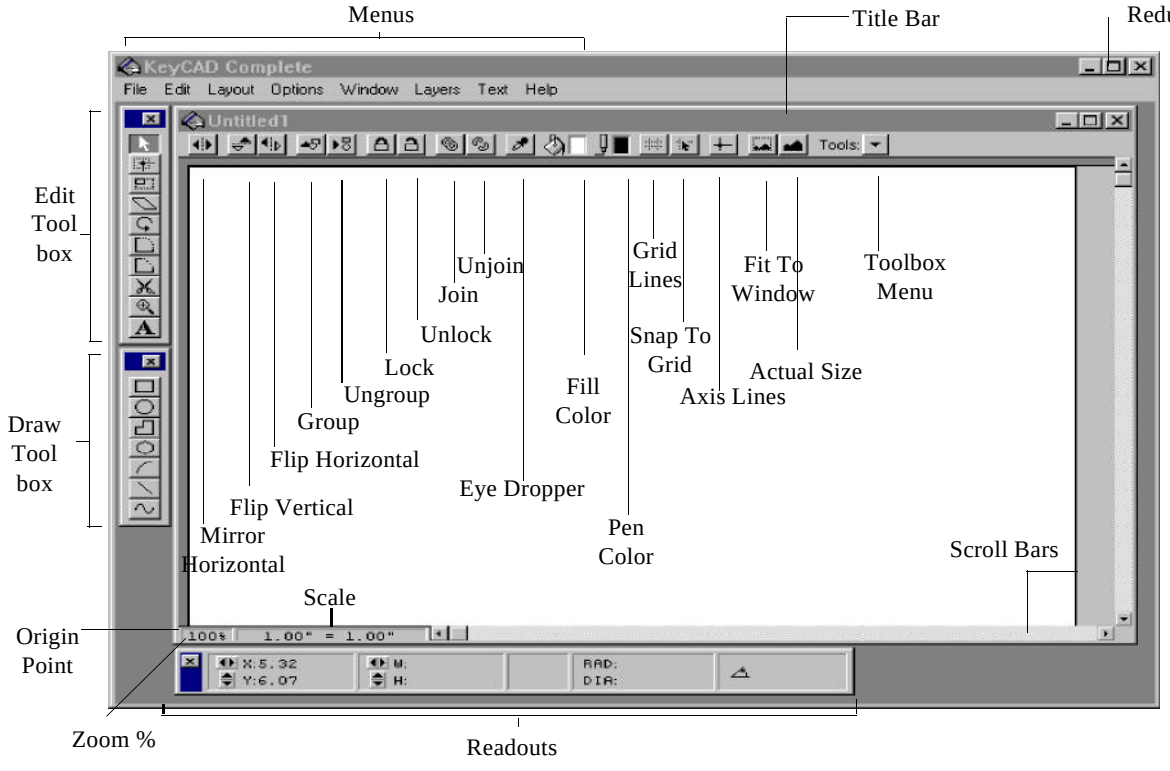
The Basics

This chapter discusses the basic layout of **KeyCAD Pro** for Windows and Macintosh. The general layout of the Program Start Up Window is displayed with all of the correct names of tools, menus, readouts, etc. Each toolbox found within the program is explained in a general overview of how those particular tools work.

This chapter is not intended to provide a step by step explanation of each toolbox and readout. To gain a more detailed explanation of how to use specific tools, menus or commands, refer to the appropriate chapter in this manual.

The Window Layout

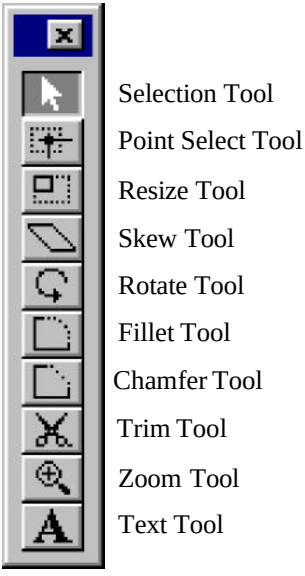


Edit Toolbox

The Edit Toolbox contains tools that are used on previously designed objects. There are only two tools within the Edit toolbox that do not edit the shape of predrawn objects. These are the *Zoom* tool and the *Text* tool.

Remember that the following tools require that an object or points within an object be selected before the *Edit* function selected can be performed. These tools are *Resize, Skew, Rotate, Fillet*, and *Chamfer*. The *Selection* tool and the *Point Select* tool are used for selecting the objects or groups of points to be edited.

The Edit tools use the mouse and keyboard entry when working with objects. When using the mouse, use a click and drag method. If the keyboard entry method is used, the objects or points must first be selected. Then, double click on the required Edit tool. The following Edit tools are the only tools that support keyboard entry. They are: *Selection, Point Select, Resize, Skew, Rotate, Fillet* and *Chamfer* tools. The *Trim, Zoom* and *Text* tools do not require any keyboard entry.



The Edit Toolbox

Draw Toolbox

The **KeyCAD Pro** Draw toolbox is positioned on the left side of the screen when the program is first opened. The toolbox can be moved to any position on the screen. This feature is very useful when working on large monitors.

To select a tool, place the mouse cursor on the tool and click once. The icon then becomes highlighted to show which tool is selected.

The tools found within the Draw toolbox and their proper names are listed below. These tools are designed to allow the user to draw individual objects such as rectangles, circles, polygons, lines, arcs, etc. Some of these tools use the click and drag method of drawing, i.e., click and hold down the mouse button while dragging. Each tool is explained in greater detail under the *Draw Tools* section of this manual.

The Draw tools also allow for keyboard entry of the object size. This is very useful for precise construction of designs. To access the Keyboard Entry Dialog for a tool, double click on the tool icon and the Dialog appears. Enter the object's information in the displayed dialog box.

Rectangle / Square Tool

Oval / Circle Tool

Polygon Tool

Multigon Tool

Arc Tool

Line Tool

Free-formed Spline Tool

The Draw Toolbox

Snap To Toolbox

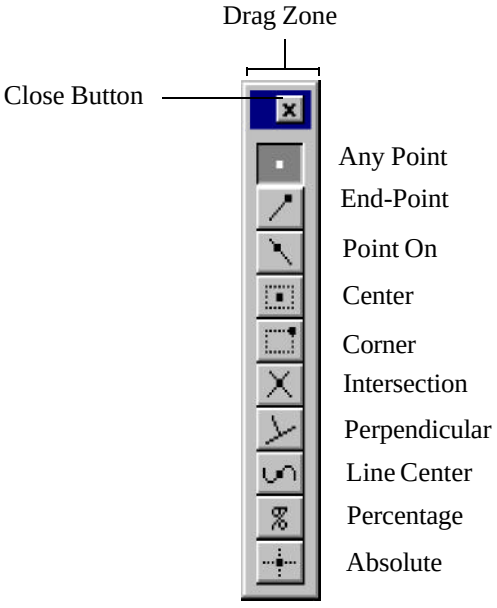
Almost all technical and engineered drawings require lines, circles, and rectangles to be drawn in a very precise location, in specific relationships to each other. For example, to start a line at the intersection of two different objects, or draw a line perpendicular to another line. **KeyCAD Pro** provides the user with the *Snap To* tools to help in locating the precise points required to create a very detailed and accurate design.

The *Snap To* toolbox can be positioned anywhere on the screen and it's display can be turned on or off. This toolbox is not displayed when the program is first started. It may be turnen on by selecting it from the Toolbox menuunder the Window menu. A checkmark beside a toolbox name indicates that it is active.

The *Snap To* tools are designed to work in combination with the *Edit, Draw*, and *Dimension* tools found within the program. When using the *Snap To* tools with the *Draw* tools, a line can be started from the end-point of another line and finish at the center of a circle. Using the *Snap To* tools with other tools ensures that the line or object created starts and ends in the exact location required.

The sensitivity can also be set to determine how close the mouse cusor must be to an object for the program to snap to a specific point. The sensitivity setting is found in the *Preference* Dialog under the **Edit** menu.

The following is a picture of the *Snap To* tools as they appear in the program. The names have been listed for the appropriate tool icon. The name indicates the Snap To command that is to be performed when using that particular tool. These tools should be used in all precise alignment of objects as well as object start or end-points.

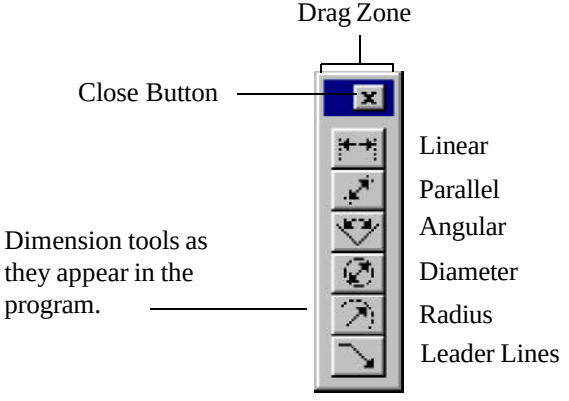


The Snap To Toolbox

NOTE: Refer to the *Snap To Tools* section of this manual for a detailed explanation on how to use these tools.

Dimension Toolbox

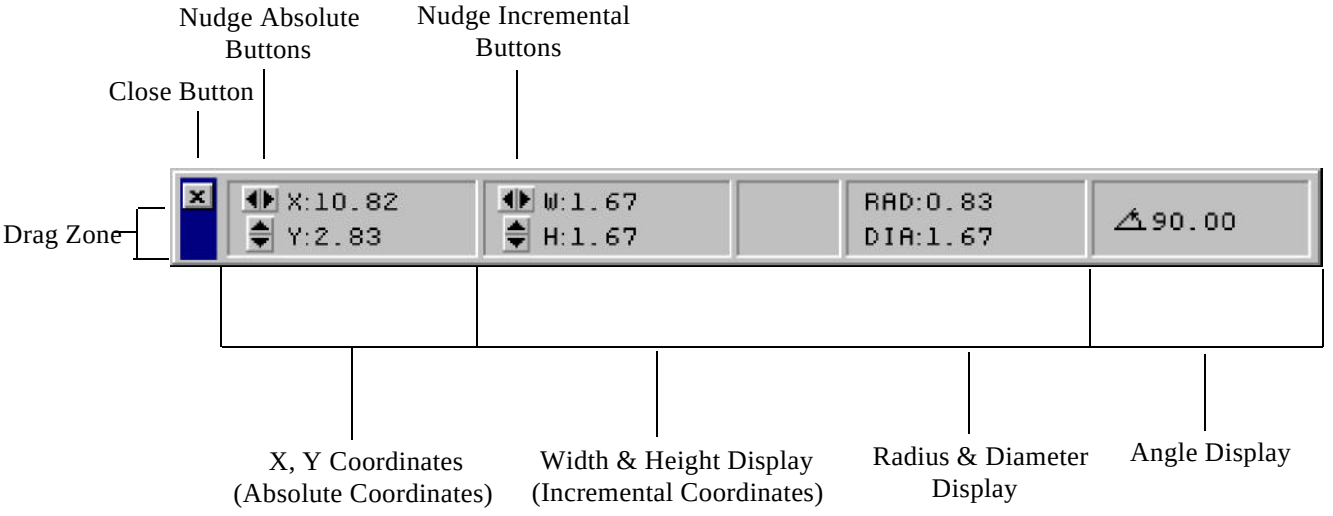
The *Dimension* tools are designed to help create dimension lines that appear with witness lines (also called extension lines), which automatically represent the correct dimension value or measurement. A dimension may contain the size of a particular object or the distance between two separate objects. Dimension lines can be created for diagonal lines that measure the vertical or horizontal distance and even the length of the angular line.



The Dimension Toolbox

One of the most important factors in using these dimension tools, is that they are designed to work in combination with the *Snap To* tools. This ensures that there is an accurate dimension between two designated points.

The Readouts



Readouts (Position Info)

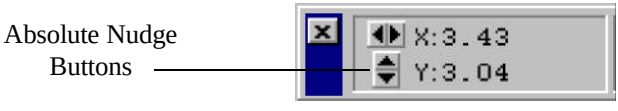
In addition to using the *Snap To* tools for locating specific points on the design, the Readouts found at the bottom of the screen can also be used. The Readouts are designed to assist in specific point locations, angles of lines, and distance or lengths of objects. Readouts are primarily designed to be used with the mouse movements on the screen.

The Readouts are a set of gauges that update automatically with accurate information as the pointer is moved to draw, edit, select or reposition objects. The Readouts are designed to work with English or Metric measurement and display feet and inches, or decimal equivalence.

Refer to the Readouts to help draw or edit any object. For example, as the cursor is positioned to draw an object, the Readouts display the X and Y coordinates of the mouse cursor, which helps to locate the exact spot to begin drawing. As the objects are rotated, the degree of rotation can be checked while watching the Readouts angle display.

Absolute Coordinates

There are two sets of coordinates, one set reads X, Y coordinates and the other reads W, H coordinates. The X, Y displays are called the Absolute Coordinates. This means that they track the mouse location from the point of origin of the screen. The X, Y coordinates assist in the accurate location of the start and end-point of the drawing or editing of an object.



A set of Absolute Coordinates

Incremental Coordinates

The *Incremental Coordinates* are displayed with the letter (W) for the *Width* and (H) for the *Height* of any object drawn.

Select a *Draw* or *Edit* tool; click to start the selected function. The W, H readout is automatically set to zero(0) to track the actual movement of the mouse when drawing or editing an object.

NOTE: The W,H or Incremental Readouts only display and track the start and end-point of the mouse movements.

A Set of Incremental Coordinates

Nudge Buttons

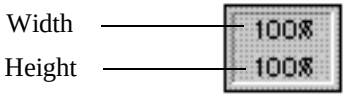
The *Absolute* and *Incremental Nudge* buttons are shown in the two previous figures. These buttons move a selected object by the amount set in the nudge increment. Set the nudging rate under the *Nudge* subheading found under the **Edit** menu at the top of the screen. The program defaults to one pixel nudge rate.

The *Absolute Nudge* buttons move the selected object according to the nudge rate. This does not change the size of the object, but rather changes the location of the object.

The *Incremental Nudge* buttons change the width and height of any selected object. The center of the selected object is used as a reference point, and nudges according to the current nudge rate.

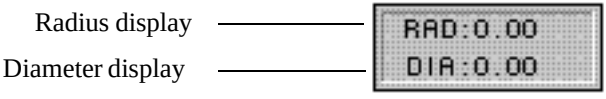
Percentage Readouts

The *Percentage* readouts are used mainly for the resizing of selected objects. When an object is drawn, the percentage display shows 100%, when the mouse button is released. This means that the object drawn is at 100% size. If the object is then resized using the *Resize* tool, there is the option of resizing by percentage versus using the coordinates. The readouts display the width and height percentages.



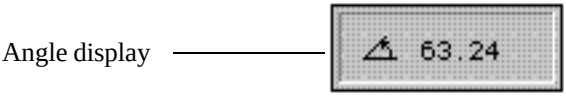
Radius and Diameter Readouts

When arcs or circles are being created, the Radius and Diameter readouts display an accurate measurement of the radius or diameter of the object being drawn.

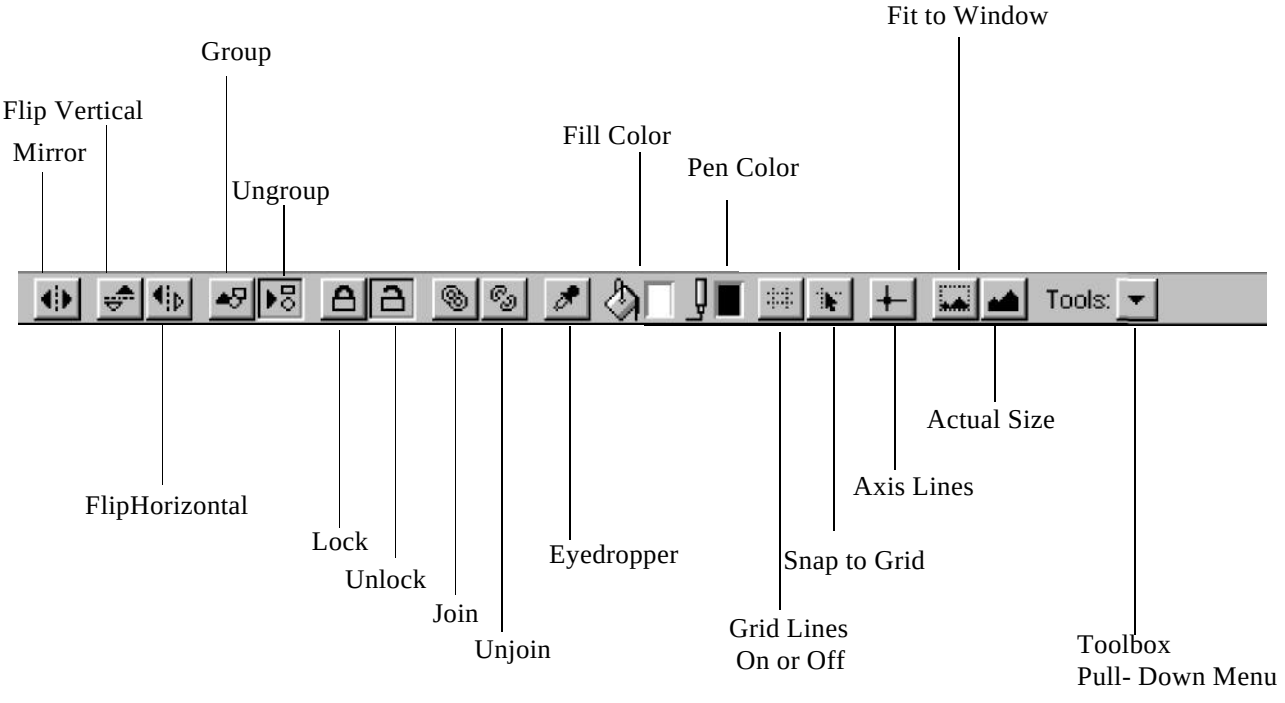


Angle Readouts

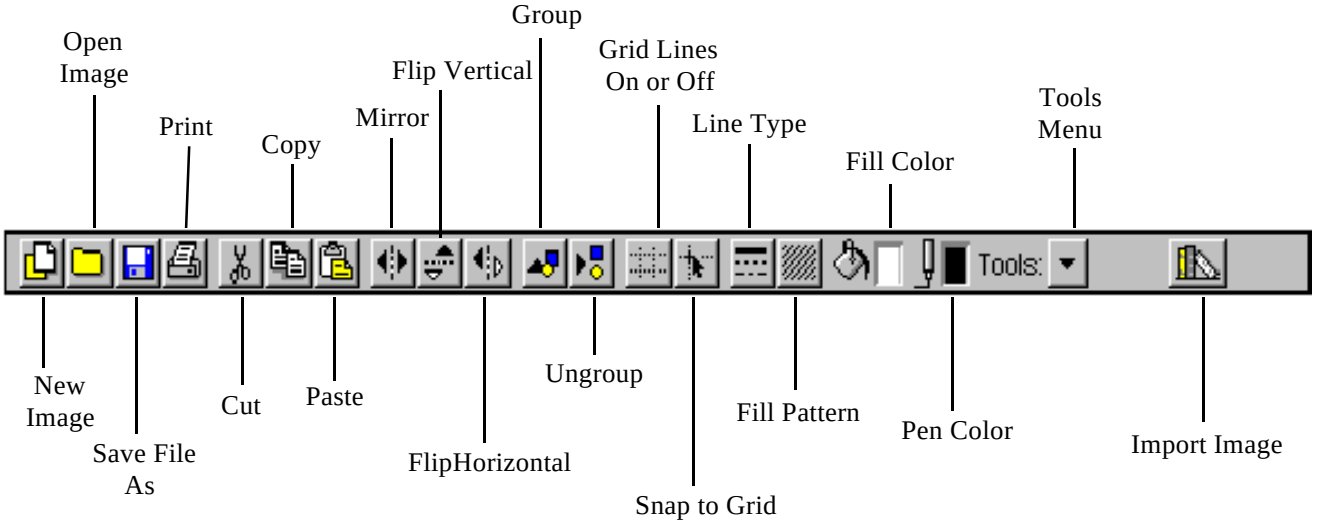
This section of the Readouts window, tracks the angular movement of the cursor. It is useful when creating angular lines or rotating objects to a specific angle.



The Menu Bar (Windows)



The Menu Bar (Macintosh)



Menu Bar

The **Menu Bar** permits more efficient work within the design. The Menu Bar can be turned On or Off as necessary. To turn the Menu Bar On or Off, first pull down the **Layout** menu and select the *Menu Bar* command. Hold down Ctrl+M to turn the *Menu Bar* On or Off.

The commands, tools, and options found within the Menu Bar can be accessed within the various menus found at the top of the screen.

The buttons and pull-down menus have been designed with minimal size. They are placed along the edge of the title bar to minimize space utilization and allow an unobstructed design environment.

If the monitor being used is smaller than the width of the Menu Bar, the program cuts off the options furthest to the right. It omits one option at a time, until the width of the Menu Bar fits within the screen size.

The following pages provide a brief definition of each option found within the Menu Bar. Refer to the Menu to access further details on selected options. The Menus provide detailed examples on how to use these options while creating a design. Refer to the *Contents* Section at the beginning of the manual for a list of commands contained within each menu.

Mirror

This function permits the selected object to be duplicated and flipped. The mirrored object is slightly offset from the original. To use the *Mirror* command, first select the object then click on the *Mirror* button within the Menu Bar. Individual objects or groups of objects can also be mirrored.

Flip Vertical

This button is used to Flip selected objects along the vertical axis. It is comparable to flipping the selected object upside down. It is possible to flip an Individual object or a group of objects.

Flip Horizontal

This button is used to Flip selected objects along the horizontal axis. It is comparable to flipping the selected object to the right. It is possible to Flip an individual object or a group of objects.

Group

This button is used to consolidate a set of selected objects as one grouped object. A grouped set of objects is treated as a single object, rather than several individual objects.

Ungroup

The Ungroup button automatically divides a grouped object into individual objects.

Lock

This button is used to Lock the characteristics and location of a selected object or groups of objects, so that they cannot be changed.

Unlock

This button is used to Unlock selected objects that were previously locked.

Join

This button is used to connect (Join) all selected lines or arcs that share an overlapping (connected) end-point.

Unjoin

This button is used to disconnect all selected lines or arcs that have been Joined together.

Eyedropper

The Eyedropper button is used to select the current color from any color within a design. When the Eyedropper button is clicked on a color within a design, that color then becomes the current color. The color selected is then displayed in the Color button.

Fill Color

The current fill color is displayed to the right of this button. To select another fill color, click once on the button to open the color picker.

Pen Color

The current pen color is displayed to the right of this button. To select another pen color, click once on the button to open the color picker.

Grid Lines On or Off

This button is used to turn the Grid Lines On or Off. This function is dependent on the current status of the Grid Lines when the button is clicked. Set the Grid spacing under the **Grid** pop-up menu found under the **Layout** menu.

Snap To Grid

This button is used to turn the Snap To Grid option On or Off.

Axis Lines On or Off

This button is used to toggle the Axis Lines On or Off. The Axis Lines represent the Origin point. This is the reference source for all Absolute coordinates and readouts.

Fit to Window

This button allows the program to automatically fit the entire design within the current window size. This is useful for the overall layout of a design.

Actual Size

Click on this button to display a design at its actual size. This is useful when the user has Zoomed In or Out and wishes to return to the Actual Size without using the *Zoom* tool or *Menu* command. To access the keyboard equivalent for Actual Size, press CTRL E. The *Menu* command is found under the **Options** menu.

New Image (Macintosh only)

This button is used to create a new image at the default settings. Selecting this button is the same as selecting the **File** menu, **New** option.

Open Image (Macintosh only)

This button displays the Open Image window. You can locate and open previously created documents.

Save (Macintosh only)

This button allows you to save the currently active file. If the file has been saved previously then this button saves the document over the original document.

