invisible layer is not visible on the screen. A visible layer can be edited or can be "uneditable." An invisible layer cannot be editable.

See Also: [Setting Invisible Layers](#), [Setting Visible Layers](#), [Setting Uneditable Layers](#), [Setting Editable Layers](#), [Setting Current Layer](#), [Manipulating Current Layer Only](#)

Manipulating Current Layer Only

Choose the Layer Setup command from the Options menu.
Type the current layer number in the Number box.
Type a name for the layer in the Name box.
Choose the Multilayer box to disable it
Press **Enter**.

See Also: [Setting Layer Options](#)

Setting Current Layer

Select the Layer box in the Coordinate Bar.
Choose the arrow button

-or-

Press **Alt+down arrow**.
Choose the layer number you want to be active.

See Also: [Setting Layer Options](#)

Setting Invisible Layers

Choose the Layer Setup command in the Options menu.
Double-click the layer in the Layer Status box.

-or-

Press **Tab**.

Choose the layer.

Then press **Alt+Shift+Ins** in the Layer Status box.

Choose the Visible box to disable it.

Press **Enter**.

See Also: [Setting Visible Layers](#), [Setting Layer Options](#)

Setting Visible Layers

Choose the Layer Setup command in the Options menu.
Double-click the layer in the Layer Status box.

-or-

Press **Tab**.

Choose the layer.

Then press **Alt+Shift+Ins** in the Layer Status box.

Choose the Visible box to enable it.

Press **Enter**.

See Also: [Setting Invisible Layers](#), [Setting Layer Options](#)

Setting "Uneditable" Layers

Choose the Layer Setup command in the Options menu.
Double-click the layer in the Layer Status box.

-or-

Press **Tab**.

Choose the layer.

Then press **Alt+Shift+Ins** in the Layer Status box.

Choose the Editable box to disable it.

Press **Enter**.

See Also: [Setting Editable Layers](#), [Setting Layer Options](#)

Setting Editable Layers

Choose the Layer Setup command in the Options menu.
Double-click the layer in the Layer Status box.

-or-

Press **Tab**.

Choose the layer.

Then press **Alt+Shift+Ins** in the Layer Status box.

Choose the Editable box to enable it.

Press **Enter**.

See Also: [Setting Uneditable Layers](#), [Setting Layer Options](#)

Changing Layer Colors and Line Style

Changing color of layer

- Choose the Set Layer Color command in the Options menu.
- Choose the new color for the current layer.
- Choose the OK button.

There are 13 line types available.

Changing the line style in layers

- Choose the Set Layer Line Style command in the Options menu.
- Choose the line type in the Line Type box.
- To change the scale of the line, type the value in the Scale box.
- To change the width of the line, type the value in the Width box.
- Choose the OK button.

See Also: [Manipulating Layers](#)

Saving and Deleting Layers

Saving layer

Choose the layer to be saved in the Layer box in the Coordinate Bar.
Choose the Save Layer command from the Options menu.
Type the file name in the File Name box.
Choose the OK button.

Deleting a layer

Choose the layer to be deleted in the Layer box in the Coordinate Bar.
Choose the Delete Layer command in the Options menu.

See Also: [Manipulating Layers](#), [Saving Layers](#), [Editing Layers](#), [Deleting a Layer](#)

Selecting and Moving Layers

Selecting an entire layer

Choose the layer to be selected in the Layer box in the Coordinate Bar.
Choose the Select Layer command in the Options menu.

Moving entities to different layers

Choose the Move Layer command in the Options menu.
Choose the layer number the current layer is to be moved to in the selection box.
Choose the OK button.

See Also: [Manipulating Layers](#), [Editing Layers](#), [Selecting Layers](#)

Separating Colors into Layers

Choose the Separate by Color command in the Layer secondary menu in the Options menu.

See Also: [Separate by Color command](#), [Manipulating Layers](#), [Editing Layers](#)

Installing Symbols

DesignCAD comes with over 400 pre-drawn symbols you can use in your drawings. These symbols are found in separate libraries.

You can install symbols by running SETUP, the DesignCAD installation program.

Installing symbols

If Windows is not already running, enter **WIN** at the DOS prompt.

-or-

If Windows is running, close any open applications.

Insert DesignCAD disk #1 into the correct drive.

Choose Run from the File menu.

Enter **A:SETUP** or **B:SETUP**.

To accept the path that Setup proposes, choose the Continue button.

-or-

To choose your own directory, type a new path in the Install To box, then choose the Continue button.

Choose the Install Symbols box to enable it.

Choose the Install DesignCAD box to disable it.

Choose the Install Sample Drawings box to disable it.

Select the symbol libraries you want to install.

Follow instructions.

When Setup is complete, choose the OK button.

See Also: [Loading Symbols](#), [Creating Symbols](#), [Symbols Libraries](#)

Manipulating Symbols

In DesignCAD, symbols are defined as a single entity. That is, the entire symbol is selected, moved, edited, and erased with DesignCAD commands.

To change a single line or entity in a symbol, you must first "explode" the symbol with the Symbol Explode command.

Exploding a symbol

Choose the Symbol Explode command from the Edit menu.

Set a point on the symbol to be exploded.

Press **Enter**.

See Also: [Using Symbols](#)

Creating Symbols

It is easy to create your own custom symbols in DesignCAD. Simply follow these rules:

Create the symbol in Layer 0. When you retrieve a drawing each layer normally remains in the same layer. However, when a drawing or symbol is saved in Layer 0 it is retrieved into the current layer. This lets you make a symbol and load the symbol into any layer of the drawing.

Set one or two points for the drawing handles with the [Set Handles command](#) before saving the drawing. When you save a drawing DesignCAD defines these points as reference points. These "reference points" determine the size and scale of a drawing or symbol when it is retrieved.

Set the correct Drawing Units before saving the symbol. The Drawing Units determine the size of the symbol. If a symbol is retrieved using a single point it will be scaled according to the Drawing Units of the current drawing.

Creating your own symbols

Choose **0** in the Layer box in the Coordinate Bar.

Draw the symbol.

Choose the Save command to save the symbol.

See Also: [Using Symbols](#)

Defining a Group

Select the entities to be defined as a Group.
Choose the Group Define command from the Edit menu.
Press **Enter**.

See Also: [Defining and Undefining Groups](#), [Group command](#), ["Undefining" a Group](#)

"Undefining" a Group

"Exploding" a group

Select the Group to be exploded.

Choose the Group Explode command from the Edit menu.

Press **Enter**.

See Also: [Defining and Undefining Groups](#), [Group Explode command](#), [Defining a Group](#)

Combining Two Drawings

DesignCAD allows you to merge more than one drawing into a single drawing. DesignCAD opens the first drawing, then merges the second drawing on top of it. The Drawing Units of the second drawing are matched to those of the drawing already on the screen.

Merging a drawing

Choose the Merge command from the File menu.

Choose the file you want to merge with the current drawing from the File Name box.

Set a point for the lower left corner of the drawing.

Set a point for the lower right corner of the drawing.

Set a point for the upper left corner of the drawing.

Set a point for the upper right corner of the drawing.

See Also: [Merge command](#), [Combining Drawings](#)

Opening a New Window

Opening multiple views

Choose the New Window command from the Window menu.

Drawing commands effect all views of the drawing.

Making a drawing view active

From the Window menu, choose the name of the drawing view you want to make active.

See Also: [Closing a Window](#), [Arranging All Open Drawing Windows](#), [Opening Views](#), [Opening Multiple Views of a Drawing](#)

Manipulating the Views

Different views of the drawing window become icons when they are minimized.

Minimizing drawing windows

Choose the Minimize button in the upper right corner of the drawing window.

You can move these icons around the screen. The Arrange Icons command straightens any icons on the screen and places them along the bottom of the screen.

Arranging Icons

Choose the Arrange Icons command from the Window menu.

See Also: Opening a New Window, Arranging All Open Drawing Windows

Changing Coordinate Bar Information

Showing Angle and Distance coordinates

Choose Show Polar Coordinates from the View menu.

The Coordinate Bar will change to show the Angle and Distance coordinates.

Showing X, Y, Delta X, Delta Y coordinates

If a check mark is next to Show Polar Coordinates in the View menu, choose the command.

The Coordinate Bar will change to show the X, Y, Delta X, and Delta Y coordinates.

See Also: [Coordinate Bar](#)

Setting the Display Grid

The display grid helps you set points and draw entities. This grid does not appear when the drawing is printed.

Enabling the Display Grid

Choose the Display Grid option from the Options menu.

See Also: Display Grid command, Snap Grid command

Setting the Grid Sizes

Choose the Grid Options command from the Options menu.
Type the display grid size in Drawing Units in the Display Grid Size box.
Type the display grid type in the Display Grid Type box.
Type the snap grid size in Drawing Units in the Snap Grid Size box.
Choose the Enable Display Grid box to enable the display grid.
Choose the Enable Snap Grid box to enable the snap grid.
Choose the OK button.

See Also: [Setting the Display Grid Size](#), [Setting the Snap Grid](#)

Displaying Cursor Crosshairs

The Crosshair option turns the [cursor crosshairs](#) on or off. These crosshairs can be used to line up points on the drawing screen.

Displaying the cursor crosshairs

Choose the Crosshair option from the Options menu.

See Also: [Crosshair command](#)

Measuring the Units

The Units command sets the units of measurement for the drawing. These units of measurement are called Drawing Units. Drawing Units can be inches, centimeters, feet, miles, any unit of measurement that makes sense for the drawing.

Setting Drawing Units

Choose the Units command from the Options menu.

Set two points a certain distance apart.

Type the distance these points represent in the New distance box.

The original distance is shown in the Original distance box.

Choose the OK button.

See Also: [Units command](#)

Compatible File Formats

Import File Formats

ASCII text files

AutoCAD DXF format files

DesignCAD 2-D ASCII files

DesignCAD 2-D Binary files

DesignCAD 3-D ASCII files

DesignCAD 3-D Binary files

Hewlett Packard plotter (HPGL) files

IGES (Initial Graphics Exchange Standard) format files

X, Y coordinate files

Export File Formats

IGES (Initial Graphics Exchange Standard) format files

AutoCAD DXF format files

PostScript format files

WordPerfect graphics files

Importing Drawings

DesignCAD allows you to import drawings from other drawing file formats into DesignCAD.