Print (Macintosh only)

This button displays the print dialog box. This dialog box allows you to set printer options and print the currently active document.

Cut (Macintosh only)

This button removes selected objects from a document and places them on the clipboard.

Copy (Macintosh only)

This button places the selected objects in the clipboard while leaving the original objects in the document.

Paste (Macintosh only)

This button places the contents of the clipboard into the currently active document.

Line Type (Macintosh only)

This button allows you to select a line style for any selected objects.

Fill Patterns (Macintosh only)

This button displays the Fill Pattern Window.

Import Image (Macintosh only)

This button allows you to import a previously saved image into the currently active image.

Toolbox Pull-Down Menu

This pull-down menu is used to turn On or Off any of the toolboxes found within the program.

The Toolbox Pull-Down Menu

The Edit Toolbox

The Edit tools offer the ability to edit or change any object that has been created within the program, in addition to designs that were imported from elsewhere. When working with imported designs, verify that the design is in a vector format, NOT a raster format such as paint files. The way that the Edit tools work, is by selecting the individual or groups of points that are to be edited. Points may be selected by clicking on the individual vertex points of the object or by dragging a selection rectangle around the points to be edited.

The Edit tools offers the following types of functions: Selecting Objects, Individual point selection, Resizing of objects, Skewing objects to any angle, Rotating objects from 0 to 360 degrees, Fillet and Chamfer capabilities, Trimming of objects, Zooming and Text.

To maintain a high level of accuracy, the Edit tools can be combined with the *Snap To* tools which are also found within the program. The *Snap To* tools are designed to work in conjunction with all of the tools found within the program. Refer to the *Snap To Tools* section of this manual for more details concerning these tools.

This section of the manual offers a detailed explanation of each Edit tool capability as well as some step by step instructions on how to use the *Edit* tools.

Edit Tools and Individual Names

The following diagram shows a complete list of all the Edit tools and the proper name for each tool. It is very important to learn their proper names.

The Edit Tools

Selection Tool

The Selection tool is primarily used for repositioning selected objects to any location within the design. This tool is used to select and move individual or groups of objects anywhere within the design area.

This tool is the first tool in the Edit toolbox, and is sometimes called a *Cursor* tool. The tool works in two different fashions when selecting objects. First, click on any part of an object to highlight the object. (more objects can be added by holding down the SHIFT key when clicking on additional objects). The second method of selection is to drag a selection rectangle around the object or group of objects.

If there is a specific location that is to be entered when moving objects, do so by using Keyboard Entry. This is explained later in this section under the subheading *Keyboard Entry*.

Selecting Objects

One of the most common ways of selecting objects is to click on a part of the object. After clicking on the object, the vertex points become highlighted. The object may then be relocated anywhere on the screen by clicking and dragging on a line in the object. When dragging the object a ghost line appears, which represents the boundary lines of the object or objects selected.

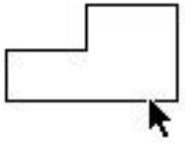
NOTE: Perfect horizontal or vertical movement can be achieved by holding down the SHIFT key while dragging the object.

Example 1

The following example provides further explanation on how to use the Selection tool by clicking on an object.

Step 1

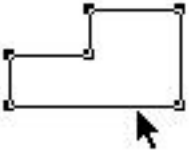
Create a simple object as shown below, then place the cursor on top of the object as shown and click the mouse button once.



Click mouse once

Step 2

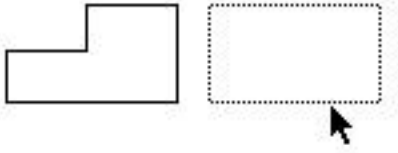
Once the mouse button is clicked, the vertex points of the object are highlighted as shown below.



— Selected vertex points

Step 3

Now that the object is selected, it can be moved. While keeping the cursor on top of the selected object hold down the mouse button and begin dragging the object to a different location. The ghost lines then appear to help align the object with other objects. Perfect horizontal or vertical movement can be acheived by holding down the SHIFT key while dragging the object.



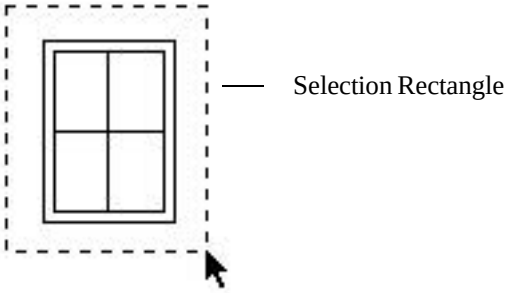
Ghost lines

Example 2

The following example provides further explanation on how to use the Selection tool with a selection rectangle.

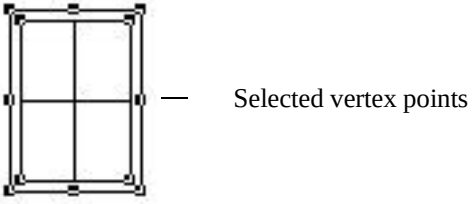
Step 1

Create a simple object as shown below, then click on the Selection tool within the Edit toolbox. Now place the cursor near the object (as shown below) and hold down the mouse button while dragging the mouse. This creates the selection rectangle and therefore selects any objects within the selection rectangle. This method is suited for selecting multiple objects to be moved together.



Step 2

Once the objects have been selected, the vertex points of the objects are highlighted as shown below.



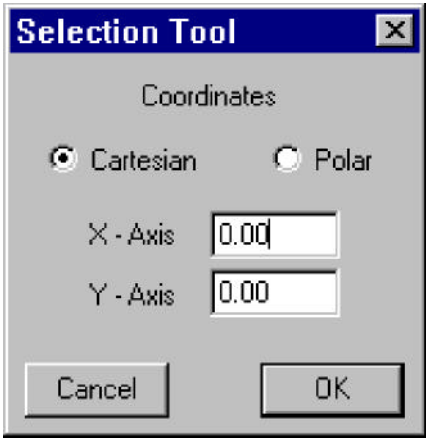
Step 3

Now that the object is selected, it can be moved. While keeping the cursor on top of the selected object hold down the mouse button and begin dragging the object to a different location. The ghost lines then appear to help align the object with other objects. Perfect horizontal or vertical movement can be acheived by holding down the SHIFT key.

Keyboard Entry with Selection Tool

When using the Selection tool there is the option of using Keyboard Entry. This is advantageous because it allows the user to enter the coordinates for a specific location to move an object to. To access the Selection tool Dialog box, double click on the Selection tool icon.

To move an object, it must first be selected. If no object is selected, snap to a point on the object, then press ENTER. This allows the program to determine which point of the object should be placed at the new coordinates. When snapping to the required point be sure to use the Snap To tools to assure complete accuracy when moving the object. The Dialog box offers a choice of Cartesian or Polar coordinates.



The Selection Tool Dialog box

NOTE: The typed coordinates are relative to the object's current position.

Point Select Tool

The Point Select tool is primarily used for repositioning individually selected points to any location within the design. To maintain perfect accuracy in working with the *Point Select* tool, combine this tool with the Snap To tools. This tool is the second tool in the Edit toolbox, and is also known as a *Cursor* tool.

There is the option to specify any X, Y coordinates by using Keyboard Entry rather than the mouse. *Keyboard Entry* is explained in greater detail later in this section.

Selecting points with the Point Select tool

This tool works in two different ways when selecting objects. First click on any individual vertex point within an object; therefore highlighting just that point. (Individual points may be added by holding down SHIFT when clicking on additional vertex points.)

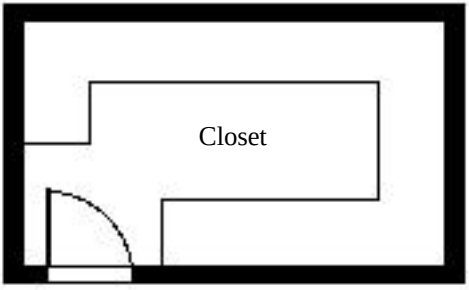
Selection Rectangle

The second method of selection is to drag a selection rectangle around the individual point or groups of points to be selected.

When the points are highlighted, place the cursor on top of a selected point within the object, hold down the mouse and move these points to any location on the screen. When dragging these points, a ghost line appears representing the boundary lines of the points selected. Perfect horizontal or vertical movement can be achieved by holding down SHIFT after the dragging of the selected points has started.

Example

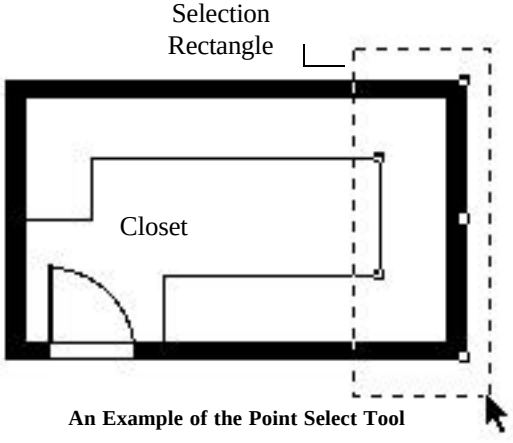
The following example shows how the Point Select tool can help in editing designs. A simple closet layout has been created for this purpose. It is possible to use any design or create a similiar design to the one below.



An Example of a Closet Layout

Step 1

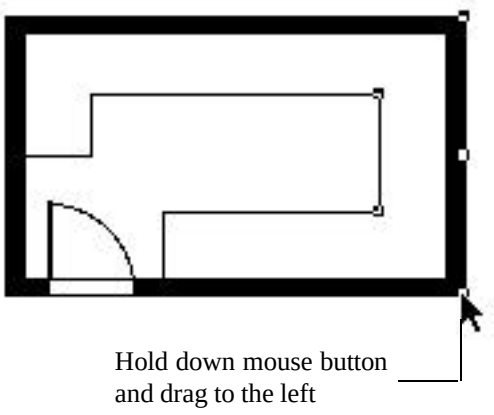
After a design is created, select the *Point Select* tool from the Edit toolbox. Now drag a selection rectangle around the end of the closet as shown below. Once selected, the individual points are highlighted, and can be moved as required.



An Example of the Point Select Tool

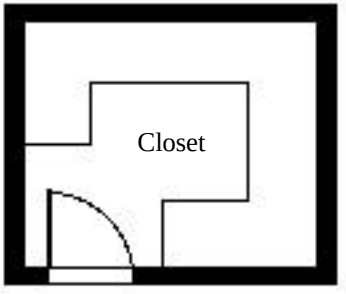
Step 2

Place the Point Selection tool cursor on top of a highlighted point as shown below. After placing the cursor on top of the point, hold down the mouse button and drag the selected points to the left. Watch the readouts at the bottom of the screen for precise movement. Remember that SHIFT can be held down to constrain movement horizontally.



Step 3

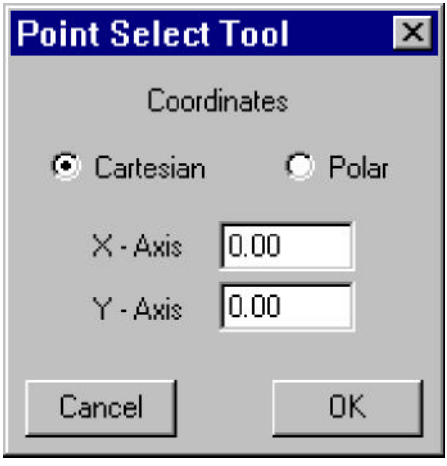
Once the desired distance has been reached, release the mouse button and the design is automatically updated to reflect the changes. This is a very useful tool for updating previously drawn designs.



Keyboard Entry with Point Select Tool

The Point Select tool has the option of using Keyboard Entry for a very precise movement. This means that if it is necessary to move any point or group of points to a specific location (XY Coordinates), it can be done easily. To access the Point Select tool Dialog box, double click on the Point Select tool icon.

It is important to remember that when moving selected points using Keyboard Entry, the points to be moved must be selected first. After selecting the points, snap to one of the selected points, then press ENTER. This allows the program to determine which point of the object should be placed at the new coordinates. When snapping to the desired point be sure to use the Snap To tools within the Snap To toolbox for complete accuracy when moving the selected points. The Dialog box allows a choice of Cartesian or Polar coordinates, as well as Relative or Absolute reference coordinates.



The Point Select Dialog box

NOTE: The Absolute coordinates entered are in reference to the point of origin of the working design. The program initially sets the origin to the lower left-hand corner of the design.

Resize Tool

When using the Resize tool the horizontal and vertical widths of any object can be changed. Also when using the Resize tool either the mouse or Keyboard Entry can be used.

In using the mouse, click and drag the corner of the object to be resized. The readouts at the bottom of the screen can be used to maintain any specific size. While resizing objects, ghost lines appear for visual representation.

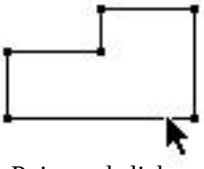
If the Keyboard Entry method is used, first select the object and then double click the Resize tool icon to access the Keyboard Entry Dialog box. Once the Dialog box appears enter the new width and height of the object.

Selecting an object for resizing

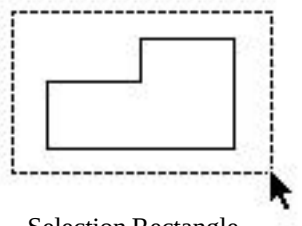
First select the Resize tool from the Edit toolbox, then place the cursor on the object and click the mouse button. The points of the object then become highlighted and are ready for resizing.

If there are multiple objects that make up one design and all of them are to be resized together, drag a selection rectangle around those objects to highlight the object's vertex points.

Example



Point and click

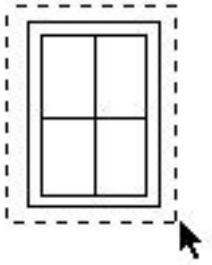


Selection Rectangle

The following example shows how to resize the window that was just created. Notice that the window is made up of multiple objects such as lines and rectangles. Therefore, to select the window, drag a selection rectangle around the entire window as shown below, to highlight the points within the window.

Step 1

First, drag the selection rectangle around the window to highlight the points within the window.

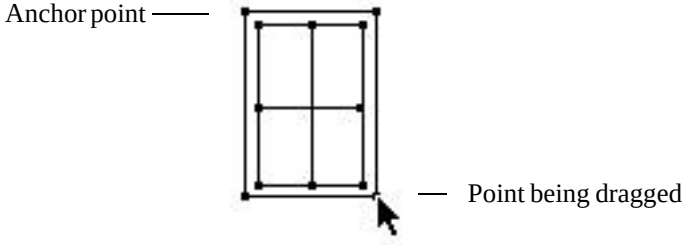


The Selection Rectangle

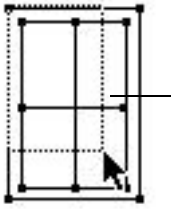
Step 2

Be sure that the Resize tool is selected, then place the cursor on top of the corner point as shown below. This is the point to be dragged to resize the window. The opposite point is used as the anchor point when resizing. This means that the point stays in its original position, while dragging the other point.

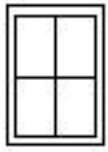
Step 3



Now hold down the mouse button and begin dragging. The ghost lines appear as a visual reference for resizing the window. Watch the readouts at the bottom of the screen to maintain a specific size. Release the mouse button at the desired size.



Drag the mouse to the desired size

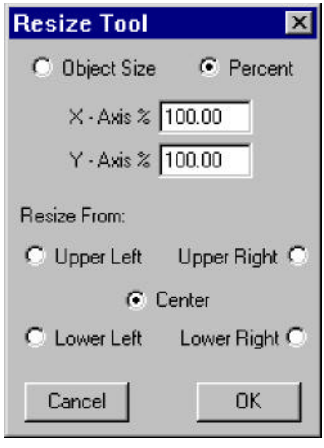


Window after resizing

Keyboard Entry with Resize Tool

When resizing objects there is the option to use numeric Keyboard Entry. This is very helpful when an object is to be resized to a specific size.

To access the Resize tool Dialog box, double click the Resize tool icon. When using Keyboard Entry with the Resize tool, an object must first be selected. The Resize Dialog offers the option to resize by Percentage or by Object size. The Dialog box also specifies from which part of the object the resizing is to be done.



The Resize Tool Dialog box

Skew Tool

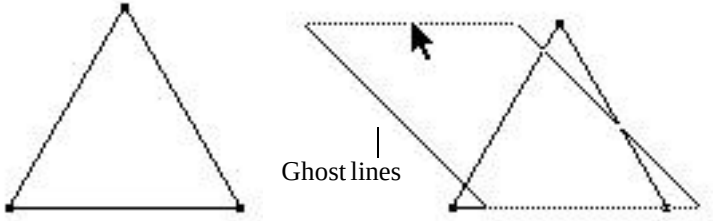
This tool is designed to give any selected object a skewed angle. When this tool is used, the selected object takes on an angular shape.

The Skew tool supports the mouse as well as numeric Keyboard Entry. When using the mouse, first select the entire object so that the vertex points are highlighted. Next place the mouse cursor on top of a highlighted vertex point; then click and drag to a desired angle. When using the mouse a ghost line appears to give a visual representation of the object as it is being skewed. Watch the readouts at the bottom of the screen to obtain the necessary angle.

Example

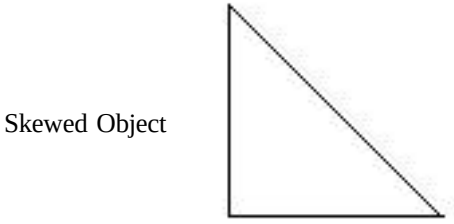
Step 1

Create a shape similiar to the one shown below. Then select the object so that the vertex points are highlighted. Now place the cursor on the top point of the triangle, then click and drag the cursor to the left as shown below.



Step 2

At the desired distance or angle release the mouse button. This example shows a triangle skewed by a 30 degree angle.



Keyboard Entry with Skew Tool

The second method of using the Skew tool is with the numeric Keyboard Entry. Using Keyboard Entry is useful when an object is to be skewed to a specific angle.

Before accessing the Skew tool Keyboard Entry Dialog box, select an object. Then double click on the Skew tool icon and the Dialog box appears. Type in the required angle and click the OK button. The selected object is automatically updated to reflect the new changes. The Dialog box offers the option to choose which direction to skew the object. Choose from Up, Down, Left or Right. The Distance entry determines how far the object is skewed.

The Skew Tool Dialog box

Rotate Tool

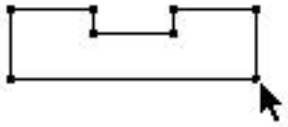
By using the Rotate tool any object can be rotated from 0 to 360 degrees. All objects are rotated from their centers. Rotating objects can be performed by two methods; the mouse or Keyboard Entry.

The following example shows how to use the Rotate tool with the mouse.

Example

Step 1

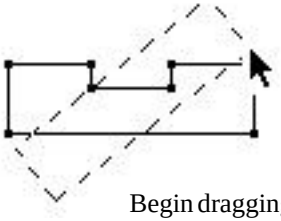
First create an object similiar to the one shown below. Then select the object so that the vertex points are highlighted.



Highlight object

Step 2

Place the cursor on the corner vertex point, then click and drag the mouse in a counter clockwise direction. Once the dragging is started, a ghost line appears representing the boundaries of the object. At the desired rotation release the mouse button. Watch the readouts at the bottom of the screen for the degree of rotation.



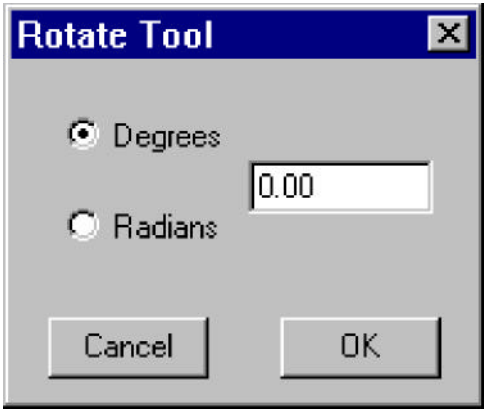
Begin dragging the mouse

Rotated object

Keyboard Entry with Rotate Tool

There is the option to rotate any object or group of objects by using numeric Keyboard Entry. An object must be selected before accessing the Keyboard Entry Dialog. After selecting an object, double click on the Rotate tool icon. Once the Dialog box appears any degree of rotation can be entered. There is also an option to choose either Degrees or Radians. After the correct information is entered click on the OK button.

NOTE: The program allows the rotation of individual objects or groups of objects when using Keyboard Entry. Be sure to have all the necessary objects selected before accessing the Dialog box.



The Rotate Tool Dialog box

Fillet Tool

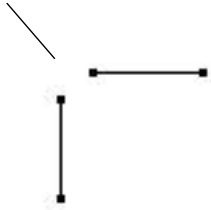
When using the Fillet tool, a radius can be added to any object. Remember that the Fillet tool works with corner points of an object. What this means is that before a fillet can be applied with the Fillet tool, the selected corner point must be joined together or made of enclosed objects such as rectangles, squares, polygons, etc. The example shown below gives a further explanation.

Example 1

With the two lines shown below, the Fillet tool cannot be used to apply a radius between the end-points of the two lines. In this instant the Arc tool from the Draw toolbox would be used, then snap each end of the arc to the end-points of the two lines. For more information concerning the Arc tool refer to the *Draw Tools* section under the Arc tool subheading.

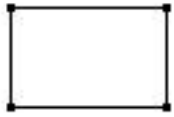
Arc tool

Use Arc tool to apply a radius between two separate end-points



The rectangle shown below is the type of object that the Fillet tool is used with. The reason the Fillet tool works on the rectangle is because the corner points are joined together and make up one vertex point rather than two as shown above.

Apply the Fillet tool to corner vertex points which are joined



Using the Fillet tool

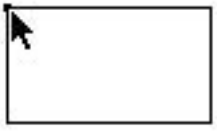
The following example guides the user through the proper usage of the Fillet tool. This tool works with the mouse and allows the user to enter a precise radius with the numeric Keyboard Entry.

Example 2

Step 1

First, create a shape similiar to the one shown. The object shown below has been created with the Rectangle tool. After creating a shape, select the Fillet tool from the Edit toolbox and click on the corner vertex point as shown below.

Click on corner vertex point



Step 2

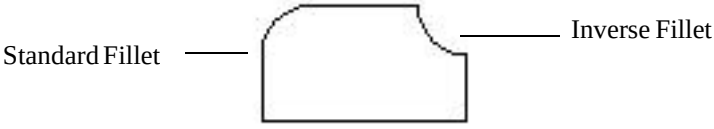
Now hold down the mouse button and begin dragging the cursor towards the center of the rectangle as shown below. Watch the readouts at the bottom of the screen to maintain a specific radius. At the desired radius, release the mouse button. The design is automatically updated to reflect the filleted corner.

Drag cursor towards the center of the rectangle

Release mouse button to display the radius

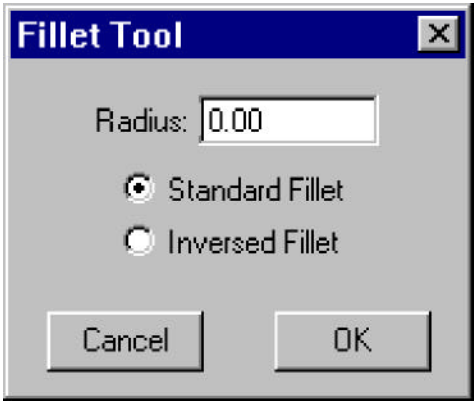
Keyboard Entry with Fillet Tool

The Fillet tool has the ability to be used by typing in the radius with numeric Keyboard Entry. The Dialog box is used to type in the exact radius and to select from a *Standard* fillet or an *Inverse* fillet.



The Fillet tool can select points in two ways. One is to place the Fillet tool cursor on top of the corner point and click the mouse button one time. The second method is to use the Fillet tool and drag a selection rectangle around the point(s) to apply a fillet to.