Importing DXF file formats

- Choose Import DXF from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing IGES file formats

- Choose Import IGES from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing HPGL file formats

- Choose Import HPGL from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing ASCII text file formats

- Choose Import ASCII from the File menu.
- Choose the file from the File Name box.
- Choose the OK button.

Importing XY file formats

- Choose Import XY from the File menu.
- Choose the Straight Line button to import straight lines.
- Choose the Curved Line button to import curved lines.
- Choose the Small Squares button to import small squares.
- Choose the Small Circles button to import small circles.
- Choose the Mark Size box to set the size of point marks.
- Choose the Color button to set the color.
- Choose the color from the Line Color box.
- Choose the OK button.
- Choose the OK button.
- Choose the file name from the File Name box.
- Choose the OK button.

See Also: [DXF Format Files](#), [IGES Format Files](#), [HPGL Format Files](#), [DesignCAD 2-D ASCII Format Files](#), [XY Format Files](#), [Opening a File Created by Another Application](#)

Exporting Drawings

DesignCAD allows you to transfer drawing files to other programs.

Exporting DXF file formats

Display the drawing you want to export on the screen.
Choose the Export DXF command from the File menu.

Exporting IGES file formats

Display the drawing you want to export on the screen.
Choose the Export IGES command from the File menu.

Exporting WordPerfect file formats

Display the drawing you want to export on the screen.
Choose the Export WordPerfect command from the File menu.

See Also: [DXF Format Files](#), [IGES Format Files](#), [PostScript Format Files](#), [WordPerfect Format Files](#), [DesignCAD Drawing File Formats](#)

ASCII File Format

DesignCAD drawing files can be saved with a .DC2 file extension by choosing the ".DC2" extension in the Save As command. This is an ASCII file format, where the data is written in character format. Each "record" in the file is actually a line in a text file.

There are four types of records, or lines, in the file:

Header Line

Entity Line

Point Line

String Line

See Also: [Understanding System Parameters](#), [Point Definitions](#)

Header Line

The header line tells, among other things, how many entity lines are to follow. The header line contains up to seven numbers:

- 1 -- The minimum X coordinate in the file
- 2 -- The minimum Y coordinate in the file
- 3 -- The horizontal length of the drawing
- 4 -- The vertical height of the drawing
- 5 -- A zero for compatibility with DesignCAD 2-D
- 6 -- A zero for compatibility with DesignCAD 3-D
- 7 -- Version number

The coordinates are in arbitrary units. The Drawing Units Size -- entity type 40 -- can be used to determine the "real world" coordinates or the size of one Drawing Unit.

See Also: [Understanding System Parameters](#), [Point Definitions](#)

Entity Line

The entity line tells the type of drawing entity and how many points are to follow. The entity line contains six or twelve numbers:

- 1 -- The type of entity:
 - 0 -- Handles
 - 1 -- Line
 - 2 -- Oval
 - 3 -- Text
 - 4 -- Curve
 - 5 -- Arc
 - 11 -- Bezier Curve
 - 15 -- Attribute
 - 16 -- Circle, Circular Arc
 - 17 -- Hatch
 - 22 -- Text Arc
 - 23 -- Layer Names
 - 25 -- Symbol
 - 31 -- Plane
 - 32 -- Grid
 - 33 -- Grid line
 - 70 -- Point
 - 74 -- Dimension Angle
 - 75 -- Dimension
 - 76 -- Dimension Diameter and Radius
 - 77 -- Dimension Arc
 - 78 -- Dimension Progressive Radius
 - 79 -- Dimension Progress
 - 80 -- Dimension Chamfer
 - 81 -- Dimension Coordinate
 - 90 -- Bitmap Image
2. The number of points to follow (0-200)
-or-
The layer number (0-255) if entity is 21
3. The line pattern scale (1 -- normal)
-or-
The hatch pattern scale
4. The line width
5. The line type (0-8)
6. The color (0-255)
7. Reserved -- Optional to dimension entities
8. Dimension Format -- Optional to dimension entities
9. Group ID -- Optional to entity grouping
10. Red Color (0-255)
11. Green Color (0-255)
12. Blue Color (0-255)

See Also: [Understanding System Parameters](#), [Point Definitions](#)

Point Line, ASCII

The Point line contains point coordinates. Each line contains two numbers: X coordinate and Y coordinate of the point on the screen. There is one point per line. These values will change when the drawing is retrieved according to the size, location, and angle of the drawing when it is loaded.

There can be no more than 200 points per entity. A line or curve entity with more than 200 points may be used by splitting it into two or more entities.

If the entity type is 40 or 41, then the number of points represents the number of lines that follow that entity. Each line may have more than one value, and these values do not represent points.

The point line can contain an X, Y, and Z coordinate. However, the Z coordinate is ignored in DesignCAD 2-D.

See Also: [Understanding System Parameters](#), [Point Definitions](#)

String Line

The string line contains a string of text. There is one string per line. Strings are found in entities drawn with the Text, Text Arc, Attribute, Hatch, Symbol, and dimension commands.

There can be no more than 80 characters per string length.

The string line must be terminated with a carriage return character. Trailing blanks will be used to determine the string length.

See Also: [Understanding System Parameters](#), [Point Definitions](#)

Understanding System Parameters

The system parameters follow entity type 41. The system parameters are saved with each drawing when the Save Parameters With Drawing box is selected in the Miscellaneous Options dialog box. These consist of:

Line 1:

- 1 -- Drawing Unit size
- 2 -- Large cursor step size
- 3 -- Small cursor step size

Line 2:

- 1 -- Reserved
- 2 -- Display grid type
- 3 -- Display grid enable
- 5 -- Snap grid enable
- 6 -- Dimension text format
- 7 -- Dynamic dimension enable
- 8 -- Tolerance enable
- 9 -- Dimension arrowhead type
- 10 -- Dimension text location
- 11 -- Dimension precision
- 12 -- Fill wide lines enable
- 13 -- Angular dimension type
- 14 -- Angular dimension precision
- 15 -- Point mark

Line 3:

- 1 -- Display grid size
- 2 -- Default text size
- 3 -- Default text angle
- 4 -- Drawing Units per inch on output
- 5 -- Snap grid size

Line 4:

- 1 -- (Not used in this version) 0
- 2 -- (Not used in this version) 1.0
- 3 -- (Not used in this version) 1.0
- 4 -- (Not used in this version) 1.0
- 5 -- (Not used in this version) 1.0

Line 5:

- 1 -- "Simplex2.vfn" - (Not used in this version)

Line 6:

- 1 -- (blank line) (Not used in this version)

Line 7 (optional):

- 1 -- dimension text size
- 2 -- dimension text horizontal scale
- 3 -- dimension text slant
- 4 -- dimension tolerance text size
- 5 -- dimension extension line overshoot

Line 8 (optional):

- 1 -- dimension extension line gap
- 2 -- dimension extension line length
- 3 -- dimension arrowhead scale
- 4 -- dimension balloon size
- 5 -- baseline dimension offset

Line 9 (optional):

- 1 -- dimension layer
- 2 -- dimension type
- 3 -- angular dimension type
- 4 -- dimension precision
- 5 -- angular dimension precision
- 6 -- dimension tolerance enabled/disabled

See Also: [ASCII File Format](#)

Point Definitions

Entity number	Entity type	Points
0	Drawing handle	1 -- Primary handle 2 -- Secondary handle(optional)
1	Line	1-200: Points for line
2	Ellipse	1 -- Center 2 -- Axis 3 -- Ellipse pass through 4 -- Minimum X, Y position 5 -- Maximum X, Y position
3	Text	1 -- Lower left corner 2 -- Lower right corner
4	Curve	1-200: Points for curve
5	Elliptical Arc	1 -- Center 2 -- Beginning point 3 -- End point 4 -- Minimum X, Y (optional) 5 -- Maximum X, Y (optional) 6 -- Radius 1 (optional) 7 -- Radius 2 (optional)
11	Bezier Curve	1 -- Beginning 2 -- First control point 3 -- Second control point 4 -- End
15	Attribute	1 -- Text location
16	Circle	1 -- Center 2 -- Beginning 3 -- End 4 -- Minimum X, Y (optional) 5 -- Maximum X, Y (optional)
17	Hatch	1-200: Area to be hatched
22	Text Arc	1 -- Center 2 -- Beginning 3 -- End 4 -- Beginning point for text 5 -- Minimum X, Y (optional) 6 -- Maximum X, Y (optional)
23	Layer Names	None One name for each "point"
24	Arrow	1-198: 1-n -- Line n -- Position of arrowhead
26	Symbol	1 -- Lower left corner (primary handle) 2 -- Lower right corner (secondary handle) 3 -- Upper right corner 4 -- Upper left corner 5 -- Primary handle 6 -- Secondary handle
31	Plane	1-200: 1 -- First vertex 2 -- Second vertex 3- n -- Vertex plane First and last - Same
32	Grid	None Number of points on Command Line tells how many grid line follow.
33	Grid Line	None

		Number of points on Command Line tells how many grid line follow.
70	Point Mark	1 -- Mark location
74	Dimension Angle	1 -- Center 2 -- Beginning 3 -- End 4 -- Beginning first dimension line 5 -- End first dimension line 6 -- Beginning second dimension line 7 -- End second dimension line 8 -- Text height, arrowhead size
75	Dimension	1 -- Beginning distance to be measured 2 -- End distance to be measured 3 -- Beginning extension line 4 -- End extension line 5 -- Beginning dimension line 6 -- End dimension line 7 -- Text height, arrowhead size
76	Dimension Diameter/Radius	1 -- Beginning dimension line 2 -- End dimension line 3 -- Beginning distance to be measured 4 -- End distance to be measured 5 -- Beginning extension line 6 -- End extension line 7 -- Text height, arrowhead size
77	Dimension Arc	1 -- Center 2 -- Beginning 3 -- End 4 -- Beginning extension line 5 -- End extension line 6 -- Text height, arrowhead size
78	Dimension Radius Progress	1 -- Base 2 -- Beginning of extension arc 3 -- End of extension arc 4 -- Beginning dimension line 5 -- End dimension line 6 -- Text height, arrowhead size
79	Dimension Progress	1 -- Base 2 -- Beginning dimension line 3 -- End dimension line 4 -- Beginning first extension line 5 -- End first extension line 6 -- Beginning second extension line 7 -- End second extension line 8 -- Text height, arrowhead size
80	Dimension Chamfer	1 -- Beginning distance to be measured 2 -- End distance to be measured 3 -- Beginning dimension line 4 -- End dimension line 5 -- Text height, arrowhead size
81	Dimension Coordinate	1 -- Base 2 -- Beginning extension line 3 -- End extension line 4 -- Text size, arrowhead type
90	Bitmap Image	1 -- Upper left corner 2 -- Upper right corner 3 -- Lower right corner