When using Keyboard Entry with the Fillet tool, the point(s) must be selected before activating the Dialog box. To access the Fillet tool Dialog box first select the points then double click on the Fillet tool icon. After the correct radius have been entered and the standard or inverse fillet chosen, click the OK button.



The Fillet Tool Dialog box

Chamfer Tool

When using the Chamfer tool, an angular corner can be applied to any object. Remember that the Chamfer tool works with the corner points of an object. What this means is that before a chamfer can be applied with the Chamfer tool, the corner point selected must be joined together or made of enclosed objects such as rectangles, squares, polygons, etc. The example below offers further explanation.

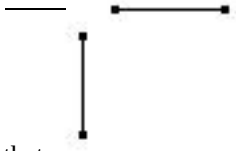
Example 1

Line tool

The Draw Toolbox

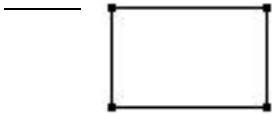
With the two lines shown below, the Chamfer tool cannot be used to apply a radius between the end-points of the two lines. In this instance the Line tool from the Draw tools is used, then snap each end of the line to the end points of the separated lines. For more information concerning the Line tool refer to the *Draw Tools* section under the Line tool subheading.

Use Line tool to apply a chamfer between two separate end-points



The rectangle shown below is the type of object that [illegible] Chamfer tool. The reason the Chamfer tool works on the rectangle is because the corner points are joined together and make up just one vertex point rather than two as shown above.

Apply the Chamfer tool to corner vertex points which are joined



Using the Chamfer Tool

The following example goes through the proper usage of the Chamfer tool. This tool works with the mouse and enables the user to enter a precise length and angle of the chamfered corner, when using the numeric Keyboard Entry.

Example 2

Step 1

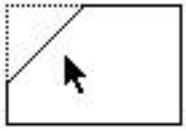
First create a shape similiar to the one below. The shape below has been created with the Rectangle tool. After creating a shape, select the Chamfer tool from the Edit toolbox and click on the corner vertex point as shown below.

Click on corner vertex point

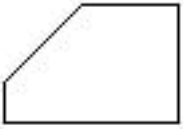
Step 2

Now hold down the mouse button and begin dragging the cursor towards the center of the rectangle as shown below. Watch the readouts at the bottom of the screen to maintain a specific length. At the desired chamfer, release the mouse button. The design is automatically updated to reflect the chamfered corner.

Drag cursor towards the center of the rectangle



Release mouse button to display the chamfer

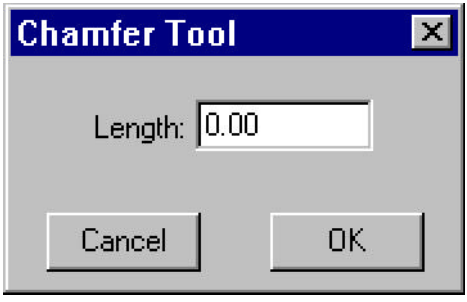


Keyboard Entry with Chamfer Tool

The Chamfer tool has the ability to be used by typing in the length of angle with numeric Keyboard Entry. The Dialog box allows the user to type in the length of the chamfer.

The Chamfer tool can select points in two ways. One is to place the Chamfer tool cursor on top of the desired corner point and click the mouse button once. The second method is to use the Chamfer tool and drag a selection rectangle around the point(s) to which a chamfer is to be applied.

When using Keyboard Entry with the Chamfer tool the point(s) must be selected before activating the Dialog box. To access the Chamfer tool Dialog box first select the points then double click on the Chamfer tool icon. After the correct length has been entered, click the OK button.



The Chamfer Tool Dialog box

Trim Tool

The Trim tool is useful for cutting objects or trimming away segments of objects that would otherwise require the user to redraw, resize or edit in some fashion. The Trim tool quickly assists in the cutting of lines, rectangles, polygons as well as the outlines of arcs or circles.

When the Trim tool is selected from the Edit toolbox, the cursor turns into a crosshair. When the crosshair is placed on a line and the mouse button is clicked, the Trim tool inserts a vertex point therefore breaking the line.

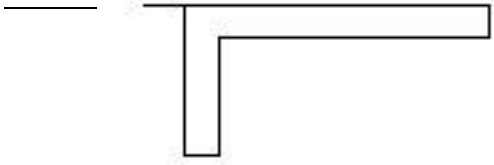
The following example explains how to properly use the Trim tool.

Example

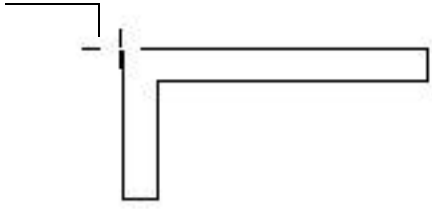
Step 1

First select the line to be trimmed and then select the Trim tool from the Edit toolbox and place the cursor at the required point to trim the excess line, as shown below. Then click the mouse button.

Excess hangover to be trimmed

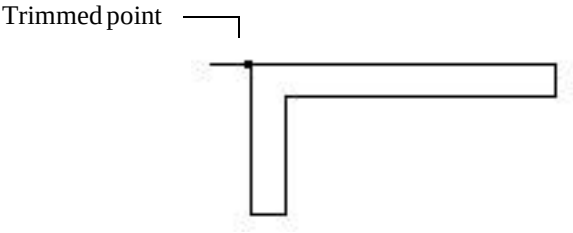


Place cursor at trim point and click mouse



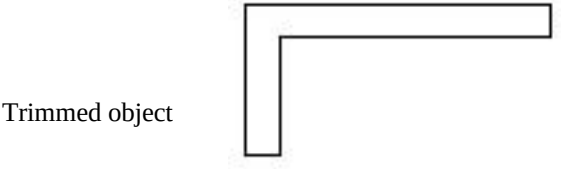
Step 2

Once the mouse has been clicked, a highlighted vertex point appears. This indicates that the program has trimmed the excess line at the highlighted point.



Step 3

After trimming the line as explained in the previous steps, the excess line can then be deleted. To delete the excess line, first select the Selection tool from the Edit toolbox and click once on the excess line, then press DEL.



NOTE: Be sure that when DEL is pressed, no other objects are selected simultaneously. If any objects other than the trimmed line are selected, all the selected objects are removed.

Zoom + icon

Zoom - icon

Regional Zoom

Zoom Tool

The Zoom tool is used to change the level of enlargement or reduction of the drawing view. It would be the same as using a magnifying glass when looking at a design. It is very useful to zoom in on a drawing for a close view of details or to zoom out for an overview of the drawing layout.
The program offers several ways to use the Zoom tool for zooming In or Out. The percentage of zoom can also be set within the Preference Dialog found under the **Edit** menu.

When the program is started initially, the Zoom tool defaults to a plus sign in the center of the icon. This indicates that this tool is set to zoom in. The plus sign can be changed to a minus sign by holding down the SHIFT (or OPTION key on a Macintosh)***, after*** the Zoom tool is selected, thus allowing the user to zoom out.

Another way to use the Zoom tool is to click and drag with the Zoom tool, therefore creating a Regional zoom. This means that the area within the selection rectangle created by dragging the Zoom tool, magnifies to the full screen size.

Zooming In

Select the Zoom tool from the Edit toolbox and place it over the point to zoom in on, then click the mouse button. The percentage of zoom can be set, which determines the degree by which the screen zooms each time the mouse is clicked. (The Zoom Percentage setting is found in the *Preference* Dialog under the **Edit** menu.)

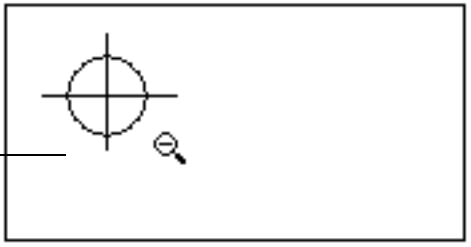
Place Zoom tool at the point to zoom in at

Zooming Out

Select the Zoom tool from the Edit toolbox and place it over the point to zoom out. The percentage of zoom can be set, which determines the degree by which the screen zooms each time the mouse is clicked. The Zoom Percentage setting is found in the Preference Dialog under the **Edit** menu.

Once the Zoom tool is selected, hold down SHIFT (or OPTION on a Macintosh) to change the magnifying glass from a **+** (plus) to a **-** (minus). Now click the mouse button and the screen zooms out. Repeat this process as many times as necessary.

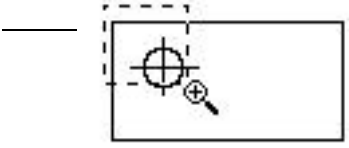
Place Zoom tool
at the point
to zoom out



Regional Zoom

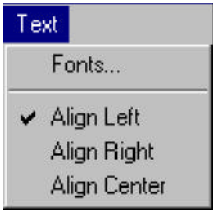
To use this method of zooming, first select the Zoom tool from the Edit toolbox. Now hold down the mouse button and drag a selection rectangle to select the part of the design to be magnified. The area identified within the selection rectangle is displayed to the maximum size of the screen. The magnification factor is determined by the size of the area selected.

Drag a selection rectangle



Text Tool

This tool can be used to add captions and paragraphs of text to any drawing. The user can mix fonts, font sizes and font styles.



The Text Menu

Before text is typed within a design, choose the font, font size, font style, and text alignment that are appropriate for the document. The program supports any font that is installed within the operating system. **KeyCAD Pro** also supports TrueType and PostScript fonts. Text can be aligned along the left or right margins, or centered between margins.

KeyCAD Pro's pop-up dialog allows fast access when selecting the desired text. To access the pop-up dialog, pull down the **Text** menu and choose the *Fonts*... option and select from options such as: *Font*, *Size*, or *Style*.

Adding Text to a document

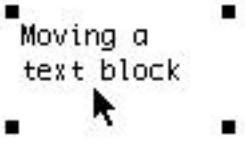
1. Choose a font from the **Font** menu.
2. Choose a font size from the **Size** menu.
3. Choose a font style from the **Style** menu
4. Choose align Left, Right, or Center from the **Text** menu.

After the choices are made, select the Text tool from the Edit toolbox and add text to any part of the design. **KeyCAD Pro** defaults to Arial 12-point text with left alignment when the program is initially started.

Moving Text Blocks

The creation of a text block is used to move the text to a preferred location. To do this, select the Selection tool from the Edit toolbox, and click on the text that has been already created.

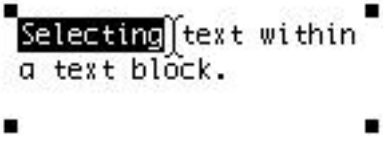
Click on the text to highlight text block



Editing Text within a Text Block

Text can be selected within a text block for editing purposes. To select part of a text object, first select the Text tool from the Edit toolbox. Then select the letters or words to be edited by dragging through the text. The selected text becomes highlighted and can now be edited.

Drag I-beam through the text to highlight individual words or letters



Deleting a Text Block

An entire text block can be deleted at one time. First, select the Selection tool from the Edit toolbox. Then, click once on the text block to be deleted and press DEL.

The *Draw* Tools

The *Draw* tools are used in the construction of basic shapes, that can be combined together to create complex shapes. The *Draw* tools consist of rectangles, squares, circles, ovals, polygons, multigons, arcs, lines, and free-formed splines. The *Draw* toolbox automatically appears to the left side of the window when the program is first started, but the toolbox can be relocated to any position on the screen.

To select a tool for drawing with the mouse, click once on the required tool, then click and drag the mouse to construct the selected object.

To access the *Draw* tools Numeric Keyboard Entry Dialog, double click on the tool icon.

These two methods of selection work the same for all *Draw* tools.

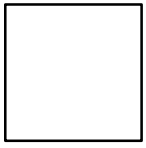
The *Draw* toolbox can be turned On or Off for convenience. Toolboxes can be turned On or Off under the Tools pull-down menu on the Menu Bar, or the Toolbox pop-up menu found under the **Windows** menu.

Combining *Draw* Tools with *Snap To* Tools

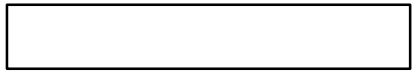
The *Draw* tools are designed to work in conjunction with the *Snap To* tools. This allows the user to precisely snap to end-points, midpoints, etc. when starting and finishing the objects found in the *Draw* toolbox.

Rectangle / Square Tool

The Rectangle / Square tool is designed to draw both rectangles and squares. **KeyCAD Pro** defaults to the Rectangle tool. To draw squares, select the tool icon and hold down SHIFT when using the tool, to draw perfect squares.



Squares



Rectangles

Rectangle Tool combined with Snap To Tools

The Rectangle / Square tool can be used with the Snap To tools, allowing the rectangle or square to start or end from another object's center, midpoint, end-point, percentage point, etc. Remember that any rectangle or square can start or end from *any* of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create selected objects.

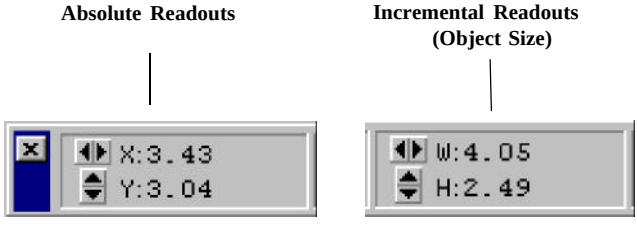
The rectangle or square can be drawn from the objects center point. The *Draw from Center* can be selected as an option under the **Layout** menu. When this option is selected, the tool draws the object double the size of the Incremental Readouts or the distance the mouse (width and height) is dragged.

Draw from Center **is located under the Layout menu**

Drawing a Rectangle/Square with the Mouse

The following instructions explain how to construct a rectangle or square, while using the mouse.

1. Select the Rectangle/Square tool from the *Draw* toolbox.
2. Place the cursor at the required start point, and click and drag the mouse until the desired size is obtained. Watch the readouts at the bottom of the screen for the object size. The **W** indicates the objects width, and the **H** indicates the object height.



The Absolute coordinates display the object size added to the Absolute coordinates of the start point. This means that the coordinates are calculated from the Origin point (Axis Lines Intersection).

The Incremental readouts display the object size in relation to the start point.

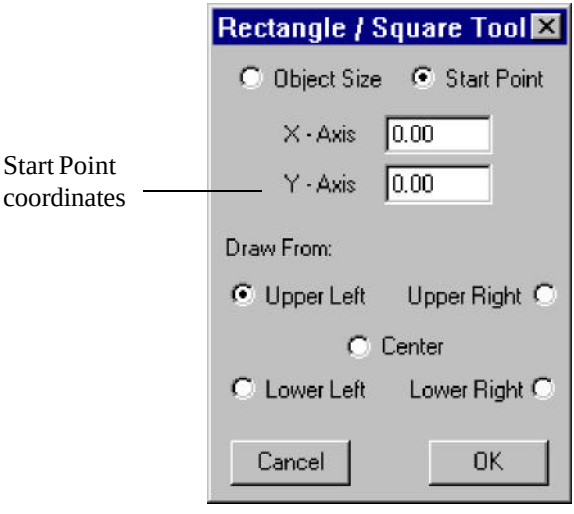
Using Keyboard Entry with Rectangle/Square Tool

The Rectangle/Square tool Numeric Keyboard Dialog can be selected by double clicking on the tool icon. A Dialog box appears, allowing the user to numerically enter the object size and start point.

After double clicking on the tool, the Numeric Entry Dialog box appears, it disregards the mouse actions and the object is drawn to the numeric information entered in the Dialog box. The object width and height as well as the start point of the rectangle or square can be set.

Start Point

When setting the start point, first select the Start Point button, notice that the Dialog box information changes to coordinate positions. When the coordinate positions are entered, the start point can also be set to be at a selected point on the object such as, Upper Left, Upper Right, Lower Left, Lower Right, or Center point of the rectangle or square. The start point is always relative to the Absolute Point of the document.

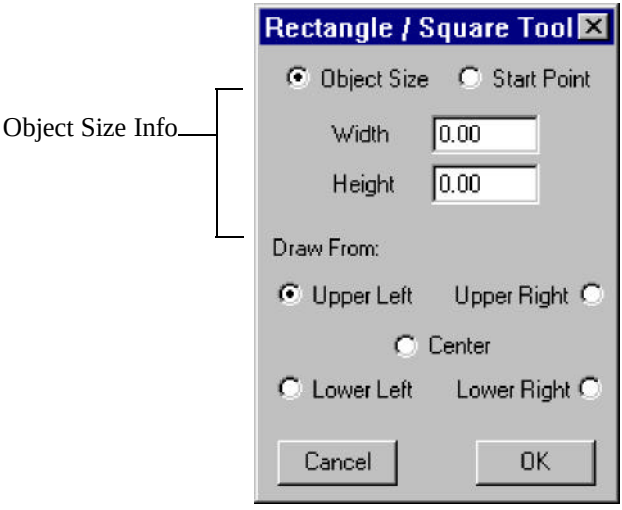


The Rectangle/ Square Tool Dialog box

Object Size

When the Object Size button is clicked the Dialog changes, so that the object's width can be entered. Remember that the Object Size information is taken from the selected *Draw* From and Start Point coordinates.

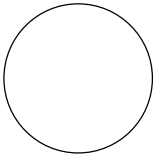
If the *Draw* From Center option is selected, the width and height are reduced to half the actual size.



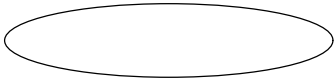
The Object Size Dialog box

Circle / Oval Tool

The Circle / Oval tool is designed to draw both circles and ovals. **KeyCAD Pro** defaults to the Oval tool. To draw perfect circles, select the tool icon and hold down SHIFT while using the tool to draw perfect circles.



Circle



Oval

Circle/Oval Tool combined with Snap To Tools

The Circle / Oval tool can be used with the Snap To tools, therefore allowing the user to start or end the objects from another object center, midpoint, end-point, percentage point, etc. Remember that any circle or oval can start or end from **any** of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create selected objects.

A circle or oval can also be drawn from the object center point. The *Draw* From Center option can be selected from under the **Layout** menu. When this option is selected, the tool draws the object and doubles the size of the Incremental Readouts or the distance the mouse (width and height) is dragged.

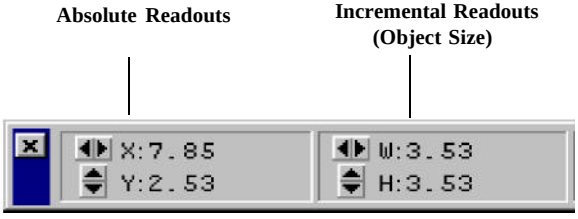
Draw from Center **is located under the Layout menu**

Drawing a Circle or Oval with the Mouse

The following instructions explain how to construct a circle or oval, while using the mouse.

1. Select the Circle/Oval tool from the *Draw* toolbox.

2. Place the cursor at the desired start point and click and drag the mouse until the desired size is reached. Watch the readouts at the bottom of the screen for the object size. The **W** indicates the object width, and the **H** indicates the object height.



The Absolute coordinates display the object size added to the Absolute coordinates of the start point. This means that the coordinates are calculated from the Origin point (Axis Lines Intersection).

The Incremental Readouts display the object size in relation to the start point.

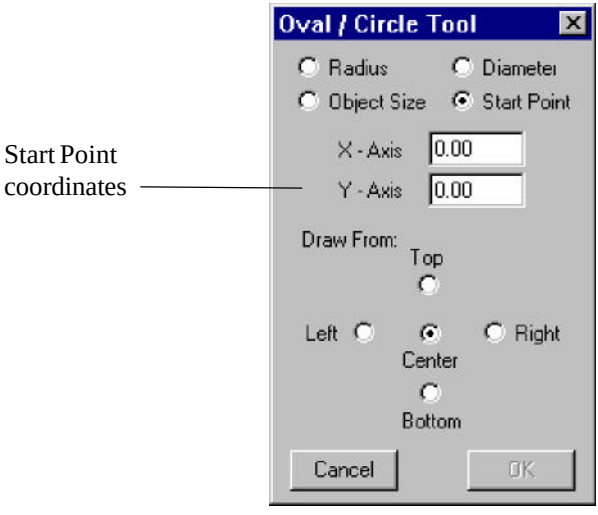
Using Keyboard Entry with Circle/Oval Tool

The Circle/Oval tool Numeric Keyboard Entry Dialog box can be accessed by double clicking on the tool icon. A Dialog box appears, allowing the object size, start point, radius and diameter to be entered.

After double clicking on the tool and the Numeric Keyboard Entry Dialog appears, the Dialog box disregards the mouse actions and the object is drawn to the numeric information entered in the Dialog box.

Start Point

When setting the start point, first select the Start Point button, notice that the Dialog box info changes to coordinate positions. When the coordinate positions are entered the start point can also be set to be at a selected point on the object such as Top, Left, Right, Center and Bottom of the circle or oval. The start point is always relative to the Absolute Point of the document.



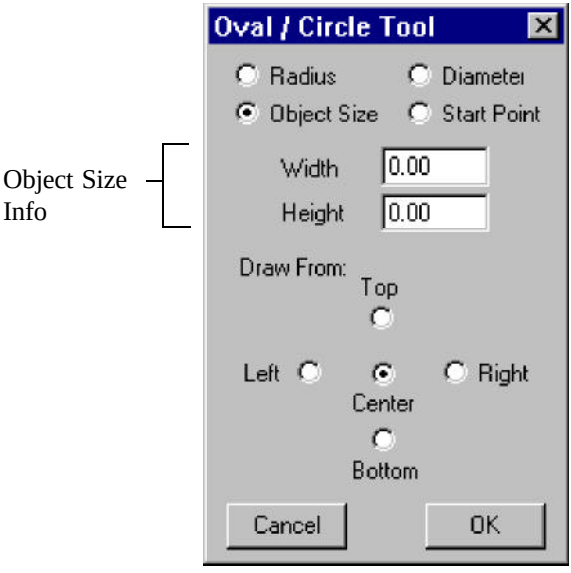
The Oval / Circle Tool Dialog box

Object Size

When the Object Size button is clicked the Dialog changes, so that the object width and height can be entered. Remember that the Object Size information is taken from the selected *Draw* From and Start Point coordinates.

The Circle/Oval tool has the options of using Radius, Diameter or Object Size for creating the object. Select one of the options after the Start Point and *Draw* From options have been set.

If the *Draw* From Center option is selected, the width and height of the circle or oval are reduced to half of the actual size.

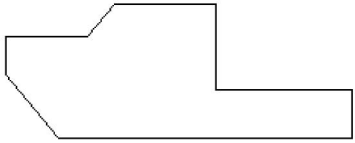


The Oval / Circle Tool Dialog box

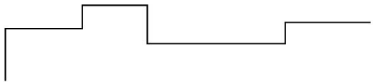
Polygon Tool (Open and Closed)

The SHIFT key together with the Polygon tool constrain each segment to a perfect Horizontal or Vertical line.

The Polygon tool is designed to draw Open or Closed polygons, that can contain many individual sides. Polygon shapes are often closed shapes, which means that the last line connects with the beginning point of the polygon. Closed polygons include triangles, parallelograms, stars, octagons, and many other shapes.

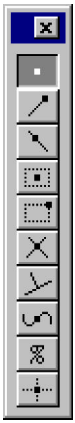


A Closed Polygon



A Open Polygon

Polygon Tool combined with Snap To Tools



The Polygon tool can be used with the Snap To tools, therefore allowing the user to start or end polygons from another object's center, midpoint, end-point, percentage point, etc. Remember that any polygon can start and end from **any** of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create selected polygons.

To use a Snap To tool with the polygons start point, first select the required Snap To tool, then select the Polygon tool and place the start point. To finish the polygon at a specific Snap To tool, press the Tab key to change the Snap To tool while drawing the polygon, and double click at the desired end-point.

Creating a Polygon with the Mouse

1. Select the Polygon tool from the *Draw* toolbox, then place the crosshair cursor at the desired start point.

2. Click the mouse button once, to set the first point. Then drag the mouse to the second point location and click the mouse button. Repeat this process as many times as necessary.

3. When the polygon is completed, double click the mouse button to end the polygon. If the end-point is within a 10 pixel radius of the start point, the polygon is automatically closed. If the end point is outside of the 10 pixel radius, the polygon ends at the point clicked, creating an open polygon.

Once either an Open or Closed Polygon have been created, it can be filled with any of the fill patterns available. The line type and size that make up the boundaries of the polygon can also be changed. Lines can be changed from under the **Layout** menu, in the pop-up menu called Lines.

The Lines pop-up menu

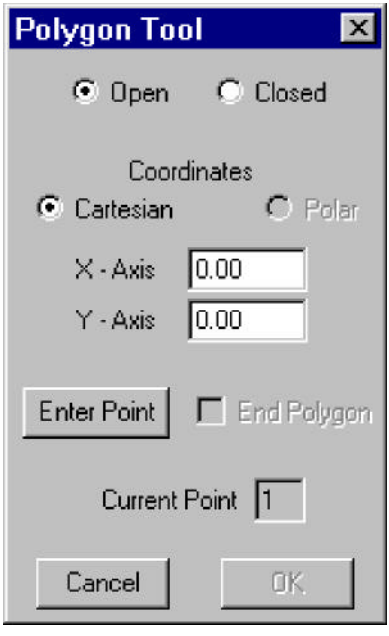
The color of the polygon can also be changed by selecting the polygon and then choosing a new color from the color dialog. If the polygon has been filled and the colors changed, the fill then reflects the color changes, therefore allowing for colored fills.

Using Keyboard Entry with the Polygon Tool

When using the Polygon tool the user has the option of using numeric Keyboard Entry, instead of the mouse. To access the Dialog box, double click on the Polygon tool icon.

Within the Polygon tool Dialog box, choose from Open or Closed polygons and set the start point of the polygon and the number of points within the polygon. Choose from Cartesian or Polar coordinates. The Dialog box also displays which point is being entered, as well as the total number of points making up the polygon.

After entering a point coordinate press the Enter Point button to place and add the new point to the polygon. Continue adding points as necessary. At the desired number of points, click on the End Polygon button,to automatically end the polygon. Click the OK button to close the Dialog box.

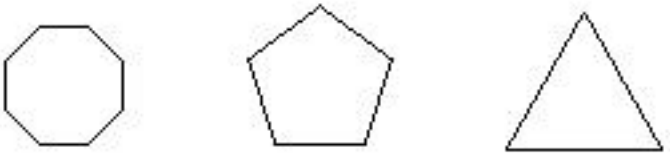


The Polygon Dialog box

Multigon Tool

The Multigon tool is designed to draw objects that contain any number of sides, with each side of equal length. This tool is used to create octagons, pentagons, triangles, etc. This tool saves a lot of time in creating various multigons. Other programs usually use the Polygon tool for creating multigons.

KeyCAD Pro defaults the Multigon tool to contain five even sides. The number of sides can be changed in the Preference Dialog, which is found under the **Edit** menu.



Different objects created with the Multigon tool

Multigon Tool combined with Snap To Tools

The Multigon tool can be used with the Snap To tools, therefore allowing the user to start or end the multigon from another object center, midpoint, corner, percentage point, etc. Remember that any multigon can start or end from **any** of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create the multigon.

The multigon can also be drawn from the object's center point. The *Draw From Center* option can be selected from the **Layout** menu. When this option is clicked, the tool draws the multigon double the size of the Incremental Readouts or the distance the mouse (width and height) was dragged.

Draw from Center **is located under the Layout Menu**

Drawing a Multigon with the Mouse

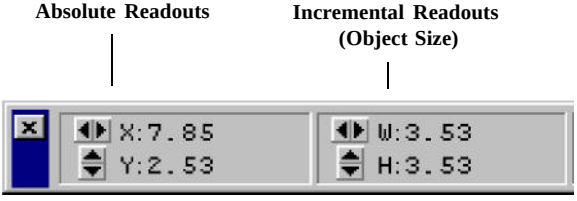
The following instructions explain how to construct a multigon, while using the mouse.

1. Set the number of required sides for the multigon. The number of sides can be set within the Preference Dialog under the **Edit** menu.

2. Second, select the Multigon tool from the *Draw* toolbox.

3. Now, place the cursor at the required start point, and click and drag the mouse until the desired size is obtained. Watch the readouts at the bottom of the screen for the multigons width. The **W** indicates the object width, and the **H** indicates the object height.

NOTE: The start and end-point of the Multigon can snap to any of the Snap tool options.



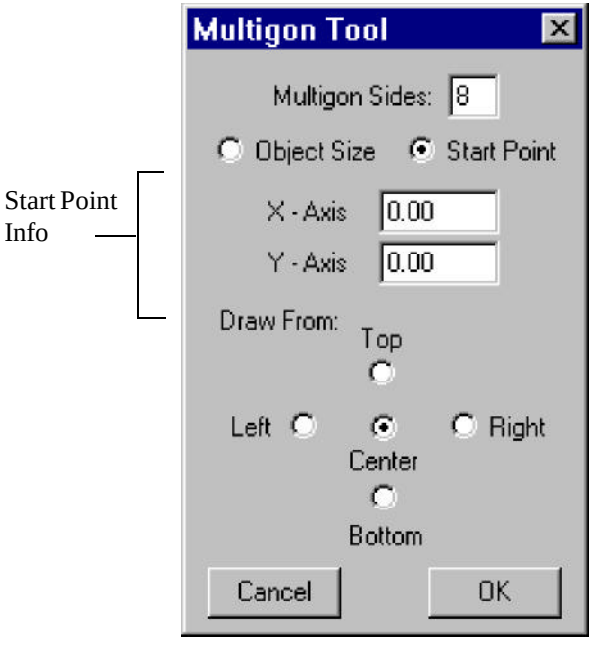
The Absolute coordinates display the object size added to the distance the cursor is from the absolute point of the document. The Absolute Readouts track the mouse movement in relation to the Origin point. This means that the coordinates are calculated from the Origin point (Axis Lines Intersection).

The Incremental Readouts display the object size in relation to the start point or the distance the mouse is dragged.

Using Keyboard Entry with the Multigon Tool

The Multigon tool Numeric Keyboard Entry Dialog box can be accessed by double clicking on the tool icon. A Dialog box appears, allowing the user to numerically enter the object size and start point.

After double clicking on the tool, the Numeric Keyboard Entry Dialog box appears. It disregards the mouse actions and the object is drawn to the numeric information entered in the Dialog box. The size and start point of the multigon can be set. The Draw From button is the point on the multigon, which is placed at the Start Point coordinates.



The Multigon Tool Dialog box

Start Point

When setting the start point, first select the Start Point button, notice that the Dialog box information changes to coordinate positions.

Start Point coordinates

When the coordinate positions are entered the start point can be set to be at a selected point on the object such as Top, Left, Right, Center and Bottom of the multigon. The start point is always relative to the Absolute Point of the document. If the *Draw* From Point is selected, the multigon doubles the width typed in the Total Width Entry box.

Object Size

When the Object Size button is clicked, the Dialog changes, so that the user can enter the multigons width. Remember that the Object Size information draws from the selected *Draw* From and Start Point coordinates.

The Multigon tool Dialog has the option to enter the number of sides that make up the multigon. Remember, that each side is even in length.

If the *Draw* From Center option is selected, the width and height of the multigon, is twice the Total Width entry.

Click on the OK button once the correct information is entered. The Cancel button can be used to exit the Dialog and cancel the multigon drawing.

Arc Tool

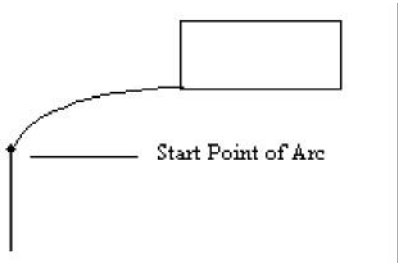
The Arc tool is designed to draw both elliptical and 90 degree arcs. **KeyCAD Pro** defaults to the elliptical Arc tool. To draw perfect circular arcs, select the tool icon and hold down SHIFT when using the tool.

An Elliptical Arc

A Circular Arc

Arc Tool combined with Snap To Tools

The Arc tool can be used with the Snap To tools, therefore allowing the user to start or end the arc from another object center, intersection, end-point, percentage point, etc. Remember an arc can start and end from *any* of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create selected objects.



The example above shows how the Arc's start point was snapped to the end-point of a line, and the end-point of the Arc was snapped to the corner point of the rectangle.

Using the Arc Tool with the Mouse

Arcs can be created in two ways while using the mouse button. The two types of arcs are elliptical arcs and perfect 90 degree arcs.

Creating an Elliptical arc

Elliptical arc

1. Select the Arc tool from the *Draw* toolbox.
2. Now, place the crosshair cursor at the desired start point, then click and drag the mouse. Note how the arc follows the crosshair cursor, allowing the user to change the length and height of the arc.
3. At the desired arc, release the mouse button.

NOTE: The start and end-point of the arc can be snapped to any of the Snap To tool options.

Creating a Circular arc

90 degree arc

1. Select the Arc tool from the *Draw* toolbox.
2. Then, place the crosshair cursor at the desired start point.
3. Now, click and hold down the mouse button while dragging the mouse. When dragging the mouse hold down SHIFT *and release the mouse button*. A straight line from the start point to the cursor appears representing the radius of the arc.
4. Drag the cursor in a clockwise or counter clockwise direction and watch the arc being created. Press the mouse button once to end the arc.

Using Keyboard Entry with the Arc Tool

The Arc tool Numeric Keyboard Entry Dialog box can be accessed by double clicking on the tool icon. A Dialog box appears, allowing the user to enter the type of the arc, and set the start and end points.

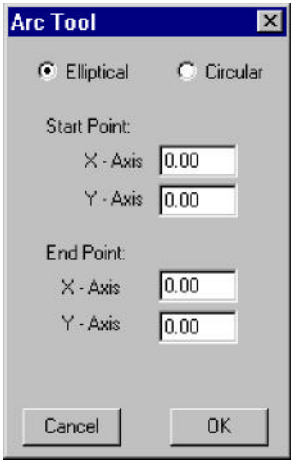
After double clicking on the Arc tool the Numeric Keyboard Entry Dialog box appears. It disregards the mouse actions and the arc is drawn to the information entered in the Dialog box. The arc radius and degree of length can be set for a circular arc. The box shown below is for an Elliptical arc.

Elliptical Arc

When selecting the Elliptical arc button, the Dialog prompts the user to type in the X, Y coordinates for the start and end-points for the arc.

Circular Arc

If the Circular arc button is selected, the End-Point Entry boxes change to Radius, Start Angle, and Arc Angle. When creating the circular arc, the start point of the arc must first be set, and then the radius and degree of length for the arc.



The Arc Tool Dialog box

Line Tool

The Line tool is designed to draw lines at any angle or length. A line can be drawn by clicking on the Line tool icon and positioning the cursor where the line is to begin. The cursor then becomes a crosshair showing the exact position where the line begins or ends. To draw perfect horizontal or vertical lines, select the Line tool icon and hold down SHIFT when using the tool.

Horizontal Lines

Vertical Lines

When creating a line, drag the cursor in the direction the line is to be created. As long as the mouse button is held down, the line can be lengthened or shortened and its angle can be changed.

Line Tool combined with Snap To Tools

The Line tool can be used with the Snap To tools, allowing the user to start or end any line from another object center, midpoint, end-point, percentage point, etc. Remember that any line can start or finish from *any* of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create lines.

For example, start the line from the end-point of another line and finish the line at the center point of a circle, by changing Snap To tools after setting the start point of the line.

Using Keyboard Entry with the Line Tool

The Line tool Numeric Keyboard Entry Dialog box can be accessed by double clicking on the Line tool icon. A Dialog box appears, allowing the user to numerically enter the length, angle and start point.

After double clicking on the Line tool, the Numeric Keyboard Entry Dialog box appears. It disregards the mouse actions and the line is drawn to the numeric information entered in the Dialog box. The length and angle of the line as well as the start point can be set.

When setting the start point, first select the Start Point button. Notice that the Dialog box information changes to coordinate positions. These coordinates allow the user to set the start and end-points of the line. The user can choose from Cartesian or Polar coordinates for drawing the line. If the Polar coordinates are selected, a start point must still be set for the line being drawn.

- Cartesian coordinates allow the user to set the start and end-point at any X, Y coordinates.

- Polar coordinates allows the entry of the length and angle of the line. The start point coordinates must still be set for the line being created.

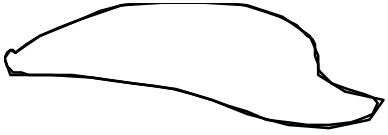
Once the correct information is entered, click on the OK button to complete the new line.

Click on the Cancel button to cancel the line drawing operation.

Spline Tool

The SHIFT key together with the Spline tool constrain each spline segment to a perfect horizontal or vertical line.

The Spline tool is designed to draw Open or Closed free-formed splines, that can contain many individual spline segments. Choose the Spline tool to create a curve that passes through specific points in the design. The points must be specified that **KeyCAD Pro** should use to draw the curve. **KeyCAD Pro** then calculates the coordinates of the curve and draws the free-formed curve.



A Closed Free-Formed Spline

A Open Free-Formed Spline

Spline Tool combined with Snap To Tools

The Spline tool can be used with the Snap To tools, therefore allowing the user to start or end curved splines from another object center, midpoint, end-point, percentage point, etc. Remember that any open or closed spline can start or end from ***any*** of the Snap To tools. Use the Tab key to change Snap To tools when dragging the mouse to create selected splines.

To use a Snap To tool with the Spline tool start point, first select the required Snap To tool, then select the Spline tool and place the start point. To finish the Spline at a specific Snap To tool, press the Tab key to change the Snap To tool while drawing the spline, and double click at the desired end-point.

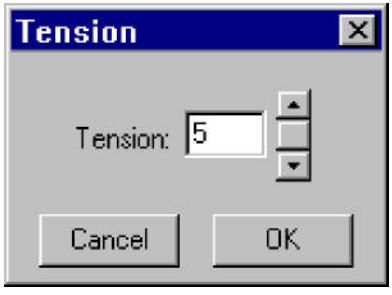
Creating a Free-Formed Spline with the Mouse

To start drawing a free-formed spline, position the cursor on the first point of the spline. Click the mouse button and move the mouse, the mouse button does not have to be held down. A line follows the mouse movement. Position the pointer on the next point that the curve should pass through and click the mouse button again. Another line begins from the previous point. Continue moving the mouse and click to specify the number of points required.

When the required number of points are specifed, double click the mouse button. **KeyCAD Pro** automatically draws the curved lines through the selected points.

After a spline is created, the tension value of the spline can be changed to loosen or tighten the curves that pass through the points. The *Smoothing* pop-out menu is found under the **Edit** menu. The *Smooth*, *Unsmooth* and the *Tension* Dialog boxes can be selected from the pop-up menu.

The Smoothing pop-up menu



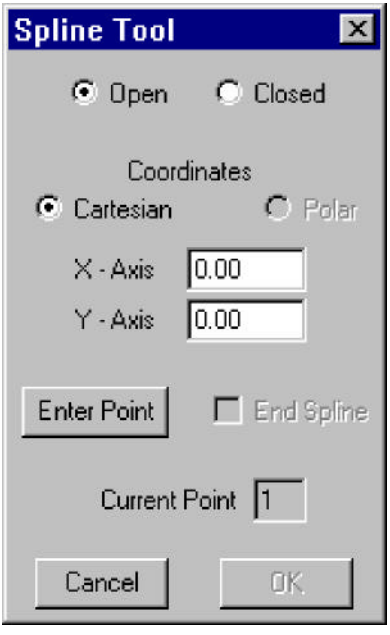
The Tension Dialog box

Using Keyboard Entry with the Spline Tool

When using the Spline tool there is the option of using numeric Keyboard Entry, instead of the mouse. To access the Dialog box, double click on the Spline tool icon.

Within the Spline tool Dialog box, choose from Open or Closed splines and set the start point of each spline and the number of points within the free-formed spline. Also choose from Cartesian or Polar coordinates. The Dialog box displays which point is being entered, as well as the total number of points making up the spline.

After the point coordinates have been entered, press the Enter Point button, to place and add the new point to the free-formed spline. Continue adding points as necessary. Once the desired number of points are entered, click on the End Spline button to automatically end the spline that is being worked on. **KeyCAD Pro** draws the curved lines through the points. Click the OK button to close the Dialog box.

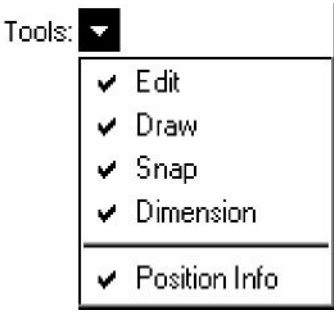


The Spline Dialog box

The *Dimension* Tools

The *Dimension* tools are very important for drafting or creating any type of blueprint. The tools found within the *Dimension* toolbox consist of Linear, Parallel, Angular, Diameter, Radius, and Leader Line tools.

The *Dimension* toolbox can be turned On or Off and can be relocated for faster access. The *Dimension* toolbox can be accessed from the **Tools** pull-down menu found on the **Menu Bar**. It can also be turned On or Off from under the **Window** menu using the **Toolboxes** pop-up menu. A check mark indicates whether the toolbox is On or Off.



The Tools pull-down menu

Combining *Dimension* Tools with *Snap To* Tools

Some of the *Dimension* tools can be used with the *Snap To* tools. The *Dimension* tools that work with the Snap To tools are Linear, Parallel, Angular, and Leader Line. The Diameter and Radius tools automatically snap to the points of the object that have been dimensioned.

The following is a complete list of the *Dimension* tools found within the toolbox.

Linear Tool

Parallel Tool

Angular Tool

Diameter Tool

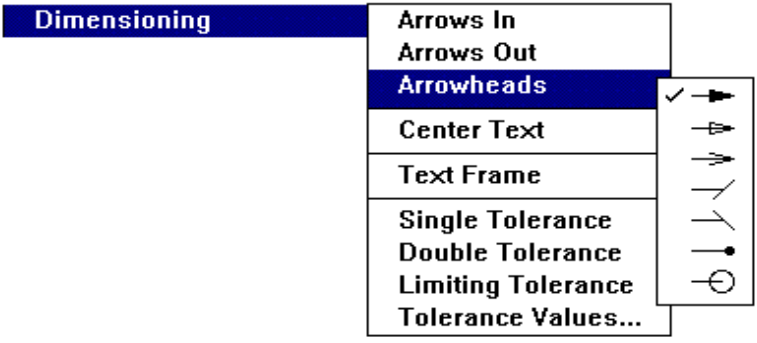
Radius Tool

Leader Line Tool

Dimension Options

The dimensions that appear when using the dimension tools, have several options available. These options are designed to give the design a more customised look. The options available appear in the *Dimensioning* pop-up menu found under the **Layout** menu. These options consist of Arrows In, Arrows Out, Arrowheads choices, Center Text, Text Frame, Single Tolerance, Double Tolerance, Limiting Tolerance, and the Tolerance Values Dialog box. The Tolerance Values Dialog is discussed in the ***Layout*** *Menu* chapter under the *Tolerance Values* sub-chapter.

The Dimensioning options pop-up menu can be accessed under the **Layout** menu.



The Dimensioning Options pop-up menu

Changing dimension colors

The color of any complete dimension including, text, arrowhead lines, and witness lines can be changed. First, select the dimension and its witness lines and then choose a new color from the color picker.

Changing font size and style

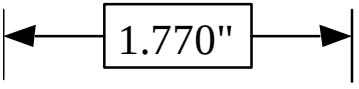
Choose the font size and style from the **Text** menu before selecting the required dimension tool.

Dimension Examples

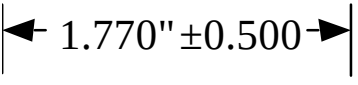
The following examples display how the different options affect the dimensions.

The Arrows In option

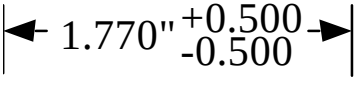
The Arrows Out option



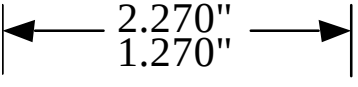
The Text Frame option



The Single Tolerance option



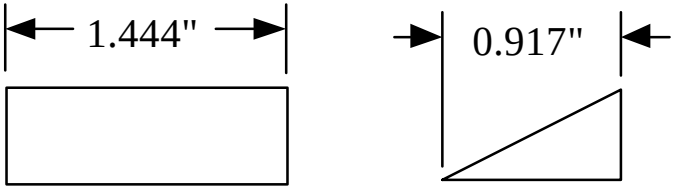
The Double Tolerance option



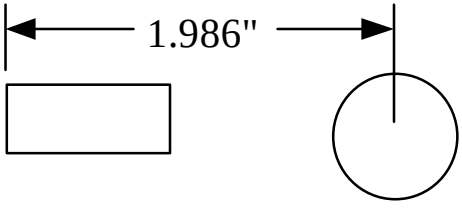
The Limiting Tolerance option

Linear Dimension Tool

The Linear Dimension tool automatically calculates the correct distance between any two points. For example, lines, squares, rectangles, sides of polygons as well as the distance between objects can be dimensioned. The Linear dimension tool is designed to display only horizontal dimensions i.e., the horizontal distance between any two points.



Linear dimension examples



Linear dimension between two objects

NOTE: The examples above show a dimension that displays the actual distance between any two points. The Linear dimension tool snaps to any of the Snap To tools.

How to use the Linear Dimension Tool

1. Select the Linear tool from the Dimension toolbox.

2. Select the Snap To tool required to snap to the first point of the object to be dimensioned.

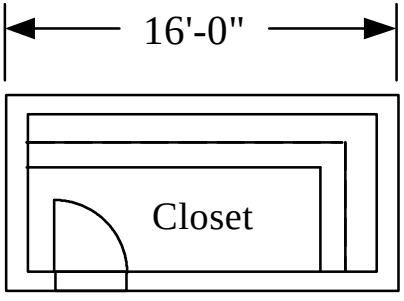
3. Place the cursor near the point to start the dimension from. Now click, hold down the mouse button and drag the cursor to the second point of the dimension, and release the mouse button near the point.

4. Now drag the dimension text out away from the object and then click once to set the dimension distance.

When using the Snap To tools, dimensioning can be done from the end-point of one object to the center point of another object. A dimension can start and end from any two combinations of the Snap To tools.

Remember that the Linear dimension tool always displays the dimension perfectly horizontal.

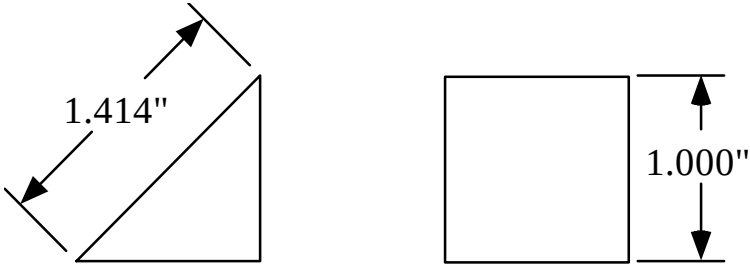
Any of the dimension options can be applied within the Dimensioning pop-up menu to any linear dimension.



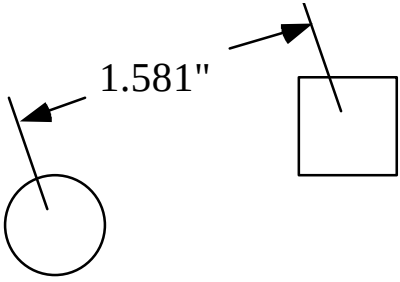
Example of Linear Dimensioning

Parallel Dimension Tool

The Parallel dimension tool automatically calculates the dimension between any two points and displays the correct distance between those points. For example, lines, squares, rectangles, sides of polygons as well as the distance between objects can be dimensioned. The Parallel tool is designed to display only parallel dimensions i.e., the parallel offset of the actual length between any two points.