- 4 -- Lower left corner
- 5 -- Lower left corner
- 6 -- Upper right corner

See Also: [ASCII File Format](#)

Binary File Format

The DesignCAD .DW2 file is a binary file containing the following information:

Header Line

Entities

See Also: DesignCAD 2-D Binary Files

Binary Header Line

The header line is the following format (sizes are in bytes):

Size	Description
2	Drawing type: 1099 -- 2-D; 1199 -- 3-D
2	DesignCAD version number
2	Data type: 4 -- single precision; 8 -- double precision

See Also: [Binary File Format](#)

Binary Entity

The entities are in the following format:

Size	Description
2	Entity code
2	Number of bytes in entity (Not including entity code and this field)
n	Information for the entity

Drawing size entity The first entity in the drawing is the "Drawing Size" entity. It has the following format:

Size	Description
2	Drawing Size Entity Code -- 1200
2	Number of bytes -- 16
4	X drawing minimum
4	Y drawing minimum
4	X drawing width
4	Y drawing height

Parameter entities The following parameter entities may appear in the drawing:

Entity	Size	Parameters
1300	2	Number of digits of accuracy
1301	2	Display attribute flag
1302	2	Font type
1303	2	Snap grid enable
1304	2	Beep type
1305	2	Current layer manipulation
1306	2	Dimension type
1307	2	Dynamic dimensioning enable
1308	4	Large cursor step size
1309	4	Drawing Unit scale
1310	4	Snap grid size
1311	4	Text size
1312	4	Text angle
1313	4	Drawing Units per inch on output
1314	4	Drawing Unit size
1328	3/color	Color Palette: R,G,B bytes for each color
1329	4	Grid size
1333	2	Display grid enable
1334	2	Display grid type
1335	4	Small cursor step size
1340	Variable	Layer names
1350	2	Arrowhead type
1351	2	Dimension tolerance
1352	2	Dimension type
1353	2	Angular dimension precision
1354	2	Dimension precision
1355	4	Dimension text position
1356	2	Wide lines fill
1357	2	Point mark
1359	4	Text horizontal scale
1360	4	Text slant

1365	Variable	Character font name
1366	4	Dimension text size
1367	4	Dimension text horizontal scale
1368	4	Dimension text slant
1369	4	Dimension tolerance text size
1370	4	Dimension extension overshoot length
1371	4	Dimension extension gap size
1372	4	Dimension extension length
1373	4	Dimension arrowhead size
1374	4	Balloon size
1375	4	Dimension baseline offset
1376	2	Default dimension layer
1377	2	Default dimension color
1378	4	Dimension arrowhead scale

Drawing entities The drawing entities are in the following format:

Size	Description
2	Entity number
2	Number of bytes in entity attributes section

Attributes section

Size	Description
2	Number of points
6	Color -- Red, green, and blue values
2	Solid ID
2	Group number (0 if not a Group)
2	Line type
4	Line width
4	Line type style

Points section

Size	Description
n*4	X coordinate for all points in the entity
n*4	Y coordinate for all points in the entity
2	Number of bytes of text -- only if command has string
n	Text -- only if command has string

See Also: [Binary File Format](#)

Creating Macros

Choose the Record Macro command from the File menu.
Type the name for the macro in the Record Macro Name box.
Choose the OK button.
Set a point for the startup cursor position.
Perform the actions you want to record.
Choose the Stop Recording Macro command.

Note: Macros should be saved to the directory where DesignCAD was installed so that they will appear in the Macro Options dialog box. The default directory is C:\DCAD.

See Also: [Recording Macro Options](#)

Materials List File Format

Attributes are listed in alphabetical order. The attribute string is terminated with a line feed and carriage return. The attribute can not be longer than 80 characters.

<u>Columns</u>	Contents
1	Quantity
2	Quantity
3	Quantity
4	Quantity
5	Quantity
6	Quantity
7	Comma
8	Attribute string

See Also: [Material List command](#), [Attribute command](#)

Saving a Drawing

Saving an existing drawing

Choose the Save command from the File menu.

Saving an unnamed drawing

Choose the Save As command from the File menu.

In the File Name box, type a filename. DesignCAD saves the drawing in the current drive and directory and automatically adds the default file extension .DW2 unless you specify another filename extension.

To save the drawing to a different directory, choose the directory in the Directories box.

To save the drawing in a different drive, choose the drive in the Drives box.

Choose the OK button.

Saving all open drawings

When you quit DesignCAD by choosing Exit from the File menu, DesignCAD closes all drawings and verifies that you want to save changes.

See Also: [Saving Drawings](#), [Saving your Drawings](#), [Saving a New, Unnamed Drawing](#)

Saving Selected Objects

Select the object to be saved.

Choose the Save Selection command from the File menu.

Type the drawing name in the File Name box.

Choose the OK button.

See Also: [Save Selection command](#), [Opening Drawings](#), [Guidelines for Naming Files](#)

Guidelines for Naming Files

DesignCAD saves drawings with any acceptable DOS filename. That is, the filename can be from one to eight characters long, followed by a period and a three-character filename extension. You can use any characters except spaces and the following characters: * ? , ; [] + = \ / : | < > . You cannot use a period except to separate the filename extension.

Filename extensions distinguish between types of files. For example, DesignCAD binary drawing files have the extension .DW2, DesignCAD ASCII drawing files have the extension .DC2, and backup files have the extension .BAK.

If you do not add a filename extension to drawing filenames, DesignCAD adds .DW2 by default.

DesignCAD assumes specific filename extension. If you use other extensions for your files, type the extension every time you type the filename.

DesignCAD uses the following filename extensions:

<u>Extension</u>	Meaning
.A2D	ASCII View file
.B2D	Binary View file
.BMP	Bitmap file
.DC2	ASCII file
.DCM	DesignCAD Macro file
.DLL	Resource file
.DW2	Binary file
.EXE	Program file
.HLP	Help file
.PAL	Color table file
.SYS	Program file

See Also: [Saving a New, Unnamed Drawing](#)

Setting Cursor Step Size

The Cursor category in the Options command sets the distance the cursor moves. There are two cursor step sizes, a large one and a small one. Pressing an arrow key moves the cursor the distance of the large cursor step size. Pressing an arrow key simultaneously with the **Shift** key moves the cursor the distance of the small cursor step size. If your keyboard has a numeric keypad, the number keys will move the cursor the large cursor step size when the **Num Lock** is off or the small cursor step size when the **Num Lock** is on.

Setting the cursor step size

- Choose the Options command from the Options menu.

- Choose the Cursor category.

- Type a new value for the Large Cursor Step Size.

- Type a new value for the Small Cursor Step Size.

- Choose the OK button.

See Also: [Using Arrow Keys](#)

Using a Mouse or Digitizer

If you have a mouse or digitizer, you can use it to move the cursor, or pointer. The pointer movement corresponds to the movement of the mouse or digitizer stylus.

See Also: [Keyboard, Mouse, and Digitizer](#)

Drawing Handles

A DesignCAD drawing can be saved with two reference points called "drawing handles."

The default drawing handles are normally the lower left and lower right corners of the screen. To specify handles other than these, select the Set Handles command or Set Handles box. Then, set a point for the primary handle, or lower left corner. A second point can be set for the secondary handle, or lower right corner.

Highlight Bar

The highlight bar shows you which command or option is being changed.

Point Set command

Use the Point Set command to set a point at the current cursor location.

Shortcuts

Mouse: left mouse button

Digitizer: first button

Keys: **Ins**

See Also: [Choosing Point Commands](#), [Points and Command Selection](#)

Geographic Center

The geographic center is the center of gravity of an entity.

See Also: [Center of Gravity command](#), [Center of Gravity Move command](#).

Enclose Mode

The Enclose mode is automatically activated when the Select Mode is active. The Enclose mode highlights only those entities entirely within the selection box.

Touch Mode

To activate the Touch mode, you must press **Ctrl** while dragging the mouse and selecting the entities. The Touch mode highlights any entity touched by the selection box.

Linear Text Format

There are four linear dimension text formats to choose from when drawing information in DesignCAD
These are:

Format	Example
Decimal	1.500
Fractional	1 1/2
Engineering	1' 6.75"
Architectural	1' 6-3/4"

Setting the linear text format

After a Dimension command is selected, choose the Dimension Text button in the Command Line.
Click the Format box.

Choose the arrow button to choose the dimension text format.

Choose the Same as button to have the dimension drawn with the same text format as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: Dimension command, Line Angle command

Linear Text Precision

You can choose from these linear precision formats in DesignCAD:

10000000
1000000
100000
10000
1000
100
10
1
1.0
1.00
1.000
1.0000
1.00000
1.000000
1.0000000

Setting the linear text precision

After a Dimension command is selected, choose the Dimension Text button in the Command Line.

Choose the Precision box.

Choose the arrow to choose the dimension precision.

Choose the Same as button to have the dimension text drawn with the same precision as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: Dimension Angle command, Line Angle-2 command

Angular Text Format

There are five angular dimension text formats to choose from when drawing information in DesignCAD. These are:

Format	Example
Decimal Degrees	45.0
Grads	50.0g
Radians	0.393rad
Degrees/Minutes/Seconds	45°0'0"
Surveyor's Units	N 45°0'0"E

Setting the angular text format

After the Dimension Angle command is selected, choose the Dimension Text button in the Command Line.

Choose the Format box.

Choose the arrow button to choose the angular text format.

Choose the Same as button to have the dimension drawn with the same text format as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Angle command](#), [Angle Between Two Lines command](#)

Angular Text Precision

You can choose these angular precision formats in DesignCAD:

1
1.0
1.00
1.000
1.0000
1.00000
1.000000

Setting the angular text precision

After the Dimension Angle command is selected, choose the Dimension Text button in the Command Line.