Parallel dimension examples



Parallel dimension between two objects

NOTE: The examples above show the dimension that displays the actual length between any two points. The Parallel tool snaps to any of the Snap To tools, allowing the user to start and end at different points of any two objects.

How to use the Parallel Dimension Tool

1. Select the Parallel tool from the Dimension toolbox.

2. Select the required Snap To tool to snap to the first point of the object to be dimensioned.

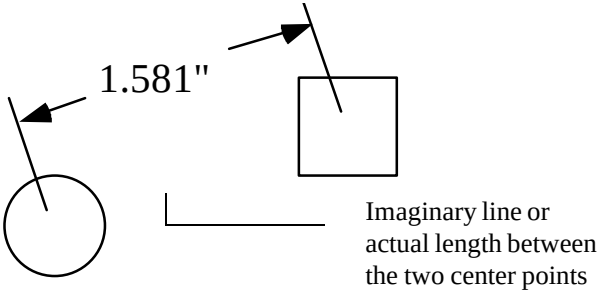
3. Place the cursor near the point to start the dimension from. Now, click, hold down the mouse button and drag the cursor to the second point, and release the mouse button near the point.

4. Now drag the dimension text out away from the object and then click once to set the parallel dimension distance.

When using the Snap To tools, dimensioning can be done from the midpoint of one object to the corner point of another object. A dimension can start and end from any two combinations of the Snap To tools.

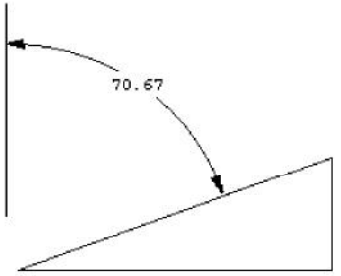
Remember that the Parallel dimension tool always displays the dimension perfectly parallel to the line or imaginary line between two objects.

Any of the dimension options can be applied within the Dimensioning pop-up menu to any parallel dimension.

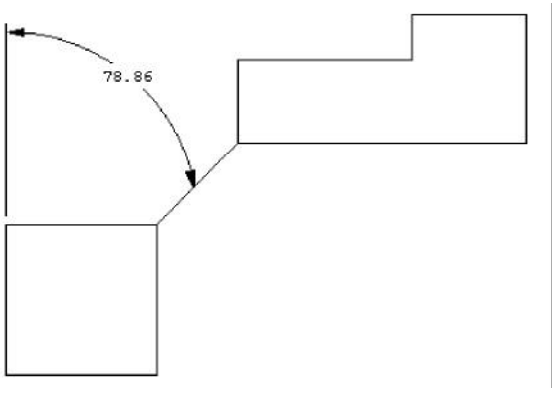


Angular Dimension Tool

The Angular dimension tool automatically calculates the angle between two lines. For example, triangles, angles of pentagons, sides of polygons as well as the angle between objects can be dimensioned. The Angular tool is designed to display only angular dimensions between any two lines.



Angular dimension examples



Angular dimension between two objects

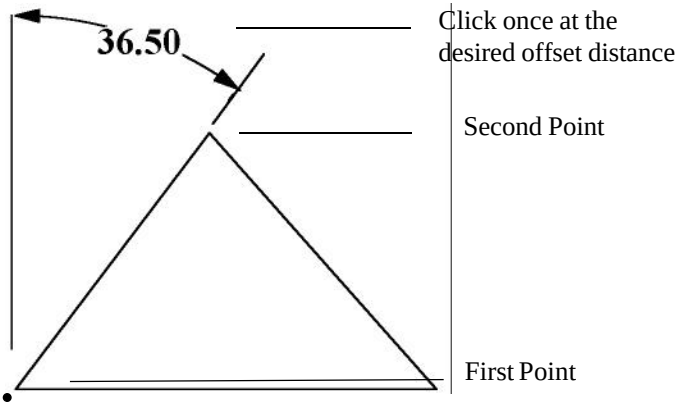
NOTE: The examples above show the dimension that displays the actual angle between any two lines. The Angle tool snaps to any of the Snap To tools, allowing the user to start and end at different points of one or two objects.

How to use the Angle Dimension Tool

1. Select the Angle tool from the Dimension toolbox.

2. Select the Snap To tool that is required to snap to the first point of the angle to be dimensioned.

3. Place the cursor near the point from which to start the dimension. Now click, hold down the mouse button and drag the cursor along the angled line, to the second point, and release the mouse button near the point

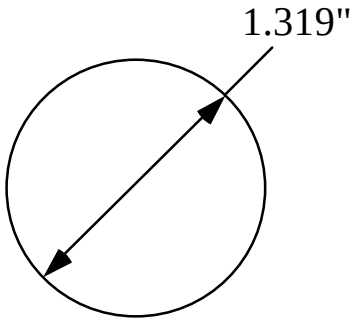


4. Now drag the dimension text out away from the object, while in a circular motion to reach any angle. Then click once to set the angled dimension distance.

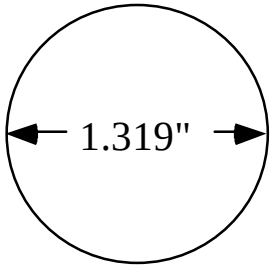
Use the Snap To tools, to accurately snap to the end-points of the angled line.

Diameter Dimension Tool

The Diameter dimension tool automatically calculates the dimension of any circle's diameter and displays the correct diameter information of the circle. This tool does not require the use of a Snap To tool. The Diameter tool automatically snaps to the circle boundary points.



Diameter dimension of a circle

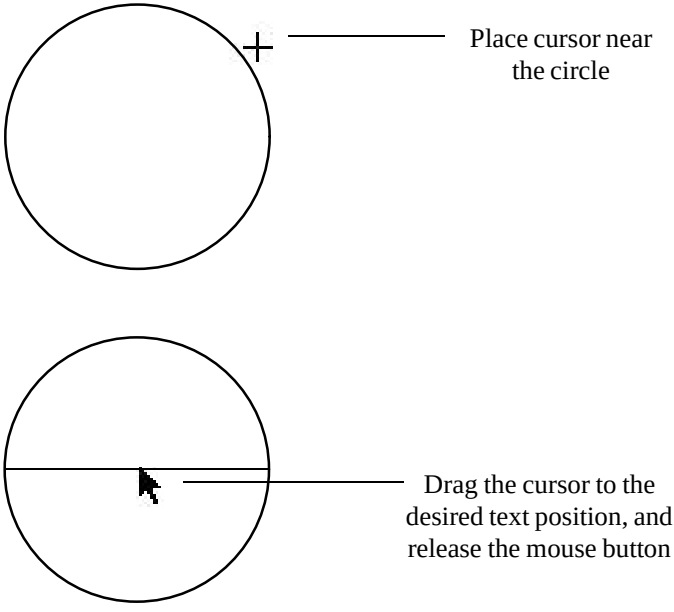


NOTE: The examples above show the Diameter Dimension of a circle. The Diameter tool automatically snaps the circle's boundary line, allowing the user to quickly display the diameter of any circle.

How to use the Diameter Dimension Tool

1. Select the Diameter tool from the Dimension toolbox.

2. Place the cursor near the circle's boundary line. Now click, hold down the mouse button and drag the cursor in a circular motion around the edge of the circle, and release the mouse button near the point to be dimensioned.

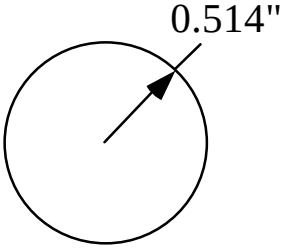


3. Now drag the dimension text to any position on the dimension line and release the mouse button. The Dimension text appears at the point where the mouse button was released.

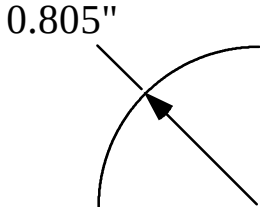
Any of the dimension options can be applied within the Dimensioning pop-up menu to any diametrical dimension.

Radius Dimension Tool

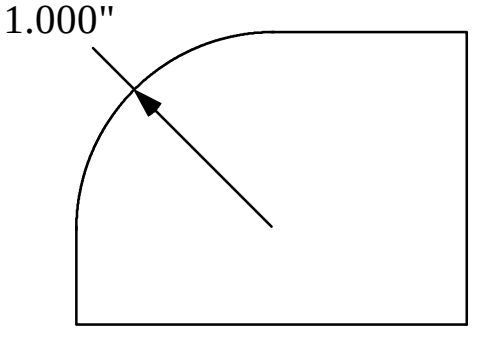
The Radius dimension tool automatically calculates the dimension of any arc radius, circle, or fillet and displays the correct radius information on the screen. The Radius tool is used only for dimensioning the radius of a 90 degree arc, radius of a circle, or the fillet of an object. This tool does not require the use of a Snap To tool. The Radius tool automatically snaps to the radius start point by clicking the mouse near the radius that is being dimensioned.



Radius dimension of a Circle



Radius of an Arc



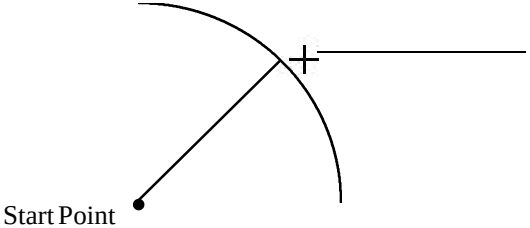
Radius of a Fillet

NOTE: In the examples above the radius dimension display three different types of radii. The Radius tool automatically snaps the radius start point allowing the user to quickly display the radius of any circle, fillet, or 90 degree arc.

How to use the Radius Dimension Tool

1. Select the Radius tool from the Dimension toolbox.

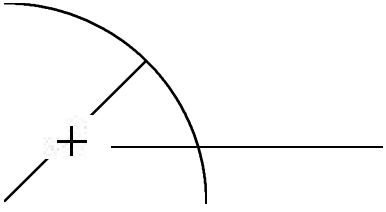
2. Place the cursor near the radius boundary line. Now click, hold down the mouse button and drag the cursor in a circular motion around the edge of the arc, circle, or fillet and release the mouse button near the point to lock the radius dimension to. The Radius tool automatically snaps to the start point.



Place cursor near the arc and hold down the mouse button

Start Point of Radius

3. Now drag the dimension text to any position on the dimension line and release the mouse button. The *Dimension* text appears at the point where the mouse button is released.

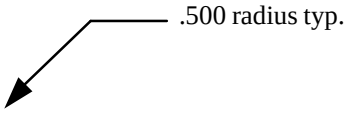


Drag the cursor to the desired text position, and release the mouse button

Any of the dimension options can be applied within the Dimensioning pop-up menu to any radius dimension. The font size, style or color can also be changed.

Leader Line Tool

The Leader Line tool is used for special quotes or parts of the design that need a line with an arrow at the beginning of the line.



Leader Line example

The Leader Line tool can be combined with the Snap To tools, so that the user can snap to specific points of other objects.

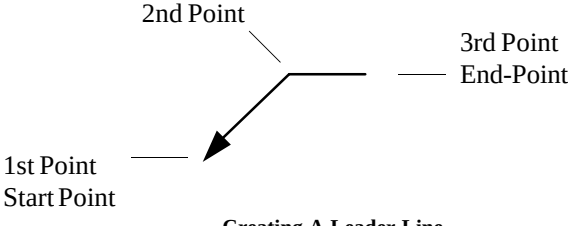
How to use the Leader Line tool

1. Select the tool from the Dimension toolbox.

2. Now place the tool at the point from which to start.

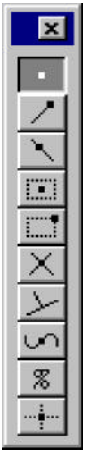
3. Then click, hold down the mouse button, and drag the cursor to a desired length. Now release the mouse button and drag the mouse to the end-point required, and click the mouse button once.

The Leader Line tool has three points to be placed in order to create the leader line.



Creating A Leader Line

The *Snap To* Tools



Almost all technical and engineered drawings require the user to draw lines, circles and rectangles in a very precise location, in specific relationships to each other. For example, to start a line at the intersection of two different objects, or draw a line perpendicular to another line. **KeyCAD Pro** provides the *Snap To* tools to assist in locating the precise points required in creating a very detailed and accurate design.

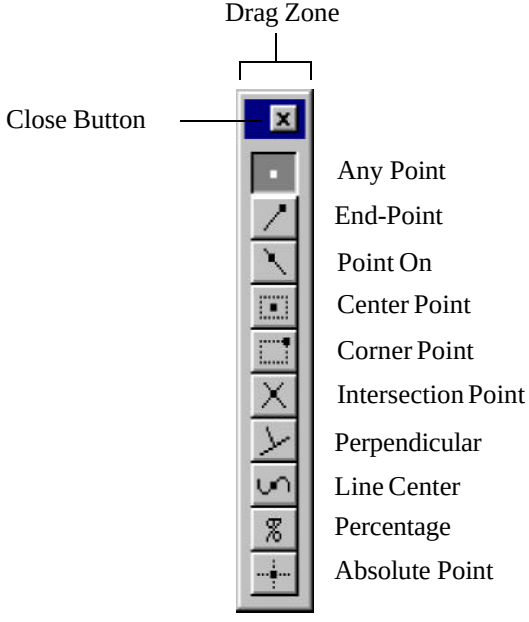
The *Snap To* toolbox has the option of being turned On or Off and can be relocated for added convenience. The *Snap To* toolbox does not appear on the screen when the program is first started. All toolboxes can be accessed under the *Toolbox* subheading found under the **Window** menu at the top of the screen. A check mark beside the toolbox name indicates that it is active.

The *Snap To* tools have been designed to work in combination with the Edit, Draw, and Dimension tools found within the program. When using the *Snap To* tools with the Draw tools a line can start from the end-point of another line and finish at the center of a circle. Using the *Snap To* tools with other tools ensures that the line or object created starts and ends in the precise location desired.

The sensitivity that determines how close the mouse cusor must be to an object for the program to snap to a specific point can also be set. The sensitivity setting is found in the *Preference* Dialog under the **Edit** menu.

When using the *Snap To* tools, the tools selected can be changed while dragging the mouse, by pressing the Tab key. This is so that drawing or editing can start with one *Snap To* tool and end with another.

The following is a picture of the *Snap To* tools as they appear in the program. The names have been listed for the appropriate tool icon. The name indicates the *Snap To* command that is performed when using that particular tool. These tools should be used in the precision alignment of objects as well as with the start and/or end-points of objects.



***Snap To* Toolbox**

NOTE: Remember, by combining these tools with the Draw, Dimensioning, and Edit tools there is more precise control overall of the tools functions.

Any Point

KeyCAD Pro defaults to the Any Point *Snap To* tool, when the program is first started. This allows the user to choose any point within the document to set a point. **KeyCAD Pro** supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the selected tool's appropriate function, allowing the user to see where the end-point is to be placed prior to releasing the mouse button. No ghost lines are visible with the Any Point tool.

NOTE: If the *Snap To* Grid option is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off, the user can locate a point at any coordinate within the current document.

Changing *Snap To* Tools

When using the *Any Point Snap To* tool, the cursor always snaps to the point on the screen, at the time the mouse button is clicked.

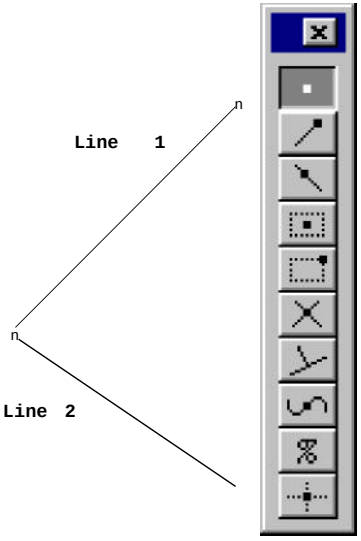
Setting Tool to Lock or Reset

The Any Point tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background and single clicking results in a gray background in the Reset mode.

End-Point

Use the End-Point *Snap To* tool to connect the object being created to the End-Point of another object. The End-Point *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest end-point, allowing the user to see where the end-point is to be placed before releasing the mouse button.

When using the End-Point *Snap To* command, first select the line to snap to (line 1). With the handles at either end of the object, select the line tool from the tool bar. Begin to draw your line (line 2) and you will notice that it automatically gravitates to either one end or the other. If the line does not snap to the ends raise the Snap Tool sensitivity in the Edit, Preferences dialog box. The Snap to line command will work with any selected object when the handle boxes are present.



NOTE: If the *Snap To* Grid option is turned Off the user can then snap to the closest end-point of the nearest object. If the *Snap To* Grid option is **On**, the user can **only** snap to points on the current grid setting.

Changing End-Point *Snap To* Tool

The *Snap To* tool can be changed at any time while pressing the Tab key. This tool allows the user to start with the End-Point tool, and end the object with a different *Snap To* tool, such as the Center tool.

Setting End-Point Tool to Lock or Reset

The End-Point tool can be set to a locked position,to prevent resetting after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

Point On

Use the Point On *Snap To* tool to connect the objects being drawn to a point on another object. The point the tool snaps to, is the closest calculated point at the current position when the mouse button is released. The Point On *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest point within the object, allowing the user to see where the point is to be placed before releasing the mouse button.

Anytime the Point On *Snap To* tool is selected, the cursor automatically snaps to the closest point within the nearest object. The tool snaps to any point that is within the sensitivity range of the point. Sensitivity can be set in the Preference Dialog, found under the **Edit** menu.

NOTE: If the *Snap To* Grid option is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off the user can snap to the closest vertex point within the object nearest to the mouse cursor.

Changing *Snap To* Tools

The *Snap To* tool can be changed at any time while pressing the Tab key. This allows the user to start with the Point On *Snap To* tool, and end with a different *Snap To* tool, such as the End-Point *Snap To* tool.

Setting Tool to Lock or Reset

Set the Point On tool to a locked position, to prevent resetting after the first point is set. Double clicking Locks the tool to a dark background while a single click sets the tool in a gray background in the Reset mode.

Center Point

Use the Center Point *Snap To* tool to connect the objects being drawn to the center point of another object. The tool snaps to the center point of the closest object in relation to its current position, when the mouse button is released. The Center Point *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest center point of an object, allowing the user to see where the point is to be placed before releasing the mouse button.

Anytime the Center Point *Snap To* tool is selected the cursor automatically snaps to the closest center point within the nearest object. The tool snaps to any center point of an object, if the cursor is within the sensitivity range of the object. Sensitivity can be set in the *Preference* Dialog, found under the **Edit** menu.

NOTE: If the *Snap To* Grid option is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off the user can then snap to the closest point.

Changing *Snap To* Tools

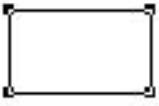
The Center Point *Snap To* tool can be changed at any time while pressing the Tab key. This allows the user to start with the Center Point *Snap To* tool, and end with a different *Snap To* tool, such as the Corner Point *Snap To* tool.

Setting Tool to Lock or Reset

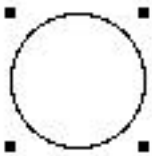
The Center Point tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

Corner Point

Use the Corner Point *Snap To* tool to connect the objects being drawn to the corner point on another object. Lines have two end-points that **KeyCAD Pro** identifies as corner points. The corners of other objects such as, rectangles, arcs, circles, and ellipses, are at the corners of the rectangular boundary lines.



Corner point of a rectangle



Corner point of a circle

The tool snaps to the corner point of the object which is closest to the current position at the moment the mouse button is released. The Corner Point *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest corner point of an object, allowing the user to see where the point will be placed before releasing the mouse button.

When the Corner Point *Snap To* tool is selected the cursor automatically snaps to the closest corner point within the nearest object. The tool snaps to any corner point of an object, if the cursor is within the sensitivity range of the object. Sensitivity can be set in the Preference Dialog found under the **Edit** menu.

NOTE: If the *Snap To* Grid option is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off the user can then snap to the corner point within the object closest to the cursor.

Changing The Corner Point *Snap To* Tool

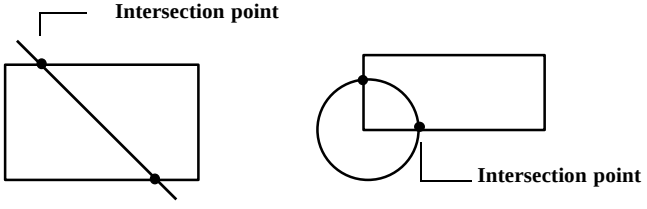
The Corner Point *Snap To* tool can be changed at any time by pressing the Tab key. This allows the user to start with the Corner Point *Snap To* tool, and end with a different *Snap To* tool, such as the Perpendicular Point *Snap To* tool.

Setting Tool to Lock or Reset

The Center Point tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

Intersection

The Intersection *Snap To* tool should be used to connect the start or end-point of an object where two other objects intersect. An intersection is defined as any point where the lines of two objects meet or cross.



When using the Intersection tool, the user can snap to any intersection point of two objects.

The Intersection *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest intersection of any two objects, allowing the user to see which intersection the point is to be snapped to before releasing the mouse button.

The Intersection tool snaps to the intersection point of any two objects, if the cursor is within the sensitivity range of the intersection. Sensitivity can be set in the *Preference* Dialog found under the **Edit** menu.

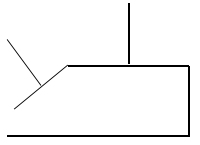
NOTE: If the *Snap To* Grid option is On, the user can only snap to the closest points on the current grid setting. If the *Snap To* Grid option is turned Off the user can then snap to any intersection point of two objects.

Perpendicular

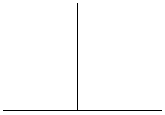
This tool should be used when a line is to be automatically drawn perpendicular to other objects within a design.

The following examples show lines drawn perpendicular to other types of objects.

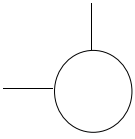
Rectangle



Open or Closed Polygons



Line



Circle

To draw a line perpendicular to an object.

1. Select the Line tool from the Draw toolbox.

2. Then, select the Perpendicular *Snap To* tool.

3. Now, place the mouse cursor near the object that the line is to be perpendicular to, click and drag the mouse to the desired line length and release the mouse button.

The user can also designate that a line be perpendicular to an object at it's end point. As the line is being drawn to the object it is to be perpendicular to, before releasing the mouse button make sure the Perpendicular tool is selected. Now when the mouse button is released near the object, the end-point of the line becomes perpendicular to the object.

NOTE: If the *Snap To* Grid option is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off the user can then have lines snap perpendicular to other objects at any position.

Changing the Perpendicular *Snap To* Tool

The Perpendicular *Snap To* tool can be set at any time after drawing a line has started, by pressing the Tab key.

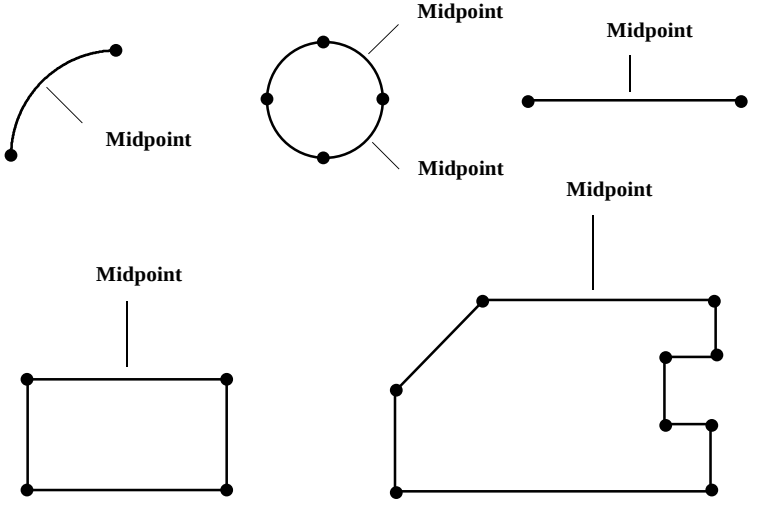
Setting Tool to Lock or Reset

The Perpendicular tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

Line Center

The Line Center tool automatically sets the start or end-point of a line at the exact midpoint of a line or free-formed spline.