Choose the Precision box.

Choose the arrow to choose the angular text precision.

Choose the Same as button to have the dimension text drawn with the same precision as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: Dimension command, Angle, Distance Two Points command

Changing Dimension Format and Precision

Changing the dimension format

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Choose the Format box in the Info Box.
- Choose the new text format.

Changing the dimension precision

- Select the dimension to be changed.
- Choose Show Info Box from the View menu.
- Choose the Precision box in the Info Box.
- Choose the new text precision.

See Also: [Understanding Dimension Basics](#), [Changing Dimension Options](#)

Hewlett Packard Plotter (HPGL) Format Files

An HPGL file is created using Hewlett-Packard Graphics Language, or an HP Plotter file. You can create an HPGL file within DesignCAD by selecting an HP plotter in DesignCAD and then plotting to a disk file. Virtually any application that supports an HP plotter can be used to create HPGL files.

If your software does not allow you to plot to a disk file instead of directly to the plotter, there are software packages available that allow you to redirect plotter output from a serial or parallel port to a disk file.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

ASCII Text Files

An ASCII text file is created using a word processor. DesignCAD allows you to convert an ASCII text file to a different file format so that you can merge text with your drawings.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

XY Coordinate Files

An ASCII file of X, Y or X, Y, Z coordinates can be translated into a different file format. An X, Y coordinate file should have each set of coordinates on a separate line. Each number should be separated by one or more spaces, or a comma. Each line should be terminated by a carriage return and, optionally, a line feed character.

A blank line separating a series of X, Y or X, Y, Z coordinates will indicate the end of a line or curve and the beginning of another.

If the lines in a file contain three coordinates per line (X, Y, and Z), the file can be translated into a DesignCAD 3-D file. The Z coordinate may be ignored if you select an output file format that does not support three dimensions.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

AutoCAD DXF Format Files

A DXF file is created using AutoCAD. This conversion format makes DesignCAD compatible with many third-party software packages that utilize the DXF format.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

IGES (Initial Graphics Exchange Standard) Format Files

IGES (Initial Graphics Exchange Standard) Format Files

IGES is a standard format supported by most larger computer-aided design systems. This makes it possible to transfer DesignCAD drawings to other CAD systems.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

DesignCAD 2-D ASCII Format Files

DesignCAD 2-D ASCII files are created with DesignCAD 2-D and saved with the extension .DC2.

See Also: [ASCII File Format](#), [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

DesignCAD 2-D Binary Format Files

DesignCAD 2-D binary files are created with earlier versions of DesignCAD 2-D and saved with the extension .DW2.

See Also: [Binary File Format](#), [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

DesignCAD 3-D Binary Format Files

DesignCAD 3-D binary files are created with DesignCAD 3-D and saved with the extension .DW3.

See Also: [Binary File Format](#), [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

DesignCAD 3-D ASCII Format Files

DesignCAD 3-D binary files are created with DesignCAD 3-D and saved with the extension .DC3.

See Also: [ASCII File Format](#), [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

PostScript Format Files

Encapsulated PostScript format files are supported by PostScript printers, as well as many popular desktop publishing programs.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

WordPerfect Graphics Files

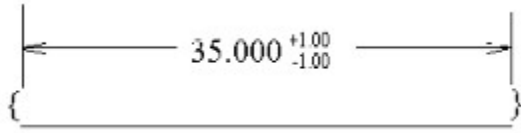
WordPerfect graphics files are saved with the extension .WPG and are supported by WordPerfect.

See Also: [Opening a File Created by Another Application](#), [Importing Drawings](#), [Exporting Drawings](#)

Set Handles Box

The Set Handles box allows you to set drawing handles before you open a drawing.

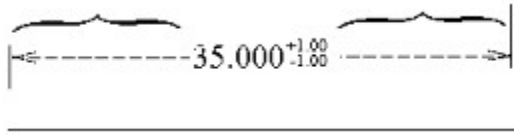
Gap



The extension line gap size is the distance between the points set for dimensioning and the beginning of the extension line.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

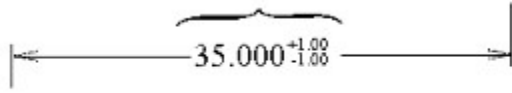
Dimension Line



The dimension line is the line drawn on either side of the dimension text to the extension line. The dimension line contains the dimension terminator.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

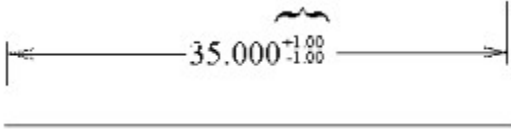
Dimension Text



The dimension text is the dimension distance, tolerance information, prefix, and suffix drawn inside the dimension line.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Tolerancing Information



The dimension tolerance is the positive and negative values given to the amount of variation allowed from the standard measurement.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Setting Dimension Offset

After the Baseline Dimension command is selected, choose the [Dimension Line button](#) in the Command Line.

Type the dimension offset in the Offset box.

Choose the Same as button to have the dimension drawn with the same offset as other dimensions.

Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set the points for the baseline dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Offset

The dimension offset determines the distance between the numbers drawn with the Dimension Baseline command.

Gap Size

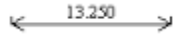
The "gap size" is the amount of space between the dimension information and the entity being dimensioned.

Dimension Text Location

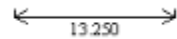
The dimension text can be drawn in these locations:



Center of the dimension line



Above the dimension line



Below the dimension line



Center and to the right of the dimension line



Above and to the right of the dimension line



Below and to the right of the dimension line



Center and to the left of the dimension line



Above and to the left of the dimension line



Below and to the left of the dimension line

Setting the dimension text location

After a Dimension command is selected, choose the Dimension Text button in the Command Line.

Choose the Location box.

Choose the arrow button to choose the dimension text location.

Choose the Same as button to have the dimension text drawn in the same location as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

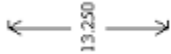
See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Text Orientation

Dimension text can be drawn with these orientations:



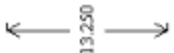
Alo. N -- Along the dimension line, normal



Per. N -- Perpendicular to dimension line, normal



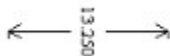
Hor. N -- Horizontal to dimension line, normal



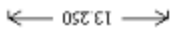
Ver. N -- Vertical to dimension line, normal



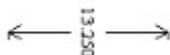
Alo. R -- Along dimension line, reversed



Per. R -- Perpendicular to dimension line, reversed



Hor. R -- Horizontal to dimension line, reversed



Ver. R -- Vertical to dimension line, reversed

Setting the dimension orientation

After a Dimension command is selected, choose the Dimension Text button in the Command Line.

Choose the Orientation box.

Choose the arrow button to choose the dimension text orientation.

Choose the Same as button to have the dimension text drawn with the same orientation as other dimensions. Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: [Dimension Options](#), [Changing Dimension Options](#)

Dimension Tolerance Text Size

The Dimension Tolerance Text Size is entered in Drawing Units in the Text Size box of the Dimension Tolerance button.

Setting the dimension tolerance text size

After a Dimension command is selected, choose the Dimension Tolerance button in the Command Line.

Type the tolerance text size in the Size box.

Choose the Same as button to have the dimension drawn with the same text size as other dimensions.

Then set a point on the existing dimension in the drawing.

Choose the Default button to reset the dimension options.

Set points for the dimension.

See Also: Dimension Options, Changing Dimension Options

Copying Objects to Clipboard

Select the objects to be copied to the Clipboard.
Choose the Copy command from the Edit menu.

See Also: [Copy command](#)

Copying Objects in a Row

Select the object to be copied.

Choose the Array command.

Type the number of copies in the No. of copies box in the Command Line.

Type the number of rows in the No. of rows box in the Command Line.

To have the first copy located on the original entity, choose the Use original as first copy box.

To force the handle to remain in the same place, choose the Fixed handle box in the Command Line.

Set a point for the handle of the first copy.

Set a point for the second copy. This point determines the direction and spacing of the objects.

Set a point for the location of the second row if more than one row was specified in the Command Line.

See Also: [Array command](#)

Copying Object in a Circle

Select the object to be copied.

Choose the Circular Array command.

Type the number of copies in the No. of copies box in the Command Line.

Type the angle for the arc in the Angle box in the Command Line.

To have the first copy located on the original object, choose the Use original as first copy box.

To force the handle to remain in the same place, choose the Fixed handle box in the Command Line.

Set a point for the center of the circle.

See Also: [Circular Array command](#)

Selecting Printer

You can only select a printer in DesignCAD that you have installed on your system. For more information on installing printer software, see your Windows documentation.

Selecting printer setup options

Choose the Printer Setup command from the File menu.

Choose the printer you want to use, if more than one is available.

Choose the OK button.

See Also: [Printing](#), [Printer Setup](#), [Printer Driver \(File\)](#)

Keyboard Techniques

The following table provides a review of basic keyboard techniques you will need to know to work in DesignCAD.

To	Do This
Double-click	Press Alt+Shift+Ins
Drag selection box	Press Alt+Ins
Move horizontally	Press Ctrl+Alt
Move vertically	Press Shift+Alt
Repeat last command	Press F3
Select Coordinate Bar	Press F6

By activating the Select Mode before selecting an entity, you can perform these commands:

To	Do This
Scale Zoom	Move pointer to highlighting node, press Shift
Select several entities	Select first entity, press Shift+Ins to select next entity
Copy entities	Select entity, press Ctrl+Ins inside selection box

By activating the Point Select Mode before selecting an entity, you can perform these commands:

To	Do This
Extend line	Move pointer to end of line, press Shift+Ins
Cut line	Move pointer to line, press Ctrl+Ins
Delete point	Move pointer to point, press Ctrl+Shift

Setting points

The Preset Point Mode allows you to set points before selecting a command. If the Preset Point Mode is not active, you must select a command before setting the points required for the command.