The following examples further explain how one object may have more than one mid-point. This means that the Line Center tool snaps to any midpoint within an object.



To draw a line at another object midpoint, first select the Line tool from the Draw toolbox and then select the Line Center tool. Now place the mouse cursor near the object to snap to. Click and drag the mouse away from the object, and the line start point automatically snaps to the line center point of the object. Remember that the line center is located between any two points with a line between them. The line can be straight or curved.

The user can also designate the end-point of the line being drawn, to finish at the line center point of another object. As the line is being drawn to set its end-point, be sure that the Line Center *Snap To* tool is selected, before releasing the mouse button. Now, when the mouse cursor is near the object to set the end-point at, release the mouse button. **KeyCAD Pro** automatically snaps the end-point of the line to the line center point.

The Line Center *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest line center point. Therefore allowing the user to see where the line center will be placed before releasing the mouse button.

NOTE: If the *Snap To* Grid option is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off the user can then have lines snap to line center points of other objects.

Changing The Line Center *Snap To* Tool

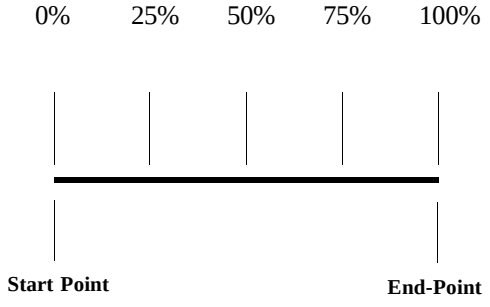
The Line Center *Snap To* tool can be changed at any time after drawing a line has started, by pressing the Tab key.

Setting Tool to Lock or Reset

The Line Center tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

Percentage

This tool allows the user to locate a point on a line, arc, or polygon by specifying a percentage value of the line. The percentage indicates a distance along the length of the line. The start point is considered 0% and the end-point is 100%. A point located at 40% of the line's length can be specified and **KeyCAD Pro** automatically locates that point.



The percentage snap value can be set in the Preference Dialog, which is found under the **Edit** menu.

To draw a line starting at a percentage point of another line, first select the line tool from the Draw toolbox. Then select the Percentage *Snap To* tool, place the mouse cursor near the line to snap the start point to, and click and drag the mouse away from the line. **KeyCAD Pro** automatically snaps the start point, to the percentage snap point of the line selected.

The previous paragraph explained how to snap the start point of a line. The same principle applies to the end-point of the line being created. As the line is being dragged, change the *Snap To* tool to the Percentage tool. Then position the end-point of the line, near the line to snap to and release the mouse button.

The Percentage *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the closest percentage point. This allows the user to see where the percentage point is placed before releasing the mouse button.

NOTE: If the *Snap To* Grid is On, the user can only snap to points on the current grid setting. If the *Snap To* Grid option is turned Off, the user can then have lines snap to any percentage point that have been set for the percentage tool.

Changing The Percentage *Snap To* Tool

The Percentage *Snap To* tool can be changed at any time after starting to draw a line, by pressing the Tab key.

Setting Tool to Lock or Reset

The Percentage tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

Sensitivity Settings

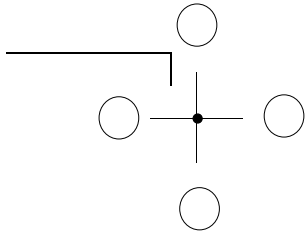
The Percentage tool snaps to any percentage point of any line, if the cursor is within the sensitivity range of the percentage point. Sensitivity *Snap To* setting can be found in the *Preference* Dialog, which is found under the **Edit** menu.

Absolute Point

The Absolute Point tool always snaps objects, start or end-points of objects to the Absolute point of a design. The absolute point is the same as the point where the X, Y - Axis lines meet; it can also be referred to as the Origin point. The absolute point is always equal to X - 0 and Y - 0. This is the reference point for all absolute readouts. The absolute point is located at the lower left corner of a design when the program is started.

The absolute point is a great point of rotation. It is used for rotating and duplicating objects simultaneously, thus providing a point of rotation that can be located at any point within a design. This is because the Axis Lines can be moved to any location.

Rotate objects around the Absolute Point (Origin Point)



The Absolute Point *Snap To* tool supports an interactive snap to function. This means the ghost line of the object being drawn or edited snaps to the current absolute point within the document, allowing the user to see where the end-point of the drawn line is to be placed before release of the mouse button.

To draw a line from the absolute point, first select the Line tool from the Draw toolbox. Then select the Absolute *Snap To* tool and place the mouse cursor near the absolute point on the document. By clicking and dragging the mouse, the start point of the line automatically snaps to the absolute point.

The end-point of the line being drawn can be designated, to finish at the absolute point within the document. As the line is being drawn to set its end-point, be sure that the Absolute Point *Snap To* tool is selected, before releasing the mouse button. Now, when the mouse cursor is near the absolute point in the document, release the mouse button. **KeyCAD Pro** automatically snaps the end-point of the line to the absolute point.

Changing The Absolute *Snap To* Tool

The Absolute Point *Snap To* tool can be changed at any time after starting to draw a line or moving an object, by pressing the Tab key.

Setting the Absolute Point Tool to Lock or Reset

The Absolute Point tool can be set to a locked position, so that it does not reset after the first point is set. Double clicking locks the tool to a dark background while a single click on the tool results in a gray background in the Reset mode.

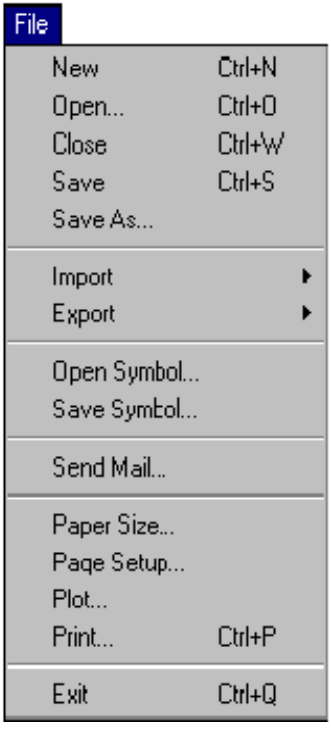
Sensitivity Settings

The Absolute Point tool snaps to any absolute point coordinate that is currently set within the document, if the cursor is within the sensitivity range of the absolute point. Sensitivity of *Snap To* settings can be found in the *Preference* Dialog under the **Edit** menu.

The File Menu

The **File** menu contains the commands which control the opening and closing of files and symbols. Printing of drawings is also controlled from within the **File** menu.

Some of the **File** menu commands can be accessed with keyboard equivalents, when combining the CTRL key (COMMAND key on the Macintosh) with the appropriate key, as shown below. The following pages in this chapter explain the proper uses for these menu commands.



The File Menu

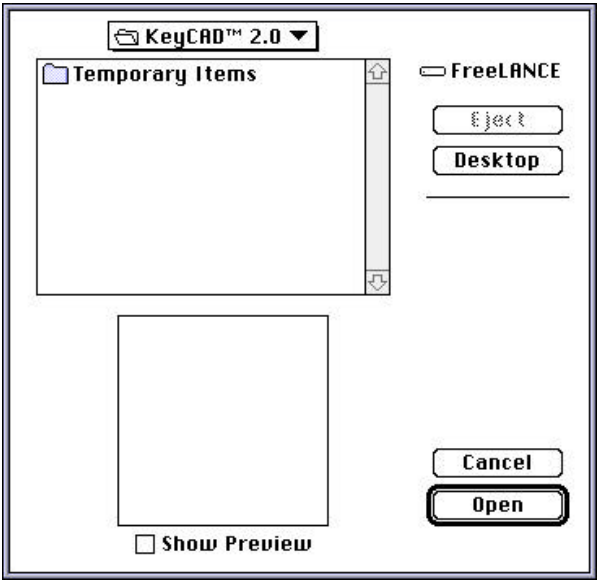
New

The New command is used to open a new document as shown below. The new document is untitled and has only one layer. The title bar automatically displays *Untitled*, which means that the document has no name. To name the document, save the document.

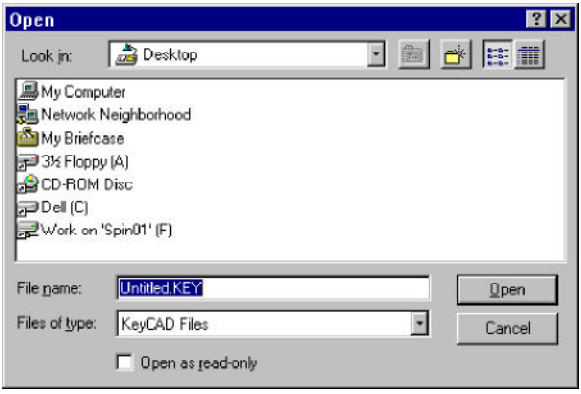
KeyCAD Pro supports multiple documents, meaning that the New command can be used even if another document is already open. The new documents appear on top of each other, without replacing the previous document. In order to change the names of the new documents, save them.

Open...

This command is used to open previously created documents. To access this command select the *Open* command from the **File** menu, or press CTRL+ O (COMMAND+O for Macintosh). After choosing the Open command, a Dialog appears allowing the user to select the name of the file to be opened.



The Open Dialog box (Macintosh)



The Open Dialog box (Windows)

On the appearance of the Dialog box select the required document from the list in the left hand box. The list contains the names of each **KeyCAD Pro** file. The directories on the current drive are shown in the right hand box. If there are more filenames than can fit in the window, use the scroll bars to find the file or directory name required.

Click the Cancel button to return to the screen before the Open command was selected. Refer to the Windows User Manual under *Opening Documents*.

Close

The Close command is used to close the currently active document. The *Close* command is found under the **File** menu or press CTRL+W (COMMAND+W for Macintosh) to activate the *Close* command.

Other open documents are not affected by the Close command. To close a window that is not active, click on the window first to activate it and then choose the Close command.

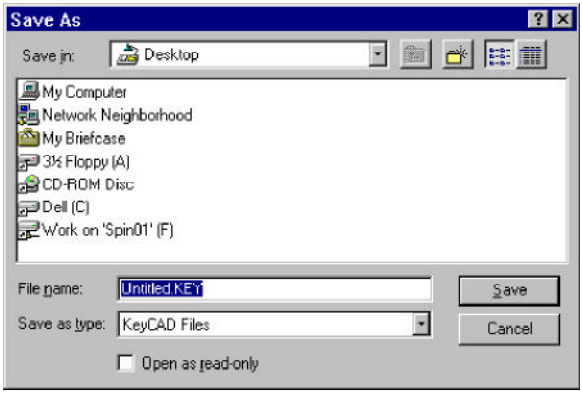
If changes have been made to a currently active document since the last time it was saved, a message Dialog appears asking if the user wants to save the changes before closing the window or document.

Save

This command saves a copy of the current document being worked on. The *Save* command is found under the **File** menu, or it can be accessed by pressing CTRL+S (COMMAND+S for Macintosh).

When choosing the Save command, only the document that is active is saved. The program always saves the file to the currently selected directory or drive. If there are several windows open, activate the window that contains the document to be saved, before selecting the *Save* command.

In the case of an untitled document, a Dialog box appears requesting a file name.



The Save Dialog box (Windows)

Enter a name for the drawing and either click the OK button or press ENTER. If the current disk does not have sufficient free space, a message appears asking to save the document to another disk.

If the document already has a name, the Dialog box is not displayed. The Save command automatically updates the current document by replacing the version on the disk with the version currently active on the screen.

Save As...

To update the most recent changes of the current document, choose the Save command. Use *Save As* to save a copy of the current document to a second file without affecting or replacing the version currently held on disk.

The Save As command saves a file or document under a different name, directory, or to a different disk drive. Save As can be used to save a new document for the first time. The Save As command is found under the **File** menu.

The Save As Dialog box (Macintosh)

The *Save As* command can be used in different ways. For example, to make a backup copy of a current work file, or keep both the original version of the file along with a more recent version. In making different versions, save the original version under one name, and then use *Save As* again to save the new document under a different name.

To save a file in a different directory, double click the directory name in the Directories box to open the new directory. Change which drive the new file is saved to, by clicking on the drive name shown in the lower right-hand corner of the *Save As* Dialog box.

Import

The Import option is a pop-up menu that is used to access the different file formats that are supported within **KeyCAD Pro**. The formats that can be imported are DXF (AutoCAD), DRW, CorelDraw, PICT, and XYZ Coordinates.

Windows

Macintosh

PICT (Macintosh only)

Choose PICT to import files that have been saved in the PICT format. This is a standard graphics file format used on the Macintosh. These files are primarily used in painting programs.

DRW (Windows only)

Choose DRW to display or open files that have been saved in the DRW format. DRW is the Micrographix Designer file format.

DXF

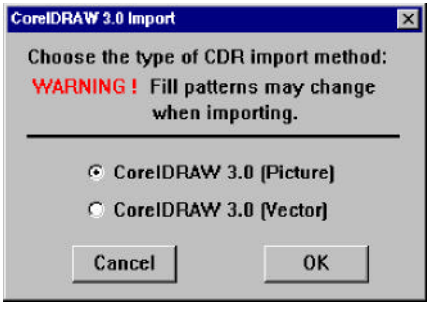
Choose DXF to display or open files that have been saved in the DXF format. The DXF format is the most popular among **AutoCAD** users, this format is used when transferring **AutoCAD** files from one computer program to another.

NB: Versions of AutoCAD DXF subsequent to Version 10 are not supported.

XYZ Coordinates

This format is used to import XYZ coordinate points. When using this format, the program only places data points in the proper coordinate position. The program does not draw lines or curves between these points.

CorelDraw... (Windows only)



The Corel Draw Dialog Box

Choose CorelDraw... to display or open files that have been saved in the CDR file format. CDR is the CorelDRAW file format. This version of **KeyCAD Pro** supports file formats from CorelDRAW version 3.0 or lower.

As the warning in the dialog box indicates, some fill patterns used in CorelDRAW are not able to be converted.

There are two ways to import a CDR file, Picture and Vector, and they are described below.

Picture format

The file will be displayed on the screen as one scaleable object. You cannot edit the individual shapes and objects that were in the file. You can resize the entire picture just as you would any other object. This is the default CDR import format.

Vector format

To use this method, click on the radio button to the left of CorelDRAW (Vector). If this method is used, the individual objects that make up the drawing can be edited and resized.

After selection of the appropriate method, and click on the OK button, you will then be presented with the standard open dialog box to locate the file.

BMP (Windows only)

Choose BMP to display or open files that have been saved in the BMP format. This is a standard graphics file format used in Windows. BMP files are bitmap or raster files. These files are primarily used in painting programs.

EPS (Preview) (Windows only)

Choose EPS to display or open files that have been saved in the EPS format. The EPS format stands for Encapsulated PostScript, which translates a file to the PostScript page-command language. This format is very popular among page layout and illustration programs.

TIFF (TIF) (Windows only)

Choose TIFF to display or open files that have been saved in the TIFF format. The TIFF (Tagged Image File Format) file format is used to save scanned images.

Export

The Export option is a pop-up menu that is used to access the different file formats that are supported within **KeyCAD Pro**. These formats can be used to export to any program that supports the DXF, HPGL, EPS, and XYZ Coordinates. When one of the formats within the **Export** menu is selected, a Dialog box appears prompting the user to name the file.

Windows

Macintosh

PICT (Macintosh only)

Choose PICT to export or save files in the PICT format. This is a standard graphics file format used on the Macintosh. These files are primarily used in painting programs.

DXF

Choose DXF to export or save files in the DXF format. The DXF format is the most popular among AutoCAD users, this format is used when transferring AutoCAD files from one computer program to another.

HPGL

This format is exported to create an HPGL file that can be used with plotter servers. A file can be plotted without having to have the **KeyCAD Pro** program running when plotting.

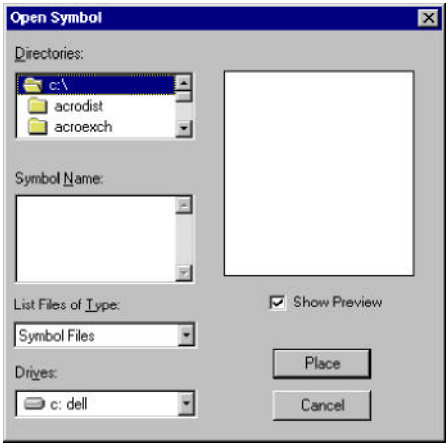
EPS

The EPS format stands for *Encapsulated PostScript*, which translates a file to the PostScript page-command language. This format is very popular among page layout and illustration programs.

XYZ Coordinates

This format is used to export XYZ coordinate points. When using this format, the program only places data points in the proper coordinate position. The program does not draw lines or curves between these points.

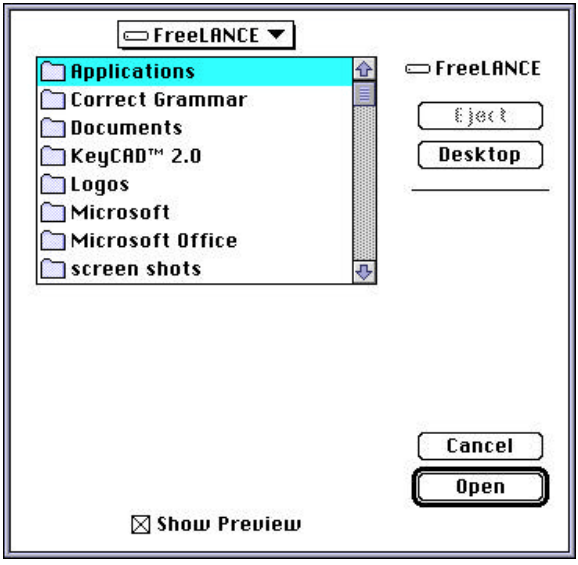
Open Symbol...



The Open Symbol Dialog Box (Windows)

Symbols are treated like individual files that can be placed within the current design. Symbols are predrawn images that can be used to save time when creating a design.

Remember that a Symbol file format is different from the regular file format that the designs are saved in when the user selects the *Save* or *Save As* command. Symbol files have the extension **.SYM**.



The Open Symbol Dialog Box (Macintosh)

Opening a symbol (Macintosh)

1. Pull down the File menu and select the Open Symbol command. A Dialog box, like the one above, will appear.
2. Select the desired disk drive.
3. Select the desired directory.
4. Select the desired Symbol (or KeyCAD) file by using the scroll box.
6. Click on the Open button. The symbol will be centered in the current view window.

Opening a symbol (Windows)

1. Pull down the File menu and select the Open Symbol command. A Dialog box will appear.
2. Select the desired disk drive by using the scroll box under the Drives heading.
3. Select the desired directory by using the scroll box under the Directories heading.
4. Select the desired file fomat, .SYM or .KEY from the List Files of Type heading. If the .KEY file format is used, the .KEY file will be read in and placed as a symbol.
5. Select the desired Symbol (or KeyCAD) file by using the scroll box under the Symbol Name heading.
6. Click on the Place button. The symbol will be centered in the current view window.

Show Preview

The Show Preview check box can be either turned On or Off. An X in the box indicates that Show Preview is turned On. If this option is on, a scaled version of the symbol is shown in the preview box on the left of the screen. If this option is turned Off, then no image appears in the preview box.

Save Symbol...

When an object has been created and it is to be saved as a Symbol, first select the object to be saved, before choosing the Save Symbol command. After selecting the *Save Symbol* command from the **File** menu, a Dialog box appears prompting for the name of the symbol. After the symbol has been saved, the symbol can be opened again from the *Open Symbol* command. All symbols are given the extension .SYM in order that they can be easily identified.

Send Mail... (Windows only)

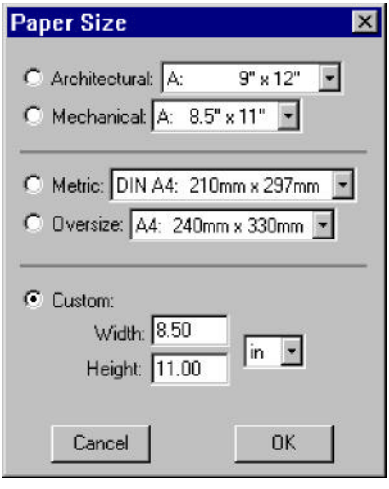
See Microsoft Windows 95 user manual.

Paper Size...

When the *Paper Size* command is selected from the **File** menu, a Dialog box appears asking the user to choose the paper size for the new document. There are several industry standards to choose from, or the option to create a custom paper size. The program also supports most of the metric standard sizes of paper. The pull-down menus have the letter or numeric name along with the paper's actual dimension size. When the program is first started, it defaults to the Mechanical "A" size (8.5"x11") piece of paper. The Paper Size can be changed at anytime while creating the design. Click the OK button after selecting the appropriate paper size.

The program represents the paper with a white background. Scroll to the edge of the paper and notice where the paper ends and the background color of the monitor begins.

Below is an example of the Paper Size Dialog box, and the following pages display the pull-down menu options.



The Paper Size Dialog Box

The Paper Size Chart

Architectural:	A	9" x 12"
	B	12" x 18"
	C	18" x 24"
	D	24" x 36"
	E	36" x 48"
	F	28" x 40"
	30/42	30" x 42"

Mechanical:	A	8.5" x 11"
	B	11" x 17"
	C	17" x 22"
	D	22" x 34"
	E	34" x 44"

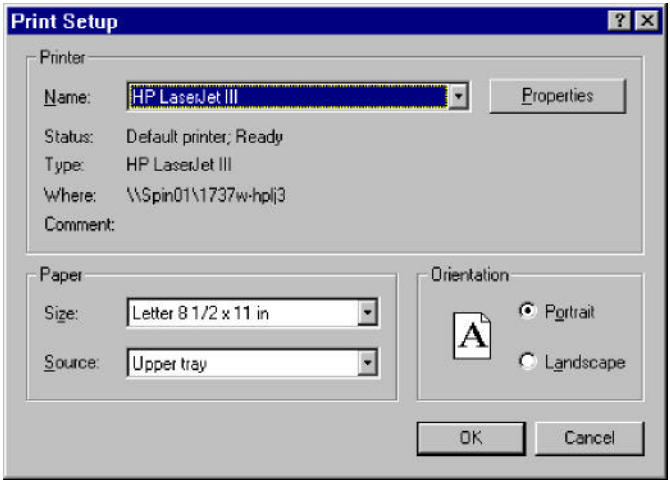
Metric:	DIN A4	210mm x 297mm
	DIN A3	297mm x 420mm
	DIN A2	420mm x 594mm
	DIN A1	594mm x 841mm
	DIN A0	841mm x 1189mm
	DIN B1	707mm x 1000mm

Metric Oversize:	A4	240mm x 330mm
	A3	330mm x 450mm
	A2	450mm x 625mm
	A1	625mm x 880mm
	AO	880mm x 1230mm

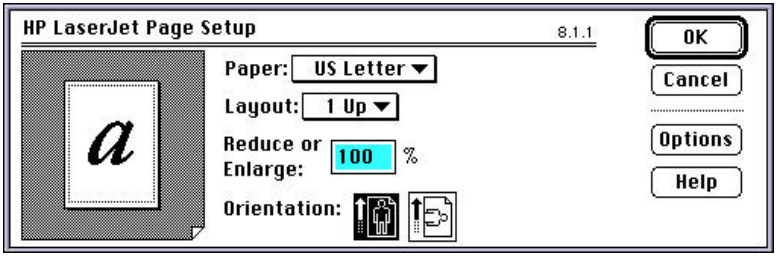
Custom Size... Can choose from **in.**, **mm** or **cm** and have custom horizontal width and vertical height paper dimensions.

Page Setup...

The *Page Setup* command specifies the paper size, orientation, and pagination of the current file when printing the active file. The *Page Setup* command is found under the **File** menu. After choosing the *Page Setup* command, the *Print Setup Dialog* box appears on screen.