See Also: [Controlling the Cursor](#), [Selecting Commands with the Keyboard](#), [Selecting Objects with the Keyboard](#)

DOS Keystroke Commands

A	<u>Arc-1</u>
<	<u>Angle, Distance-Two Points</u>
Ctrl+A	<u>Angle Between Two Lines</u>
>	<u>Arrow</u>
\$	<u>Attribute</u>
] 	<u>Box</u>
 	<u>Break Line</u>
Ctrl+F	<u>Chamfer</u>
O	<u>Circle-1</u>
Ctrl+C	<u>Copy</u>
+	<u>Crosshair</u>
Ctrl+F11	<u>Cursor Step Size</u>
C	<u>Curve</u>
Ctrl+X	<u>Cut</u>
@	<u>Dimension</u>
G	<u>Display Grid</u>
E	<u>Erase</u>
Ctrl+E	<u>Erase Last</u>
F8	<u>Exit</u>
F	<u>Fillet</u>
Ctrl+W	<u>Fit to Window</u>
F7	<u>Text Options</u>
.	<u>Gravity</u>
,	<u>Gravity Move</u>
Ctrl+F12	<u>Grid Options</u>
F4	<u>Group Explode</u>
#	<u>Hatch</u>
F1	<u>Help Index</u>
I	<u>Show Info Box</u>
N	<u>Intersect-1</u>
Ctrl+N	<u>Intersect-1 Move</u>
L	<u>Layer Setup</u>
V	<u>Line</u>
K	<u>Line Snap</u>
Ctrl+K	<u>Line Snap Move</u>
Ctrl+F9	<u>Load Selection</u>
Ctrl+Q	<u>Macro Record Options</u>
F2	<u>Midpoint</u>
Ctrl+F2	<u>Midpoint Move</u>
Y	<u>New</u>
F9	<u>Open</u>
Ctrl+O	<u>Original Size</u>
=	<u>Parallel-1</u>
P	<u>Point Mark</u>
*	<u>Point Move</u>
;	<u>Point Polar</u>
'	<u>Point Relative</u>
Ctrl+S	<u>Point Select Mode</u>
Ins	<u>Point Set</u>
:	<u>Point XY</u>
F5	<u>Plot</u>
F5	<u>Print</u>
(<u>Quarter Circle</u>

Ctrl+U	<u>Redo</u>
Ctrl+R	<u>Regenerate</u>
F3	<u>Repeat</u>
F10	<u>Save</u>
Shift+X	<u>Save Selection</u>
Ctrl+F5	<u>Select All</u>
Shift+O	<u>Select Ortho</u>
Shift+P	<u>Select Previous</u>
S	<u>Select Mode</u>
Shift+R	<u>Select Rotate</u>
Shift+Z	<u>Select Zoom</u>
)	<u>Semi-Circle</u>
Shift+Alt+A	<u>Show Attribute</u>
H	<u>Show Color Table</u>
Ctrl+Alt+B	<u>Show Coordinate Bar</u>
/	<u>Show Line Style Box</u>
Shift+Alt+T	<u>Show Hot Toolbox</u>
Shift+Alt+M	<u>Show Main Toolbox</u>
J	<u>Show Polar Coordinates</u>
Shift+Alt+R	<u>Show Rulers</u>
Shift+Alt+N	<u>Show Snap Toolbox</u>
Shift+Alt+B	<u>Show Status Bar</u>
Ctrl+G	<u>Snap Grid</u>
&	<u>Stop Recording Macro</u>
T	<u>Text</u>
F7	<u>Text Options</u>
B	<u>Trim Two Lines</u>
Ctrl+B	<u>Trim Double Lines</u>
Ctrl+U	<u>Undo</u>
!	<u>Unerase</u>
U	<u>Units</u>
Z	<u>Zoom</u>
{	<u>Zoom In</u>
}	<u>Zoom Out</u>
Ctrl+P	<u>Zoom Previous</u>

You can use the function keys on your keyboard to perform many tasks faster.

F1	Set a point at the intersection of two lines	F1
F2	Set point at intersection of two lines	F2
	Move pointer to intersection of two lines without setting point	Ctrl+F2
F3	Repeat the last command	F3
F4	Explode Group into individual entities	F4
F5		

Send drawing to printer	F5
F6	
Send drawing to plotter	F6
F7	
Set text options	F7
F8	
Exit DesignCAD and end current session	F8
F9	
Open existing drawing	F9
Load a selected part of a drawing	Ctrl+F9
F10	
Save current drawing	F10
F11	
Set cursor step size	Ctrl+F11

Setting Print Options

After the Print command is selected, you can choose print options from the print options box.

The Drawing Size of your drawing is displayed at the top of the dialog box. Information such as print area, paper size, and panel marks can be changed within this dialog box.

[Setting Drawing Units](#)

[Setting Print Area](#)

[Paneling Drawing](#)

[Printing Several Copies](#)

[Setting Paper Margins](#)

[Printing Selected Objects](#)

[Printing to File](#)

[Previewing a Drawing](#)

[Setting Up Printer](#)

Setting Drawing Units with Print command

By selecting the Units box, your drawing can be printed using one of several units of measurement available.

Changing the Drawing Units

Open the drawing to be printed.

Choose the Print command.

Choose the units of measurement in the Units box.

Inches

Meters

Centimeters

Millimeters

Choose the Print button

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Setting Print Area

Changing Drawing Scale

Changing Drawing Height

Changing Drawing Width

Centering Drawing

Changing Drawing Scale

Open the drawing to be printed.
Choose the Print command.
Choose the Fit to Paper box to disable it.
Type the new scale in the Scale box.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Changing Drawing Height

Open the drawing to be printed.
Choose the Print command.
Choose the Fit to Paper box to disable it.
Type the new height in the Height box.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Changing Drawing Width

Open the drawing to be printed.
Choose the Print command.
Choose the Fit to Paper box to disable it.
Type the new width in the Width box.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Centering Drawing

Open the drawing to be printed.
Choose the Print command.
Choose the Center Drawing box to enable it.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Paneling a Drawing

If the drawing size is larger than the paper loaded in the printer, the Panel section of the Print Options dialog box displays the number of panels, rows, columns, and panel mark options if the drawing is to be printed in panels.

Panel marks, or lines, can be drawn inside the corner of each panel to help you put the drawing together once it is printed. These panel marks can be drawn in the corner of each panel or around the entire panel, outlining the drawing. These panels can contain numbers so that you can put them together sequentially.

Marking Panels

Numbering Panels

See Also: Setting Print Options, Print command, Printing, Printer Setup, Printing a Drawing

Marking Panels

Open the drawing to be printed.
Choose the Print command.
Choose the type of panel mark in the Mark box.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Numbering the Panels

Open the drawing to be printed.
Choose the Print command.
Choose the Mark Panel Number box.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Panel Marks

Panel marks are lines drawn inside the corner of each panel to help you put the drawing together once it is printed. These panel marks can be drawn in the corner of each panel or around the entire panel, outlining the drawing. These panels can contain number so that you can put them together sequentially.

Printing Several Copies

The DesignCAD Print Options dialog box allows you to print several copies of a drawing at once.

Printing more than one copy

Open the drawing to be printed.

Choose the Print command.

Type the number of copies to be printed in the Number of Copies box.

Choose the Print button.

See Also: [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Setting Paper Margins

The Paper section of the Print Options dialog box displays the size of paper loaded in your printer at the top of the box. The page margins and orientation are changed within this section.

[Changing Page Margins](#)

[Setting Page Orientation](#)

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Changing Page Margins

Open the drawing to be printed.
Choose the Print command.
Type the top page margin in the Top box.
Type the bottom page margin in the Bottom box.
Type the left page margin in the Left box.
Type the right page margin in the Right box.
Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Setting Page Orientation

Open the drawing to be printed.

Choose the Print command.

Choose Portrait in the Orientation box to have your drawing printed with the drawing width the same direction as the printer width.

Choose Landscape in the Orientation box to have your drawing rotated 90 degrees, or printed sideways on the page.

Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Printing to File

The Print to File box in the Print Options dialog box allows you to send a drawing to a disk file.

Printing part of a drawing

- Choose the Print command.
- Choose the Print to File box.
- Choose the Print button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Previewing a Drawing

[Previewing Drawing](#)

[Setting Margins](#)

[Setting Margins without Redraw](#)

[Entering Exact Margins](#)

[Positioning Drawing](#)

[Changing Drawing Scale](#)

[Previewing Panels](#)

Previewing Panels

Choose the Print command.

Choose the Preview button.

Choose the Panel button to display the Preview Panel dialog box.

Type the column number of the panel you want to preview in the Column box.

Type the row number of the panel you want to preview in the Row box.

Choose the OK button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Changing Drawing Scale with Print Preview

Choose the Print command.

Choose the Preview button.

Choose the Scale button to display the Preview Scale dialog box.

Choose the Center Drawing box to have the drawing centered on the paper.

Choose the Fit to Paper box to have the drawing fill the entire page when printed.

-or-

Disable the Fit to Paper button to have the drawing printed with adjustable width and height.

Type the drawing scale in the Scale box.

Type the drawing height in the Height box.

Type the drawing width in the Width box.

Press **Enter**.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Positioning Drawing

Choose the Print command.

Choose the Preview button.

Choose the Position button.

Set a point inside the drawing in the preview box.

Use the arrow keys or the mouse to drag the drawing to its new location.

Set another point.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Previewing Drawing

The Preview button in the Print Options dialog box allows you to see how the drawing will be printed on the page. The Preview screen also shows you how a drawing will be printed in panels if it is too large to print on a single page.

Previewing drawings

Choose the Print command.

Choose the Preview button.

The panel coordinates show the number, column, and row of the current panel.

The pointer coordinates show the X and Y coordinates of the pointer in relation to the drawing.

Choose the Print button to print the drawing.

Choose the Current Panel box to print the current panel if your drawing has more than one panel.

-or-

Choose the All button to print the entire drawing.

Choose the OK button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Setting Margins

Choose the Print command.

Choose the Preview button.

Choose the Margins button to display the margins.

Choose the Top Left margin handle to change the margins.

Move the margin handles.

Choose the Print button to print the drawing.

Choose the Current Panel box to print the current panel if your drawing has more than one panel.

-or-

Choose the All button to print the entire drawing.

Choose the OK button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Setting Margins without Redrawing Screen

Choose the Print command.

Choose the Preview button.

Choose the Margins button to display the margins.

Press **Shift** and choose the Top Left margin handle to change the margins.

Press **Shift** and while moving the left margin handle.

Choose the Print button to print the drawing.

Choose the Current Panel box to print the current panel if your drawing has more than one panel.

-or-

Choose the All button to print the entire drawing.

Choose the OK button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Entering Exact Margins

Choose the Print command.

Choose the Preview button.

Choose the Margins button to display the margins.

Press **Ctrl** and choose the a margin handle.

Type the new margin in the margin box.

Choose the OK button.

Choose the Print button to print the drawing.

Choose the Current Panel box to print the current panel if your drawing has more than one panel.

-or-

Choose the All button to print the entire drawing.

Choose the OK button.

See Also: [Setting Print Options](#), [Print command](#), [Printing](#), [Printer Setup](#), [Printing a Drawing](#)

Panel Coordinates

The Panel Coordinates shows the number, row, and column of the panel you are viewing.

Pointer Coordinates

The Pointer Coordinates shows where the pointer is in relation to the drawing. When the pointer is on one of margin handles, the pointer coordinates displays the margin options.

Setting Up Printer

The Printer Setup button in the Print Options dialog box opens the Printer Setup command. The Printer Setup command allows you to choose from a list of available printers and plotters.

For more information on installing printers and plotters, refer to your Windows, printer, or plotter documentation.

The dialog box that appears depends on the type of printer or plotter you choose. You can also change Printer Setup options within this dialog box.

See Also: [Printing](#), [Printer Setup](#), [DesignCAD Print Options](#)

Reading and Saving Views

[Saving Drawing Views](#)

[Opening Drawing Views](#)

Saving Drawing Views

Choose the drawing view to be saved.
Choose View Save from the File menu.
Type the name of the drawing view in the File Name box.
Press **Enter**.

See Also: [View Read command](#), [View Save command](#)

Opening Drawing Views

Choose View Read from the File menu.
Type the name of the drawing view in the File Name box.
Press **Enter**.

See Also: [View Read command](#), [View Save command](#)

Recording Macro Options

[Recording Layer, Color, and Line Style](#)

[Recording Command Options](#)

[Recording Input Information](#)

[Recording Points](#)

[Disabling Record Input Information](#)

Recording Layer, Color, and Line Style

The Record Layer Information box records layer information such as layer color, layer name, and current layer with the macro. The Record Color Information box records the current color of the drawing with the macro. The Record Line Style Information box records the current line style and width of the lines drawn within the macro.

Recording options

When the macro is being recorded, choose the Record Options command from the File menu.

Choose the Layer Info box in the Save box.

Choose the Color Info box in the Save box.

Choose the Line Style Info box in the Save box.

Choose the OK button.

See Also: [Recording a Macro](#), [Recording Macro Options](#)

Recording Command Options

The Record Options box records all drawing options including color, line type, dimension options, and layer options. However, information entered in the Command Line or in the Text Block dialog box is not recorded with the Record Options box.

Recording options

When the macro is being recorded, choose the Record Options command from the File menu.
Choose the Command Opts box in the Save box.
Choose the OK button.

See Also: [Recording a Macro](#), [Recording Macro Options](#)

Recording Input Information

The Record Input Information box records any information entered in the Command Line or the Text Block dialog box.

Recording options

When the macro is being recorded, choose the Record Options command from the File menu.
Choose the Input Info box in the Save box.
Choose the OK button.

See Also: [Recording a Macro](#), [Recording Macro Options](#)

Recording Points

The Record Points Set box records any points set within the macro.

Recording options

When the macro is being recorded, choose the Record Options command from the File menu.

Choose the Points Set box in the Save box.

Choose the OK button.

See Also: [Recording a Macro](#), [Recording Macro Options](#)

Disabling Record Input Information

If the Record Input Information box is disabled, any information entered in the Command Line or Text Block dialog box will not be recorded with the macro.

If both the Record Input Information box and the Record Points Set box are disabled, neither input or point information will be recorded. When the macro is executed, DesignCAD will execute the commands, stopping when it reaches the input for the points. The macro will pause until points are input or until **Enter** is pressed.

If the Record Points Set box is enabled and the Record Input Information box is disabled, the point information will be recorded and the input information will not. When the macro is executed, DesignCAD will execute the commands, set the points, and wait for further information to be entered or **Enter** to be pressed.

Disabling input information

When the macro is being recorded, choose the Record Options command from the File menu.

Disable the Input Info in the Save box.

Choose the OK button.

See Also: [Macro Options](#)

Aligning a Drawing

Choose the Align Drawing command from the View menu.
Set a point for the first reference point.
Set a point for the new location of the first reference point.
Set a point for the second reference point.
Set a point for the location of the second reference point.

See Also: [Align Drawing command](#)

Scaling Three Sides of Object

DesignCAD allows you scale objects disproportionately along the X, horizontal, or Y, vertical, axis. This command "stretches" three side of the object to make them taller, shorter, or wider.

Changing the width of an object

Select the object to be scaled.
Press **Shift** and click the left or right select nodes.
Press **Shift** and move the pointer in the desired direction.
Set a point.

Changing the height of an object

Select the object to be scaled.
Press **Shift** and click the top or bottom select nodes.
Press **Shift** and move the pointer in the desired direction.
Set a point.

Changing diagonal width or height of object

Select the object to be scaled.
Press **Shift** and click the left or right select nodes.
Press **Shift** and move the pointer in the desired direction.
Set a point.

See Also: [Select Scale command](#)

Scaling One Side of Object

DesignCAD allows you scale objects disproportionately along the X, horizontal, or Y, vertical, axis. This command "stretches" one side of the object to make it taller, shorter, or wider.

Changing the width of an object

- Select the object to be scaled.
- Choose the left or right select nodes.
- Move the pointer in the desired direction.
- Set a point.

Changing the height of an object

- Select the object to be scaled.
- Choose the top or bottom select nodes.
- Move the pointer in the desired direction.
- Set a point.

Changing diagonal width or height of object

- Select the object to be scaled.
- Choose the left or right select nodes.
- Move the pointer in the desired direction.
- Set a point.

See Also: [Select Scale command](#)

Setting Points Along a Line

Choose the Point Line Distance command from the Options menu.

Type the distance.

Choose the Point Line Snap command from the Point menu.

Set a point for the starting point.

Set a point for the direction.

See Also: [Setting Regular Points](#), [Entering Coordinates for Points](#), [Selecting Points](#)

Selecting a Hatch Pattern

Hatch patterns can be used to "fill in" areas and entities in the drawing with solid or line drawings. These hatch patterns are drawn as scalable and non-scalable patterns.

When the Hatch, Hatch Fill, or Hatch Line commands are selected, the hatch pattern. When the Hatch button in the Command Line is selected, the Hatch Pattern dialog box is opened.

You can use the mouse, arrow keys, or **Tab** and **Space bar** to choose a hatch pattern within the Hatch Pattern dialog box.

Selecting a hatch pattern

Choose the Hatch command from the Draw menu.

Choose the Hatch button in the Command Line.

Choose the Hatch button in the Command Line.

Press **Enter**.

Set the points for the hatch.

Press **Enter**.

See Also: [Using Hatch Patterns](#), [Hatch Patterns](#)

Scalable

The size of scalable hatch patterns is entered when the Hatch commands are selected. The size of these patterns will change when a drawing is zoomed and can be changed with the Info Box.

Non-Scalable

The size of non-scalable patterns cannot be changed and will not change when a drawing is zoomed.

Running BasicCAD Programs

Choose the Run command from the File menu.
Type the program name in the File Name box.
Press **Enter**.

See Also: [Run_command](#)

Moving Objects Vertically

Activate the Select Mode.

Select the object to be moved.

Move the pointer to the area inside the selection box.

Press **Alt+Shift** while dragging the pointer.

Move the selection box to the new location.

Set a point.

See Also: [Moving Objects](#), [Moving Objects with Handles](#), [Moving Objects Horizontally](#)

Moving Objects Horizontally

Activate the Select Mode.

Select the object to be moved.

Move the pointer to the area inside the selection box.

Press **Alt+Ctrl** while dragging the pointer.

Move the selection box to the new location.

Set a point.

See Also: [Moving Objects](#), [Moving Objects with Handles](#), [Moving Objects Vertically](#)

Visible Layer

A visible layer is an active layer that is displayed on the screen.

Invisible Layer

An invisible layer is an inactive layer that is not displayed on the screen.

Using Printer Fonts

If your printer cannot handle TrueType fonts at lower resolution, such as 75 d.p.i. which can be used to print a "preview" of a drawing, a Use Printer Font check box can be selected in the Printer Setup dialog box.

Using printer fonts

- Choose the Printer Setup command from the File menu.
- Choose the Use Printer Font Only box if it is available.
- Choose the OK button.

See Also: [Printing](#), [Printer Setup](#), [DesignCAD Print Options](#)

Viewing Bitmaps

[Saving Bitmaps](#)

[Reading Bitmaps](#)

Saving Bitmaps

Open the drawing.

Choose Save Bitmap from the File menu.

Set a point in the lower left corner of the area to be saved.

Set a point in the upper right corner of the area to be saved.

-or-

Press **Enter** to save the entire bitmap.

Type the bitmap file name in the File Name box.

Choose the OK button.

See Also: [Save Bitmap command](#), [Reading Bitmaps](#)

Reading Bitmaps

Choose Load Bitmap from the File menu.
Type the bitmap file name in the File Name box.
Choose the OK button.
Set a point for the lower left corner of the bitmap.
Set a point for the upper right corner of the bitmap.

See Also: [Load Bitmap command](#), [Saving Bitmaps](#)

Listing Attributes

Choose the Material List command from the Dimension menu.
Choose the Save command from the File menu to save the list.
Choose the Print command from the File menu to print the list.
Choose the Printer Setup command from the File menu to setup the printer.
Choose the Window Control-menu to close the Material List window.

See Also: [Material List command](#), [Attribute command](#)

The Command Line








Command Line

Default Command Line

Default Command Line



The Command Line displays the most-commonly used file commands.

<u>Click</u>	To
	Cut Cuts object and places it in Clipboard.
	Copy Copies object to Clipboard.
	New Opens new drawing screen.
	Open Opens drawing.
	Paste Pastes Clipboard contents to drawing.
	Print Prints current drawing.
	Save Saves current drawing.