Page Setup Command (Windows)



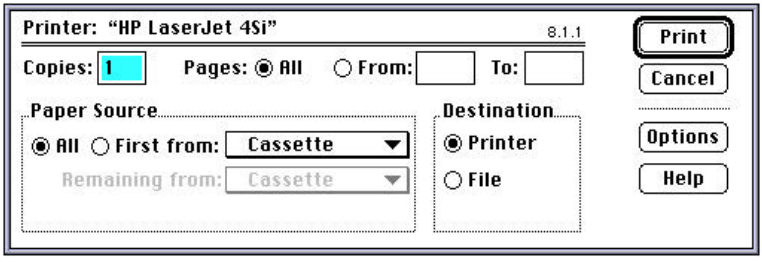
Page Setup Command (Macintosh)

Print/Plot... (Macintosh Only)

The *Print/Plot* command can be selected from the **File** menu or by pressing Command+P. The program prints only the active view on the screen. Any other documents or files in the background that are opened are not printed.

NOTE: The information which is displayed in the *Print/Plot Dialog* box depends on the system software and printer drivers that are in use.

The following example displays and discusses the Print/Plot Dialog box for a Hewlett Packard printer. The settings and choices within the Dialog box vary for different printers. Refer to the Printer or Plotter Manual for more details concerning the printing device.



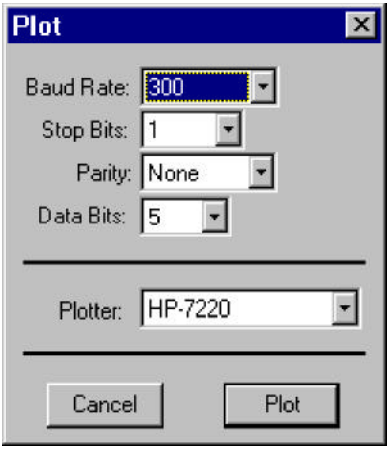
Print/Plot Dialog Box

Plot... (Windows Only)

The *Plot* command is used to plot designs to any HPGL (Hewlett Packard) compatible plotter. The *Plot Dialog* box is used to set the program's parameters to the plotters specification. For plotter specifications refer to the Plotters Manual.

There is the option to set the Baud Rate, Stop Bits, Parity, Data Bits, and a choice of several different HP plotter models.

Before clicking on the *Plot* button, be sure that the Dialog box settings are set for the plotter.



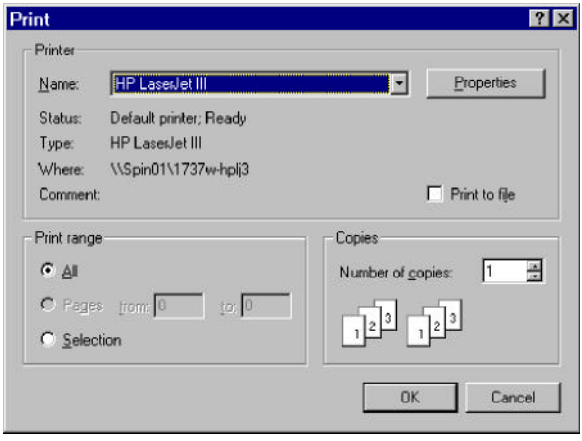
The Plot Dialog Box

Print (Windows Only)

The *Print* command can be selected from the **File** menu or by pressing CTRL+P (COMMAND+P for Macintosh). The program prints only the active view on the screen. Any other documents or files in the background that are opened are not printed.

NOTE: The information which is displayed in the *Print Dialog* box depends on the system software and printer drivers that are in use.

The following example displays and discusses the Print Dialog box for an Epson printer. The settings and choices within the Dialog box vary for different printers. Refer to the Printer or Plotter Manual for more details concerning the printing device.



Print Dialog Box

Exit

The *Exit* command is used to exit the **KeyCAD Pro** program. Choose the *Exit* command from the **File** menu or by pressing CTRL Q.

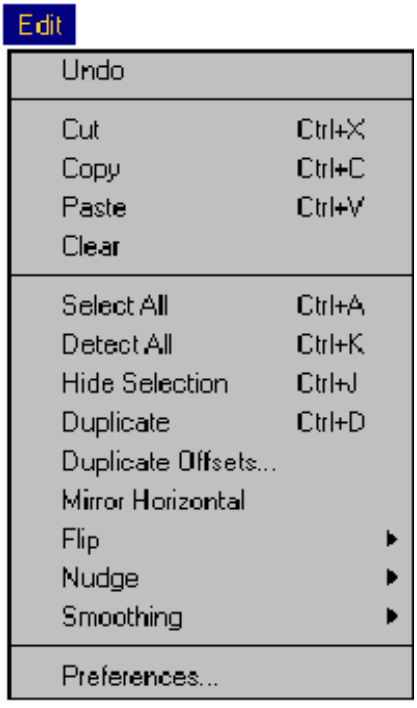
If any changes are made to the currently opened file, the warning "*Save changes before closing?*" appears. Respond appropriately. In the case of untitled files, the *Save As Dialog* box is displayed which prompts the user to enter a name for the file.

The Edit Menu

The **Edit** menu is designed to offer the commands for changing, editing or modifying any object and text within a design. Many of these commands have menu equivalents for quicker access.

The *Preferences* Dialog box is accessed under the **Edit** menu. This is where the preference of certain defaults found within the program can be changed. Some of the Menu Bar button selections are also found under the **Edit** menu.

Some of the **File** menu commands can be accessed with keyboard equivalents, when combining the CTRL key (COMMAND key on the Macintosh) with the appropriate key, as shown below.



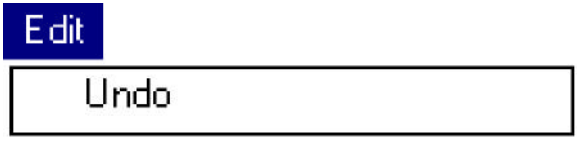
The Edit Menu

Show Clipboard

An additional command is located under the **Edit** menu in **KeyCAD Pro for Macintosh**, Show Clipboard. This command will display the current contents of the Clipboard.

Undo

The *Undo* command is used to cancel the last action that changed the file being worked on. For example, if a door is created then resized, to revert to the original size, choose the *Undo* command from the **Edit** menu.



Undo is found under the Edit Menu

The *Undo* command permits cancellation of the last action on an object, such as duplicating, resizing, reshaping or changing position and location.

NOTE: *Undo* works only on the last action performed. Once another action changes the file, the preceding action cannot be undone.

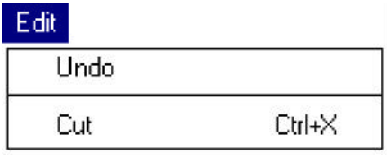
Scrolling, selecting a drawing, changing of a Dialog box, or the resizing of a window, are actions that cannot be undone and effectively do not change a file.

There are two ways to activate the *Undo* command. Select *Undo* from the *Edit* menu or press CTRL Z.

Cut

The *Cut* command removes any selected objects or text from the design and automatically places them on the Clipboard. Remember that when the *Cut* command is used, anything currently on the Clipboard is replaced with the new selection that was Cut.

The *Cut* command can be selected from the **Edit** menu or by pressing CTRL+X.



Cut is found under the Edit Menu

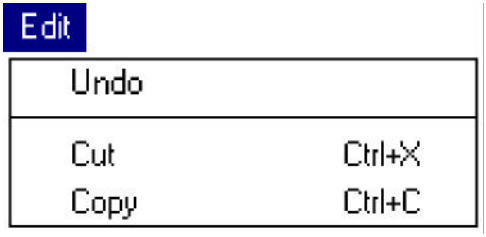
Use *Cut* to delete objects or text, or to move them to a new location on the design. To use the *Cut* command first select the object or text and choose the *Cut* command; it cuts the selected objects to the Clipboard. Move or scroll the document to a new location and choose the *Paste* command to place the Clipboard contents at the center of the new location. The *Paste* command is also found under the **Edit** menu.

Cut can be used to move objects from one **KeyCAD Pro** document to another. Cut the object from the first document, then open the second document and choose the *Paste* command. If there are multiple windows opened, the user can Cut from the active window then click on the target window to activate it, then choose the Paste command.

Copy

Choose the *Copy* command to copy selected objects or text to the Clipboard. The copied object replaces anything previously on the Clipboard. Use the *Copy* command to duplicate objects or text within the current document or different files.

The *Copy* command can be selected from the **Edit** menu, or by pressing CTRL+C.



Copy is found under the Edit Menu

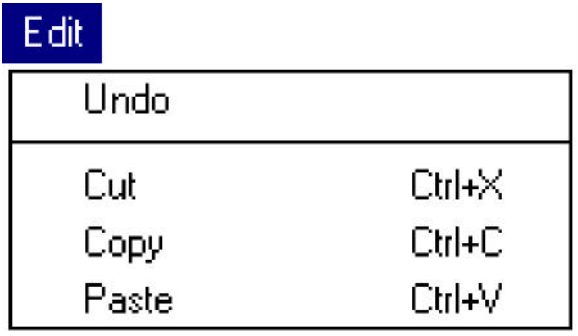
Objects or text *must be* selected prior to selecting the *Copy* command. After selecting the *Copy* command, the objects or text remain selected. Deselection is done by clicking anywhere on the drawing area.

Once a selection is copied to the Clipboard, the object or text can be pasted to another section of the document or even to the file of another application that supports the *Paste* command.

Paste

The *Paste* command places a current copy of the Clipboard contents into the active layer of the current window. Pasting an object from the Clipboard does not clear the Clipboard contents. Because the contents of the Clipboard remain on the Clipboard, the *Paste* command can be used repeatedly to make multiple copies of text or objects within the same or different files. For a description on other methods of duplicating objects, refer to *Duplicate* and *Duplicate offsets* later in this chapter.

Paste can be selected from the **Edit** menu, or by pressing CTRL+V.



Paste is found under the Edit Menu

Pasting an Object

KeyCAD Pro pastes an object by placing it in front of all other objects within the current file.

- If the current file has just been opened and the Paste command selected, the copy is placed in the center of the screen.
- If the file in the window has been scrolled, the copy is placed in the center of the screen.
- Since **KeyCAD Pro** supports multiple layers within one file, the Cut, Copy, and Paste commands can also be used when moving objects between layers.
- Be sure to activate the layer first before choosing the Copy command, then activate the layer to Paste the object into.

Transferring Pasted Objects

An option exists for the transferring of complete drawings, individual objects, and text created in other applications. The files designed in other Draw or CAD programs, can be transferred to the Clipboard, and then

pasted into **KeyCAD Pro** documents. The entire contents of the Clipboard, are pasted into the **KeyCAD Pro** document as one object and not separately. Once the object has been pasted within the document, resizing of the object can be performed.

NOTE: To paste a drawing, object or text from any kind of *Paint* product, remember the applications draw the images on pixel basis dot by dot (bitmapped). The user cannot edit the objects that appear within the boundaries of the pasted object. These objects cannot be Ungrouped or Unjoined. Files created in a Paint application are called *Raster* files.

If *Cut*, *Copy*, and *Paste* are used to transfer objects from applications that support vector objects such as **KeyCAD Pro**, the user can edit, ungroup, or unjoin these objects. Files created in a draw application are called *Vector* files.

Clear

The *Clear* command removes (clears) selected objects or text from the design. The *Clear* command does not place the cleared object on the Clipboard. *Clear* is used to delete or permanently remove any selected object or text. This is useful to clear an object from the design, while maintaining the contents of the Clipboard. The DEL key functions in the same manner as the *Clear* command.

NOTE: The *Undo* command may be used to cancel the most recent operation.

How to Clear an Object

1. Select the object or text to be cleared.
2. Pull down the **Edit** menu and select Clear or press DEL.
3. The object selected is removed from the design. If necessary *Undo* can be used to retrieve the object.

Select All

The *Select All* command allows for the selection of all the objects within the current design including text and geometric shapes. The vertex points of each object are highlighted indicating that all the objects are selected.

Upon choosing the objects within the design, it is easy to move them to a different location.

Moving objects with Select All

1. Pull down the **Edit** menu and choose the *Select All* command or press CTRL A. Notice that all the vertex points are highlighted.
2. Click on the Selection tool (Arrow tool), from the *Edit* toolbox.
3. Place the cursor on top of one of the selected objects and hold down the mouse button while dragging the objects to a new position. Watch the readouts at the bottom of the screen for a precise location.

Another use of the *Select All* command, is to select all objects before choosing a command that affects the entire file. For example, all the objects within the design can be locked to a specific grid point, or even group all of the objects together. To clear everything from the design, choose the *Select All* command, and press DEL, or select *Clear* from the **Edit** menu. This is easier than selecting each individual object.

Using Select All to Move Objects

With the *Select All* command subsequent mouse repositioning moves all the objects together. Individual objects cannot be repositioned but the place of the entire design can be changed by dragging the objects to a new location. If the entire design is bigger than the monitor, choose *Fit to Window* to assist in design adjustment.

Resizing with Select All

Once the *Select All* command is chosen, each individual object cannot be resized. If a handle is dragged, it proportionally and simultaneously resizes all of the selected objects. For example, if a handle is dragged to reduce the size of one object, all selected objects are reduced by the corresponding amount that the mouse is dragged. However, the relative distance between all objects does not change. The proportional distance from the center of one object to the center of the next object remains the same. Only the object size changes either by enlargment or reduction. Objects can be reduced by pushing the mouse towards the center of an object and enlarged by dragging it away.

An entire design can be reduced or enlarged yet maintaining proportional spacing between objects. First, select the *Select All* command from the **Edit** menu, and then choose the *Group* command found under the **Options** menu. Then select the *Resize* tool and grab a selected handle, and begin reducing or enlarging. An example for this option is to reduce the entire size of a design to fit on a particular size of paper.

Detect All

The *Detect All* command should not be confused with the *Select All* command. These two commands have completely different end results. The *Detect All* command can be accessed under the **Edit** menu, or by pressing CTRL K.

Detect All is designed to undo the Hide Selection command also found under the **Edit** menu. The Hide Selection command hides any selected objects from being selected again. To select an object which is currently hidden use the Detect All command.

Example

- First, select one or more objects, and then select the *Hide Selection* command from the **Edit** menu; the chosen objects cannot be now selected.
- To deactivate *Hide Selection* choose *Detect All* from the **Edit** menu.

This feature is useful when working in very detailed design areas and problems are encountered selecting a particular point due to the close proximity of several points.

Hide Selection

The *Hide Selection* command is used to select any object or groups of objects such as lines, rectangles, circles, polygons, and hide them to prevent selection with any of the tools.

Example

1. Select the objects to be hidden using the Selection tool from the Edit toolbox.
2. Now, pull down the **Edit** menu or press CTRL J to activate the *Hide Selection* command.

- To release the *Hide Selection* command, select Detect All from the **Edit** menu or press CTRL K.

This feature is useful when working in very tight design areas and problems are encountered selecting a particular point due to the close proximity of several points.

Duplicate

The *Duplicate* command immediately copies any and all selected objects, including text blocks. The duplicate objects are placed in the current window. This command does not place a copy of the selected objects onto the Clipboard, therefore the *Paste* command is not necessary. *Duplicate* is an option that allows copies of any selected object in one step.

The *Duplicate* command is found under the **Edit** menu, or press CTRL D.

The *Duplicate Offsets* affect the location of the newly duplicated objects. Refer to the upcoming section called *Duplicate Offsets*, for more information.

Using Duplicate

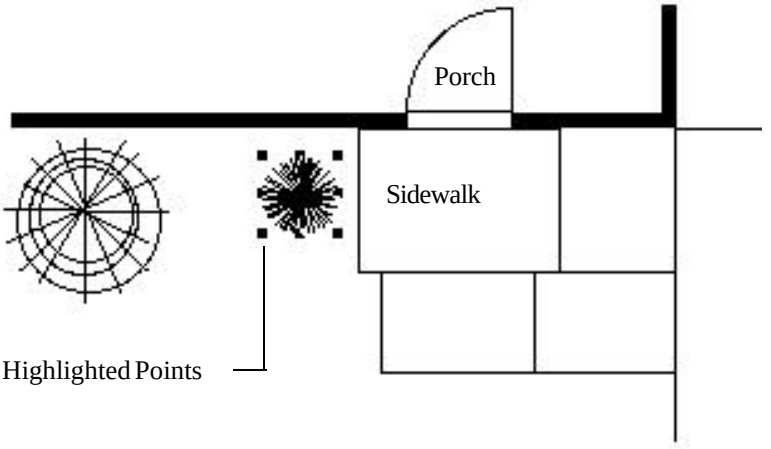
When using the *Duplicate* command, first select the object(s) to duplicate. Then, pull down the **Edit** menu and select the *Duplicate* option, or press CTRL D. The copy that appears is slightly offset from the original. Once the object is selected, it can be edited, reduplicated or relocated.

If objects that appear on different layers are selected and duplicated, the duplicates appear on the same layers as the original objects. To duplicate an object and place it on a different layer, use the *Cut*, *Copy* and *Paste* commands. This copies the object and places it on the Clipboard. Pasted objects appear in the active layer.

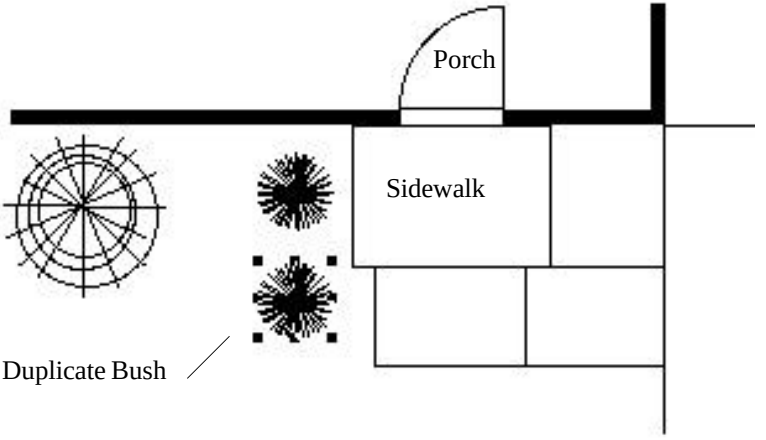
Example

The example below shows the duplication of a bush for a landscape layout.

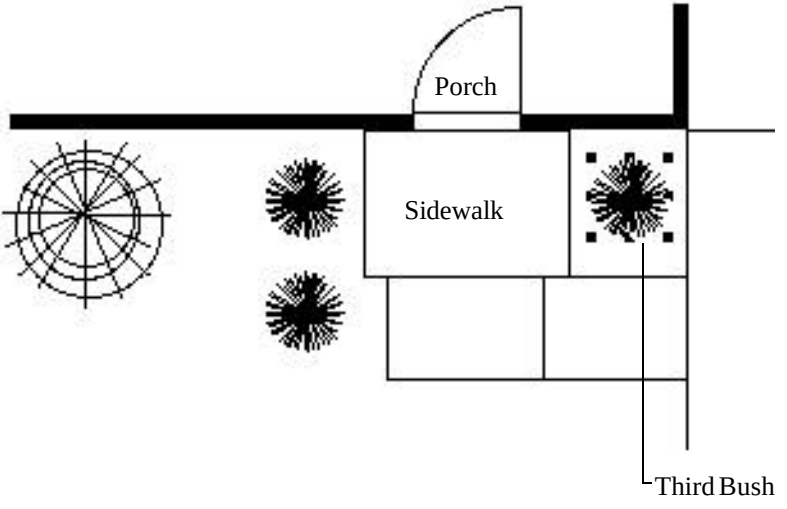
1. Select the bush as shown below. Notice the highlighted points around the object.



2. Now pull down the **Edit** menu and select *Duplicate*, or press CTRL D. After selecting *Duplicate*, notice the new object is highlighted. Use the Selection tool to drag the duplicate to a new location.



3. After moving the second duplicate bush, select the *Duplicate* command and make a third bush. Reposition the third bush to its proper location.



4. When moving the objects after duplication monitor the readouts at the bottom of the screen to assist in relocating the selected object to a precise coordinate.

NOTE: The Duplicate Offsets Dialog box determines the offset distance of the duplicate object(s). The *Duplicate Offsets Dialog* box can be accessed from the **Edit** menu.

Duplicate Offsets...

The *Duplicate Offsets Dialog* box is useful for laying out charts and other design elements such as evenly spaced bolt patterns or multiple shrubs in a landscape design. The *Duplicate Offsets Dialog* function duplicates a selected object requiring a number of evenly spaced copies.

The *Duplicate Offsets Dialog* can be accessed by pulling down the **Edit** menu. After typing the Horizontal and Vertical offsets together with the number of Repeats, click the OK button or press ENTER to close the Dialog.

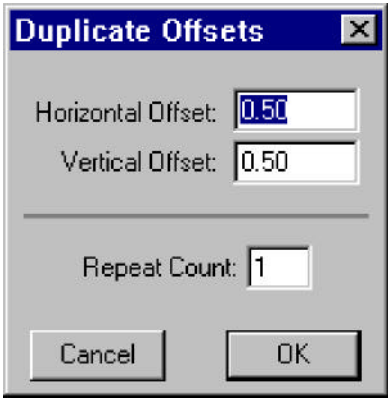
The Horizontal or Vertical offsets are changed within the *Duplicate Offsets Dialog* box. The *Duplicate* command duplicates the number of objects and their offsets in correlation to the settings within the *Duplicate Offsets Dialog* box.

All *Duplicate Offset* objects are measured from the origin of the preceding object. Once the *Duplicate Offsets* and *Repeat Count* of a selected object is chosen, the position of the first copy is measured from the origin of the original object. The position of the second copy is measured from the origin of the first duplicate copy. This process is repeated as many times as the *Repeat Count* entry displays.

Copies of objects created via the *Duplicate Offsets* or *Duplicate* command are placed in front of the original. This applies to each duplicated object regardless of the *Repeat Count* number.

NOTE: The *Duplicate Offsets* and *Duplicate* command do **not** affect the contents of the Clipboard.

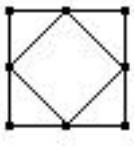
The example below shows the actual *Duplicate Offsets Dialog* box settings found within the program.



The Duplicate Offsets Dialog box

How to use the Duplicate Offsets

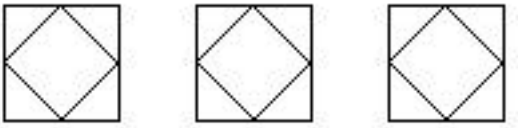
1. First, select the object to be duplicated by using the *Duplicate Offsets Dialog* box.



The Selected object

2. After selecting the object, pull down the **Edit** menu and select *Duplicate Offsets*. After the Dialog box appears type in a Horizontal Offset of 1.00" and a Vertical Offset of 0", and a Repeat Count of 2.
3. Now click the OK button or press ENTER

The design repeats as shown below.



The Objects after a Duplicate Offset command

Mirror Horizontal

The *Mirror Horizontal* command is used to flip any selected object or group of objects in a diametrically different direction. Select an object and perform the Mirror command to make a duplicate of the object flipped horizontally and placed by a certain offset.

The *Mirror Horizontal* command is found under the **Edit** menu. For quick access to the *Mirror Horizontal* command, activate the Menu Bar and click the *Mirror* button

. **Mirror Button**

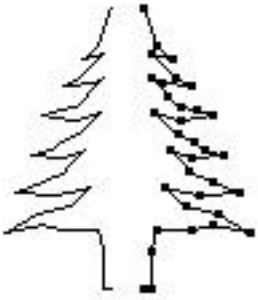
The Menu Bar can be turned On or Off by pressing CTRL M or by pulling down the **Layout** Menu and selecting Menu Bar.

How to use Mirror Horizontal

The following example shows how to use the *Mirror* command to create a frontal view of a tree. Creation of the first half of the tree is devised using the Open Polygon tool.