When a command is selected that requires information to be entered, the Command Line commands are hidden. The Command Line then contains information for the selected command.

See Also: [Parts of the DesignCAD Screen](#)

Saving Drawing to Drive or Directory

Choose the Save As command from the File menu.

In the File Name box, type the complete path and filename.

-or-

In the File Name box, type the filename. Then choose a new directory in the Directories box.

Choose a new drive in the Drives box.

Choose the OK button.

See Also: [Saving a Drawing](#), [Saving Drawings](#)

Setting Text Style

Setting style within the command

Choose the Text command.

Type the text in the Text box in the Command Line.

Choose the Bold button to have the text drawn in bold type.

Choose the Italic button to have the text drawn in italic type.

Setting style with the Text Options command

Choose the Text Options command.

Choose the text style in the Style box.

Choose the OK button.

See Also: [Changing Text Options](#)

Setting Text Height and Angle

Setting style within the command

Choose the Text command.

Type the text in the Text box in the Command Line.

Type the text size in the Text Size box in the Command Line.

Type the text angle in the Text Angle box in the Command Line.

-or-

Set two points for the lower left and lower right corners of the text.

Setting style with the Text Options command

Choose the Text Options command.

Enter the text size in the Size box.

Enter the text angle in the Angle box.

Choose the OK button.

See Also: [Changing Text Options](#)

Setting Text Orientation

Choose the Text command.

Type the text in the Text box in the Command Line.

Choose Left in the Orientation box to have the text left-justified.

Choose Center in the Orientation box to have the text centered.

Choose Right in the Orientation box to have the text right-justified.

See Also: [Changing Text Options](#)

Setting Preferences

Changing Default Options
Default Options

Changing Default Options

When you choose the Options command in the Options menu, DesignCAD displays the Options dialog box.

The Options dialog box is divided into two parts: the Category box on the left and the options panel on the right. When you select a category, the panel changes to display the options and default settings in that category. You determine the default settings for any category by selecting or clearing the options in the panel. The new default settings take effect when you choose another category or close the dialog box.

Setting options

Choose Options from the Options menu.

In the Category box, choose the category containing the default options you want to change.

In the panel on the right, choose the options to specify the default settings you want DesignCAD to use.

Select another category or close the dialog box by choosing the OK or Cancel button.

Changing startup file

Change the options.

Choose the Save options box.

Choose the OK button.

Note: The Save options box saves the current options to the DCW.INI file.

See Also: [Options command](#), [Default Options](#)

Default Options

Toolbox

Symbol Toolbox

Macro

Coordinate Bar

View

Drawing

Text

Cursor

Grid

General Dimension

Dimension Text

Dimension Prefix and Suffix

Digitizer Menu

Miscellaneous

Symbol Toolbox Options

Symbol Toolbox

Select the Symbol Toolbox category to specify the number of columns in the Symbol Toolbox and to create a Symbol Toolbox.

Symbol Toolbox

Use options under "Symbol Toolbox" to set toolbox columns, specify the directory for the symbol files, and add symbols to the symbol menu. The maximum size for a toolbox is 18 by 18.

Column No Select this option to set the number of columns in the Symbol Toolbox.

Directory Use this option to enter the drive and directories for the symbols files.

Current Content Displays the contents of the Symbol Toolbox.

Available Symbols Select the symbol to assign to the toolbox from the list of available symbol.

Add Adds the selected symbol to the Symbol Toolbox.

Delete Deletes the selected symbol from the Symbol Toolbox.

See Also: [Options command](#), [Changing Default Options](#)

Digitizer Menu Options

Digitizer Menu

Use the Digitizer Menu category to set digitizer menu options

Modify Digitizer Menu

Use options under "Modify Digitizer Menu" to remove, disable, and enable the current digitizer menus.

DigiMenu The options modify the digitizer menus loaded in DesignCAD.

Check box When the check box next to the digitizer menu name is checked, the digitizer menu is loaded. Clear the check mark next to the digitizer menu name to remove the digitizer menu.

Enable When the Enable button next to the digitizer menu is selected, the digitizer menu is active.

Disable When the Disable button next to the digitizer menu is selected, the digitizer menu is no longer active.

See Also: [Options command](#), [Changing Default Options](#)

Toolbox Options

Select the Toolbox category to specify the number of columns in the Main, Snap, and Hot toolboxes. Then each time you open the drawing, the toolbox columns will be set. Once the drawing is open, you can change the toolbox columns at any time by changing the appropriate options.

Toolbox Column No.

Use options under Toolbox Column No. to set individual toolbox columns. The maximum size for a Toolbox is 18 by 18.

Main Toolbox Column no: Select this option to set the number of columns in the Main Toolbox.

Snap Toolbox Column no: Select this option to set the number of columns in the Snap Toolbox.

Hot Toolbox Column no: Select this option to set the number of columns in the Hot Toolbox.

Hot Toolbox

Use options under Hot Toolbox to add or delete tools to the Hot Toolbox.

Current Content Shows the current icons in the Hot Toolbox.

Available Tools Choose the tool icons from this box. Use the scroll bar to view additional tools.

Add Select this button to add a selected tool to the Hot Toolbox.

Delete Select this button to delete a selected tool from the Hot Toolbox.

See Also: [Options command](#), [Changing Default Options](#)

Macro Options

Select the Macro category to define the macro buttons in the Command Line, set macro options, and define the startup macro.

Modify Macro Settings

Use options under Modify Macro Settings to set macro options.

Button to Change Select the buttons, icons, and macros for the macro buttons.

Which Select the macro button to change.

Icon Select the icon to use for the defined macro button.

Available Macros Select the macro to define for the button from the list of available macros.

Keystroke Shows the keystroke used for the defined macro.

Change Select this button to change the button definitions.

Save Select the macro record options to be saved with each macro.

Layer Info Choose this box to record layer information such as layer color, layer name, and current layer with the macro.

Color Choose this box to record the current color of the drawing with the macro.

Line Style Choose this box to record the current line style and width of the lines drawn within the macro.

Input Choose this box to record most information entered in the Command Line or the Text Block dialog box. If the Record Input Information box is disabled, most information entered in the Command Line or Text Block dialog box will not be recorded with the macro.

If both the Record Input Information and the Record Points Set boxes are disabled, neither input or point information will be recorded. When the macro is executed, DesignCAD will execute the commands, stopping when it reaches the input for the points. The macro will pause until points are set or until **Enter** is pressed. This allows interactive macros.

If the Record Points Set box is enabled and the Record Input Information box is disabled, the point information will be recorded and the input information will not. When the macro is executed, DesignCAD will execute the commands, set the points, and wait for further information to be entered or **Enter** to be pressed.

Command Opts Choose this box to record all drawing options including color, line type, dimension options, and layer options. However, some information entered in the Command Line or in the Text Block dialog box is not recorded with the Macro Record Options command.

Points Set Choose this box to record any points set within the macro.

Startup macro Select the macro file to be used as the startup macro.

Note: Macros should be stored in the directory where DesignCAD was installed so that they will appear in the Macro Options dialog box. The default directory is C:\DCAD.

See Also: [Options command](#), [Changing Default Options](#)

Coordinate Bar Options

The options under Coordinate Bar change the Coordinate Bar information.

Coordinates Display Type This option switches between Cartesian and Polar coordinates.

Cartesian Choose this option to have the X, Y, Delta X, and Delta Y coordinates displayed in the Coordinate Bar.

Polar Choose this option to have the polar coordinates, or angle and length from last point, displayed in the Coordinate Bar.

Distance Format This option sets the format that the coordinates are displayed in Coordinate Bar.

Decimal - 1.400

Fractional - 1-1/2

Engineering - 1' 6.75"

Precision (Fractional Digits) Use this option to set the precision of fractional digits displayed in the Coordinate Bar.

10000000

1000000

100000

10000

1000

100

10

1

1.0

1.00

1.000

1.0000

1.00000

1.000000

1.0000000

Angle Format This option sets the angle format for polar coordinates.

Decimal degrees - 45.0

Grads - 50.0g

Radians - 0.393rad

Degree/minutes/seconds - 45°0'0"

Surveyor's units - N 45°0'0"E

Angles Use this option to set Mathematical or Geographical angles.

Mathematical With Mathematical angles, the degrees progress counter-clockwise with zero degrees to the right.

Geographical With Geographical angles, the degrees progress clockwise with zero degrees straight up.

See Also: [Options command](#), [Changing Default Options](#)

View Options

Select the View Options category to disable or enable screen elements.

Show/Hide

Select options under Show/Hide to enable or disable individual screen elements. A check mark in the box indicates that the element will be displayed on the screen.

Main Toolbox The Main Toolbox contains the most-commonly used drawing commands.

Snap Toolbox The Snap Toolbox contains the most-commonly used point commands.

Hot Toolbox The Hot Toolbox can contain the commands you use most often.

Status Bar The Status Bar displays information on selected commands, the zoom factor, and the number of points set.

Coordinate Bar The Coordinate Bar displays the X, Y, Delta X, Delta Y or polar coordinates of the cursor and the current layer.

Rulers The Rulers show where the cursor is in the drawing.

Color Table The Color Table displays the current color and color table.

Line Style Box The Line Style Box displays the current line style, line width, and line scale.

Zoom In/Out Enter the percentage a drawing is zoomed with the Zoom In and Zoom Out commands.

See Also: [Options command](#), [Changing Default Options](#)

Drawing Options

Select the Drawing Options category to set options for DesignCAD drawing commands.

Modify Drawing Options

Use options under Modify Drawing Options to set line, color, point mark, and point options.

Line Style Select the line type, scale, and width for lines in the drawing with this option.

Line Type Select the line type from the available list of line types.

Scale Enter the scale for the line.

Width Enter the width of the line.

Fill Wide Line Choose this box to have wide lines filled.

Color Use this option to set the default color for DesignCAD.

Point Mark Use this option to select the default point mark for marks drawn with the Point Mark command.

Point Mark Size Enter the size of the point mark in Drawing Units.

Point Type Select the type of point made with DesignCAD commands with this option.

Hatch Pattern Select Hatch pattern to choose the hatch pattern.

See Also: [Options command](#), [Changing Default Options](#)

Text Options

Select the Text category to set the text size, angle, font, style, and spacing.

Text Options

Use the options under Text Options to set the text information.

Font This option determines the font used for text in the drawing. The fonts available depend on your system. See your windows documentation on how to install fonts.

Font Style This option determines the font style used for text in the drawing.

Regular Regular text is drawn normally, without italics or bold.

Italic This option draws italics text.

Bold This option draws bold text.

Bold Italic This option draws both bold and italic text.

Size Type the size for the text in this box.

Angle Type the angle of the text in this box.

Spacing Type the amount of spacing between each letter in this box.

Display Attribute Choose this box to have attributes drawn with the Attribute command displayed on the screen.

See Also: [Options command](#), [Changing Default Options](#)

Cursor Options

Select the Cursor category to set the large and small cursor step size and to determine cursor step consistency.

Cursor Options

Use options under Cursor Options to set cursor step information.

Large Step Size This option determines the distance the cursor will move when an arrow key is pressed.

Small Step Size This option determines the distance the cursor will move when an arrow key is pressed with the **Shift** key.

Cursor Step Relative to: Screen or Drawing This option determines if the cursor step size will be consistent with the drawing or the screen.

Screen This option sets the cursor step sizes to remain consistent with the screen. That is, when the drawing is zoomed, the cursor step sizes remain the same relative to the screen, not the drawing. After a zoom command has been performed, the cursor step values may have changed. The cursor will move the same distance across the screen when an arrow key is pressed.

Drawing This option sets the cursor step sizes to remain consistent with the drawing. That is, the cursor step sizes will not change when the drawing is zoomed.

See Also: [Options command](#), [Changing Default Options](#)

Grid Options

Select the Grid Options category to set options for the snap and display grids.

Grid Options

Use options under Grid Options to set the grid types and sizes for DesignCAD.

Display grid type This option sets the line type for the display grid.

Display grid size Select this option to set the display grid size.

Snap grid size This option sets the snap grid size.

Enable display grid This option displays or "hides" the display grid.

Enable snap grid Select this option to enable or disable the snap grid.

See Also: [Options command](#), [Changing Default Options](#)

General Dimension Options

Use the General Dimension category to set options for dimensions.

Dimension Options

Use options under Dimension Options to set the arrowhead, extension line, layer, color, and text sizes for the dimension commands.

Arrowhead Select this option to set options for the arrowhead associated with the dimension.

Type This option sets the arrowhead, or terminator, type for the dimension arrows.

Size This option sets the size for the arrowhead.

Extension Line Select this option to set overshoot and extension lengths.

Overshoot This option sets the overshoot, or distance the extension line extends past the dimension line.

Fixed Length This option sets the distance between the dimension information and the entity to be a certain length.

Variable Length Select this option to set the distance between the dimension information and the entity to be a variable length.

Gap Size This option sets the "gap size," or space between the dimension information and the entity being dimensioned.

Default Layer Select this option to set the default layer for all dimensions.

Default Color This option sets the default color for all dimensions.

Text Size This options sets the default text size for all dimensions.

Tolerance Size This option sets the positive and negative values given to the amount of variation allowed from the standard measurement.

Balloon Size Use this option to set the default size for the balloon drawn around text with the Balloon command. The size of the balloon is determined by multiplying this number by the text size. The minimum size for the balloon size is 1.5 Drawing Units.

Dim Base Offset This option sets the distance between the numbers drawn with the Dimension Baseline command.

See Also: [Options command](#), [Changing Default Options](#)

Dimension Text Options

Use the Dimension Text category to set the text options for the dimension commands.

Modify Dimension Text Options

Use options under Modify Dimension Text Options to set the linear and angular dimension format and precision.

Linear Dimension Use options under Linear Dimension to set linear format and precision.

Format Choose the linear text format. There are four linear dimension text formats to choose from:

Format	Example
Decimal	1.500
Fractional	1 1/2
Engineering	1' 6.75"
Architectural	1' 6-3/4"

Precision Choose the linear text precision. You can choose these linear precision formats:

10000000
1000000
100000
10000
1000
100
10
1
1.0
1.00
1.000
1.0000
1.00000
1.000000
1.0000000

Angular Dimension Use options under Angular Dimension to set angular format and precision.

Format Choose the angular text format. There are five angular dimension text formats to choose from:

Format	Example
Decimal Degrees	45.0
Grads	50.0g
Radians	0.393rad
Degrees/Minutes/Seconds	45°0'0"
Surveyor's Units	N 45°0'0"E

Precision Choose the angular text precision. You can choose these angular precision formats:

1
1.0
1.00
1.000
1.0000
1.00000
1.000000

Text Size Set the dimension text size.

See Also: [Options command](#), [Changing Default Options](#)

Dimension Prefix & Suffix Options

Use the Prefix & Suffix category to set the text preceding and following the dimension information for the dimension commands.

Prefix

Use options under "Prefix" to set the prefix text for the dimension commands.

Linear Prefix text for linear dimensions.

Angular Prefix text for angular dimensions.

Arc Prefix text for arc dimensions.

Radius Prefix text for radius dimensions.

Diameter Prefix text for diameter dimensions.

Chamfer Prefix text for chamfer dimensions.

Coord Prefix text for coordinate dimensions.

Suffix

Use options under "Suffix" to set the suffix text for the dimension commands.

Linear Suffix text for linear dimensions.

Angular Suffix text for angular dimensions.

Arc Suffix text for arc dimensions.

Radius Suffix text for radius dimensions.

Diameter Suffix text for diameter dimensions.

Chamfer Suffix text for chamfer dimensions.

Coord Suffix text for coordinate dimensions.

See Also: [Options command](#), [Changing Default Options](#)

Miscellaneous Options

Use the Miscellaneous category to set options that do not fall under a specific category.

Point Line Snap Distance This option sets the distance used for the Point Line Snap command.

Retrieve/Copy Scale This option determines if the drawing handles used to load or copy a drawing are scalable or changeable.

Fixed When this option is selected, drawings or symbols being loaded will be rotated to fit between two points. The drawings will not be scaled. The drawings are retrieved at the size they were saved, regardless of the distance between the two points.

Changeable When this option is selected, drawings or symbols being loaded will be rotated and sized to fit between two points.

Sound This options turns the sound on or off.

Off

Error Only

On

Create Backup Copy This option determines if a backup copy will be saved when the drawing is saved with the Save command.

Save Parameters with Drawing This option determines if system parameters such as Drawing Units size, cursor type, and sound are saved with the drawing file.

See Also: [Options command](#), [Changing Default Options](#)

Drawing Text in an Arc

Choose the Text Arc command.

Type the text in the Text box in the Command Line.

Choose the font in the Font box in the Command Line.

Type the text size in the Text Size box in the Command Line.

Type the angle of the text in the Text Angle box in the Command Line.

Choose the orientation of the text in the Orientation box in the Command Line.

Choose the Bold button in the Command Line to have the text drawn in bold.

Choose the Italic button in the Command Line to have the text drawn in italic.

Choose the Same as button in the Command Line to have the text drawn with the same options as text in the drawing. Then choose the text in the drawing.

Set a point for the corner of the text arc.

Set a point for the middle of the text arc.

Set a point for the end of the text arc.

See Also: [Text Arc command](#)

Finding Information on DesignCAD

Choose the About DesignCAD command from the Help menu.
Choose the OK button.

See Also: [About DesignCAD command](#)

Shortcut Keys

Default Shortcut Keys

DesignCAD DOS Keystrokes

Understanding Material List File Format

Materials List File Format

ASCII File

DesignCAD drawing files can be saved with a .DC2 file extension by choosing the ".DC2" extension in the Save As command. This is an ASCII file format, where the data is written in character format. Each "record" in the file is actually a line in a text file.

There are four types of records, or lines, in the file.

Angular Text

The Angular Text options such as format and precision determine how text drawn using the Dimension Angle and Line Angle-2 commands.

Bezier

A Bezier is a curve drawn using the Bezier Curve command. Points are set for the endpoints and the control points.

DOS Keystrokes

DOS Keystrokes are the Shortcut Keys used in the DOS version of DesignCAD. In DesignCAD 2-D for Windows, you may use either the original keystroke commands created for the DOS version, or the new shortcut keys created for the Windows version.

Function Key Commands

F1

Get help on command or a message	F1
Go to the next field	Alt+F1
Get help on a selected command or key combination	Shift+F1
Go to previous field	Alt+Shift+F1

F2

Set a point at the midpoint of line	F2
Move the cursor to the midpoint of line without setting a point.	Ctrl+F2

F3

Repeat the last completed command	F3
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F4

Close the current drawing	Ctrl+F4
Exit DesignCAD and end current session	Alt+F4

F6

Select Coordinate Bar	F6
Choose next window as active window	Ctrl+F6

F7

Change text options	F7
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F9

Load a selected part of a drawing	Ctrl+F9
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F12

Save a new, unnamed drawing or save a drawing with a different name	F12
Open a new drawing	Ctrl+F12
Save changes to a drawing	Shift+F12
Print the current drawing	Ctrl+Shift+F12

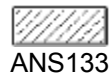
See Also: [Shortcut Keys](#), [DOS Keystroke Commands](#)

Hiding Hot Toolbox

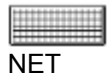
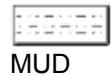
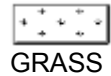
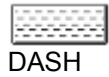
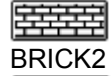
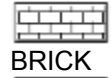
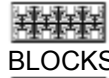
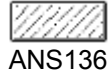
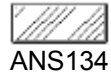
Choose the Hot Toolbox option from the Show/Hide secondary menu in the View menu.

See Also: [Showing Hot Toolbox](#), [Customizing DesignCAD](#)

Scalable Hatch Patterns



ANGLE



Non-Scalable Hatch Patterns



WIN_SOLID



WIN_HORIZONTAL



WIN_VERTICAL



WIN_FWDDIAGONAL



WIN_BKDIAGONAL



WIN_CROSS



WIN_DIAGROSS

Application Control-Menu Button

The Application Control-menu button opens the [Application Control-menu](#), in the upper-left corner of the DesignCAD screen.

See Also: [Title Bar](#)

Drawing Control-Menu Button

The drawing Control-menu button opens the [drawing Control-menu](#).

See Also: [Title Bar](#)

Selecting Certain Entities

Selecting Certain Entities with the Keyboard

Selecting Certain Entities with Mouse