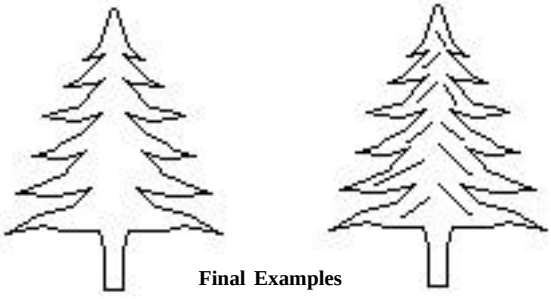
1. First select the object or group of objects to mirror. Use the Selection tool for selecting objects.
2. Once the object is selected, pull down the **Edit** menu and select *Mirror Horizontal*, or press the *Mirror* button on the Menu Bar.

3. Notice that after the *Mirror* command is selected, the newly duplicated copy is selected and slightly offset in the Horizontal axis. This is so the object can be moved or edited if necessary.



4. Now that the second half of the tree is created, use the Selection tool to drag the mirrored copy in alignment with the original half of the tree. The example below shows the end result after alignment of the mirrored objects.

5. Once the mirrored objects are aligned, take the Line tool and make accent lines within the trees boundary lines. This adds a more realistic appearance.



Final Examples

Flip

The *Flip* command flips any selected object or group of objects horizontally or vertically. When flipping objects, the program uses the center of the selected objects for a reference point.

The *Flip* command is found under the **Edit** menu. The *Flip* options are found within a pop-up menu.

Hold down mouse button to access pop-up Menu

For quick access to the Flip Horizontal and Vertical commands use the Menu Bar buttons. To turn the Menu Bar On or Off, pull down the **Layout** menu or press CTRL M.

Flip Vertical Button

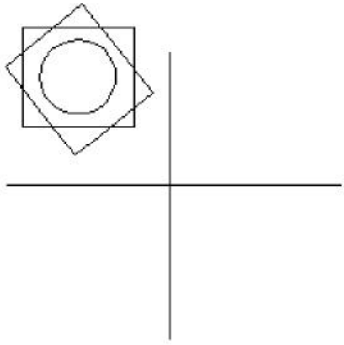
Flip Horizontal Button

Example

The Flip Horizontal command is useful for creating objects that should be symmetrical along a vertical axis. An example of this, is if a free-formed shape is drawn, copy the shape, and then reposition and flip the copy to produce two free-formed shapes with symmetrical curves and lines. The Vertical Flip command works in the same manner, except the objects flip on the Vertical axis.

Flip Horizontal flips the object along the Horizontal axis. Think of it as flipping the object to the right of the original object.

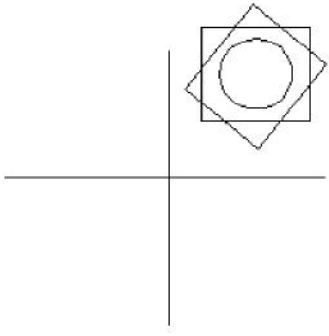
Vertical Axis Line
Up or Down



Horizontal Axis
Left to Right

Flip Vertical flips the object along the Vertical axis. Think of it as flipping the object down from the original object.

Vertical Axis Line
Up or Down

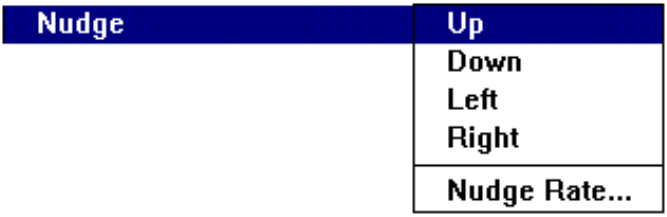


Horizontal Axis
Left to Right

Nudge

The *Nudge* commands are very useful to move selected objects in small or large increments without using the Selection tool (Arrow tool). It sometimes becomes difficult to move selected objects using the Selection tool (Arrow tool). With this in mind, **KeyCAD Pro** has designed several convenient ways of Nudging (moving) selected objects. The distance an object is to be nudged can be set, by accessing the *Nudge Rate Dialog* box.

To access the *Nudge* command pull down the **Edit** menu, highlight *Nudge* and a pop-up menu appears, from which the user can select either *Nudge UP, DOWN, LEFT,* or *RIGHT*. The *Nudge Rate Dialog* box can also be accessed.



The Nudge Pop-up Menu

The Nudge Buttons

KeyCAD Pro has placed Nudge buttons on the Readout display window. The display buttons support two types of Nudge, i.e., *Incremental Nudge* and *Absolute Nudge*. The following examples display two types of *Nudge*.

Nudge Absolute Buttons

Nudge Incremental Buttons

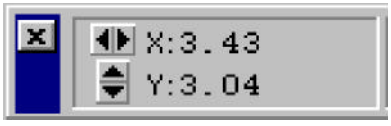
Nudge Absolute

The *Nudge Absolute* buttons are used to nudge any selected object(s), and individual vertex points from the *Absolute* reference point (Origin Point). The *Absolute (Origin) Point* is where the axis lines meet. The program defaults to the lower left of the screen.

For example, select an object then click on one of the *Nudge* buttons, either to the left or to the right and up or down. The selected object moves in that direction according to the distance set within the *Nudge Rate Dialog* box.

NOTE: Remember the reference point from which the *Nudge* is calculated, is the Absolute (Origin) point of the design.

Nudge Absolute Buttons



Nudge Left or Right

- **Nudge X (Left or Right)**, when clicking on these buttons the object moves along the X-Axis.

Nudge Up or Down

- **Nudge Y (Up or Down)**, when clicking on these buttons the object moves along the Y-Axis.

- Click the Nudge buttons as many times as necessary.

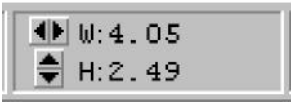
Nudge Incremental

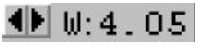
The *Nudge Incremental* buttons are used to nudge any selected object(s) and individual vertex points in Incremental coordinates. Incremental means that the selected object is resized in reference to the original size of the object or points. For example, if a rectangle is selected and then click on the **W** Nudge buttons, the rectangle changes in width, which changes the size of the selected object. The nudge distance is set in the Nudge Rate Dialog box.

If an object is selected then click on one of the Incremental Nudge buttons, either **W** left or right and **H** up or down, the selected objects change in Width or Height according to the distance set within the Nudge Rate Dialog box.

When using the **W** or **H** nudge buttons on selected objects, the width and height are referenced from the center of the selected object.

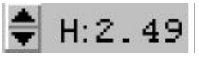
Incremental Nudge Buttons





Nudge Width (Left or Right)

- **Nudge W**, when clicking on these buttons the object changes along the X-Axis.



Nudge Height (Up or Down)

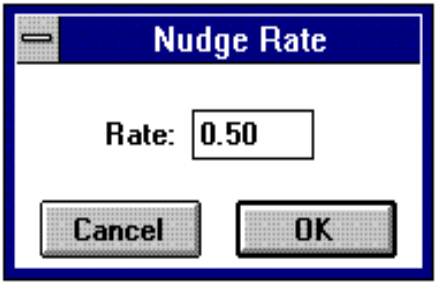
- **Nudge H**, when clicking on these buttons the object changes along the Y-Axis.

Setting the Nudge Rate

When using the *Nudge* commands, the distance an object moves each time the Nudge command is used can be specified. To access the *Nudge Rate Dialog* box, pull down the **Edit** menu and drag the cursor on top of the word Nudge. A pop-up menu displays. Now select the *Nudge Rate...* Once the dialog box appears the nudge rate can be entered. Click OK after entering the correct rate. Click on the Cancel button to cancel and return to the previous setting.

The *Nudge Rate Dialog* affects all of the *Nudge* commands.

NOTE: The *Nudge Rate* is always tied to the current scale of the design.



The Nudge Rate Dialog box

Smoothing

This command is designed to round the angles of polygons, splines and unsmoothed freehand shapes. This command can also work conversely by changing the rounded angles back into sharper angles.

The *Smoothing* commands can be accessed under the **Edit** menu; hold down the mouse button on top of the word *Smoothing* and a pop-up menu appears. From the pop-up menu the *Smooth*, *Unsmooth*, and *Tension Dialog* box can be accessed.

The Smoothing Pop-up Menu

- **Smooth**

  Smooth changes selected polygons to smoothed splines. *Smooth* rounds the corners of polygons and unsmoothed free-formed shapes.

- **Unsmooth**

  The Unsmooth command changes splines, smoothed polygons and any smoothed free-formed shapes into polygons with sharp angles.

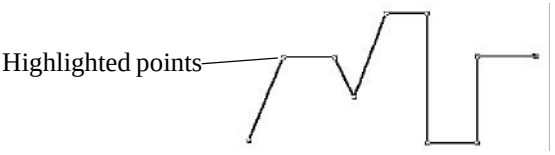
- **Tension...**

  This command is used to access the *Tension Dialog* box and change the tension value from which the *Smooth* command is tied. The smoothness of selected objects can be loosened or tightened.

Using the Smooth command

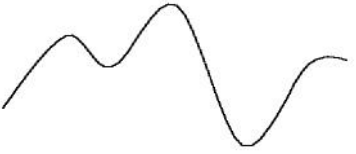
The following example shows the effects of the *Smooth* command when applied to a polygon.

1. First, create a simple open polygon, then select the polygon by using the Selection (Arrow) tool, so that all of the points within the polygon are highlighted as shown.



An Unsmoothed Polygon

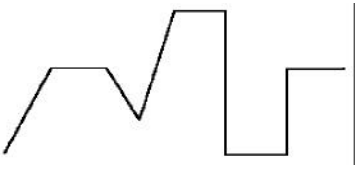
2. Now, select the *Smooth* command from the *Smoothing* pop-up menu. This command automatically smooths the angled corners of the polygon.



A Smoothed Polygon

Unsmooth

1. After smoothing the polygon, select the smoothed polygon and choose *Unsmooth*. The polygon returns to its original angular shape.



An unsmoothed Polygon

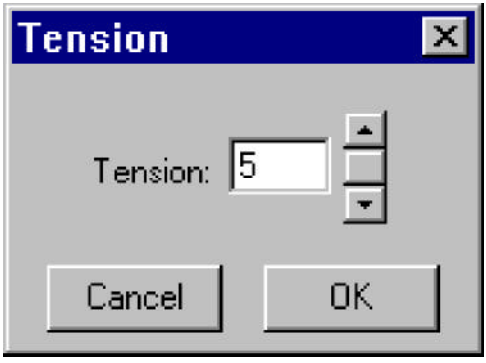
Tension Dialog Box

The *Tension Dialog* box is accessed to set the amount of smoothness that is to be applied when the *Smooth* command is used.

The *Tension Dialog* box can be selected from the *Smoothing* pop-up menu, which is found under the **Edit** menu.

When entering the tension value, the smaller the number the tighter the curves are going to appear. If the number is too small, the corner points become angular. This has a similar effect to the *Unsmooth* command.

If the tension value is increased, the curve smoothness becomes looser, taking on a larger curve effect. If the value is too high, objects can be made with no straightness.

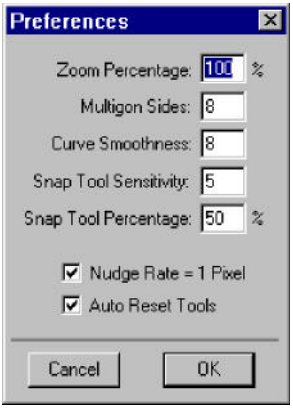


The Tension Dialog box

Preferences...

This Dialog box is used to personalize **KeyCAD Pro**. The program defaults for tools and commands can be changed. The *Preferences* options are useful to adjust the **KeyCAD Pro** environment to fit the user's working habits.

The *Preferences Dialog* box can be accessed under the **Edit** menu. The Dialog permits a choice of several options.



The Preferences Dialog box

Zoom Percentage

This setting determines the percentage that the *Zoom* tool Zooms In or Out.

Multigon Sides

The number in this box represents how many even sides are drawn when using the *Multigon* tool found in the *Draw* toolbox.

Curve Smoothness

KeyCAD Pro uses splines when creating any curves within the program. This setting is used to determine how many points are placed on each curve segment. The higher the number, the smoother the curves appear. The lower the number, the rougher the look to the curves

Snap To Tool Sensitivity

This setting determines how close the cursor should be to the object outline for the *Snap To* tools to function. The number represents how many pixels away from the object the cursor can be.

Snap To Tool Percentage

The percentage value entered in this box determines the point on the line the Percentage *Snap To* tool snaps to. It can be set from 0 to 100%.

Auto Reset Tools

This option can be turned On or Off. An **X** in the box indicates that it is On. This option when turned On, resets the tool selected back to the Arrow tool (Selection tool), after the selected tool has been used once. If this option turned Off, then the tool selected stays selected until another tool is chosen.

Show Clipboard

The *Show Clipboard* command opens the Clipboard. The Clipboard window shows the objects or text recently cut or copied to the Clipboard. The Clipboard is used to copy (duplicate), or move selected objects from any part of the design to another part, or even to a different file. Objects can also be moved to other applications that support the *Clipboard Cut, Copy*, and *Paste* commands.

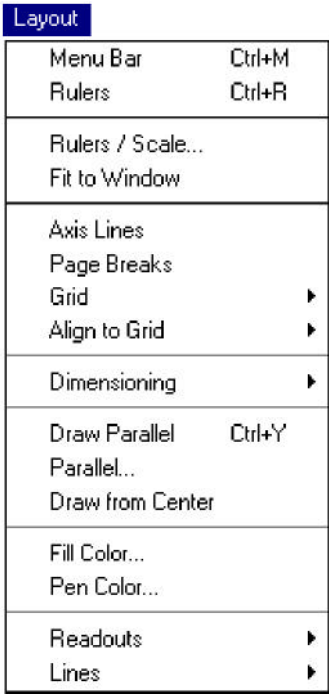
The *Show Clipboard* window is accessed under the **Edit** menu.

When *Show Clipboard* is chosen, a window appears which displays the most recently copied object or text from the Clipboard. To close the window click on the Close button of the window.

The Layout Menu

When using the **Layout** menu, think of the commands as items that customize design requirements. The **Layout** menu allows the user to access commands such as turning the Menu Bar On or Off, Drawing Scale, Fit to Window, Axis Lines on or off, Grid specifications, and many other commands used for creating, viewing, and measuring objects.

Several commands have keyboard equivalents for fast access, when combining the CTRL key (or COMMAND key on the Macintosh) with certain letters as shown in the menu below.



The Layout Menu

Menu Bar

The *Menu Bar* is designed to let the user work efficiently and quickly within a design. The Menu Bar can be turned On or Off by selecting the *Menu Bar* command under the **Layout** menu or pressing CTRL M.

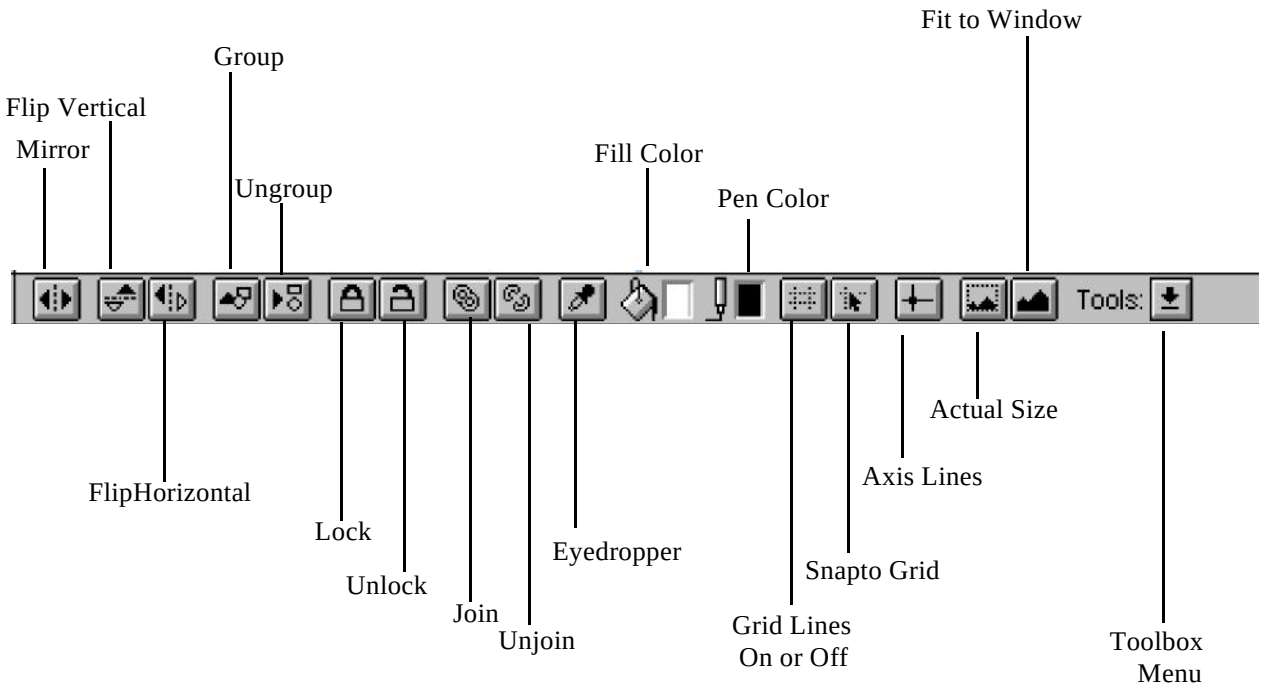
The commands, tools, and options found within the Menu Bar can be accessed within the various menus found at the top of the screen.

The buttons and pull-down menus have been designed with minimal size. They follow along the edge of the title bar to minimize space utilization and should not interfere with design work.

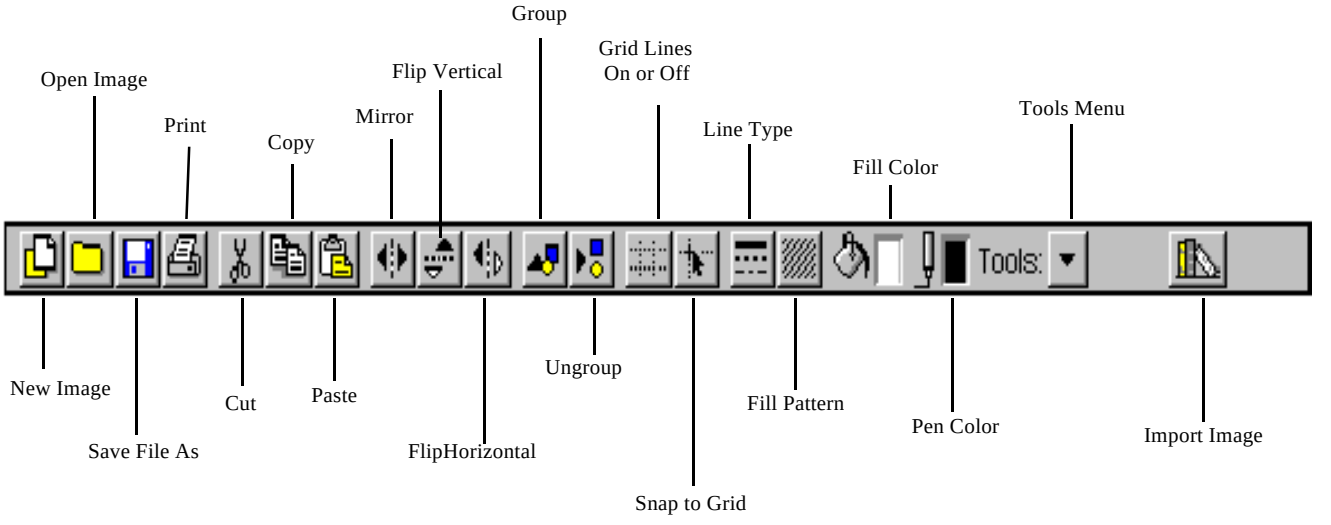
If the monitor that is being used is smaller than the width of the Menu Bar, the program cuts off the options furthest to the right. It cuts off one option at a time, until the width of the Menu Bar fits within the screen size.

A brief definition of each option found within the Menu Bar is provided on the following pages. Further information on these options is available by referring to the menu that contains the required option. The Menus have more detailed examples on how to use these options in the creation of a design. Refer to the Table of Contents at the beginning of this manual for a list of the commands contained within each menu.

Menu Bar Layout with Names (Windows)



Menu Bar Layout with Names (Macintosh)



Mirror
This button is used to duplicate and flip a selected object. The mirrored object is slightly offset from the original. To use the *Mirror* command, first select the object then click on the Mirror button within the Menu Bar. Individual objects or groups of objects can be mirrored.

Flip Vertical
This button is used to *Flip* selected objects along the vertical axis, comparable to flipping the selected object upside down. Individual objects or groups of objects can be flipped.

Flip Horizontal
This button is used to *Flip* selected objects along the horizontal axis, comparable to flipping the selected object to the right. Individual objects or groups of objects can be flipped.

Group
This button is used to consolidate a set of selected objects as one grouped object. A grouped set of objects is treated as a single object, rather than several individual objects.

Ungroup
The *Ungroup* button automatically divides a grouped object into individual objects.

Lock (Windows only)
This button is used to *Lock* the characteristics and location of any selected object or groups of objects, so that they cannot be changed.

Unlock (Windows only)
This button is used to unlock locked objects.

Join (Windows only)
This button is used to connect (Join) all selected lines or arcs that share overlapping (connected) end-points.

Unjoin (Windows only)
The *Unjoin* button disconnects all selected lines or arcs that have been previously Joined.

Eyedropper (Windows only)
The *Eyedropper* tool is used to select the current color from any color currently in use within a design. Once the *Eyedropper* tool is clicked on a color within the design that color becomes the current color. The color that was clicked on is then displayed in the *Current Color* button.

Fill Color
The current Fill color is displayed to the right of this button. To select another fill color, click once on the button to open the color picker.

Pen Color
The current Pen color is displayed to the right of this button. To select another pen color, click once on the button to open the color picker.

Grid Lines On or Off
The *Grid* button turns the Grid Lines On or Off, depending on the current status of the Grid Lines when the button is clicked.

Snap To Grid
This button is used to turn the *Snap To Grid* option On or Off.

Axis Lines On or Off (Windows only)
Clicking this button toggles the *Axis Lines On* or *Off*.

NOTE: The Axis Lines represent the Origin point, a reference site for all Absolute coordinates and Readouts.

Fit to Window (Windows only)
This button automatically makes the entire design fit within the current window size. This is useful for viewing the overall layout of a design.

Actual Size (Windows only)
This button displays a design at its actual size. This is very useful to return to *Actual Size* without using the Zoom tool or Menu command. This command may be accessed from the *Options* menu or by pressing CTRL+E.

New Image (Macintosh only)
This button is used to create a new image at the default settings. Selecting this button is the same as selecting the **File** menu, **New** option.

Open Image (Macintosh only)
This button displays the Open Image window.You can locate and open previously created documents.

Save (Macintosh only)
This button allows you to save the currently active file. If the file has been saved previously then this button saves the document over the original document.

Print (Macintosh only)
This button displays the print dialog box. This dialog box allows you to set printer options and print the currently active document.

Cut (Macintosh only)
This button removes selected objects from a document and places them on the clipboard.

Copy (Macintosh only)
This button places the selected objects in the clipboard while leaving the original objects in the document.

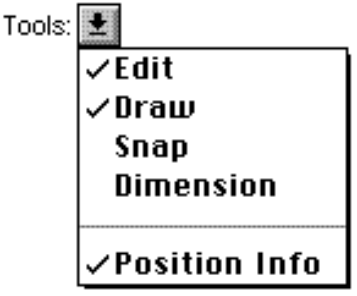
Paste (Macintosh only)
This button places the contents of the clipboard into the currently active document.

Line Type (Macintosh only)
This button allows you to select a line style for any selected objects.

Fill Patterns (Macintosh only)
This button displays the Fill Pattern Window.

Import Image (Macintosh only)
This button allows you to import a previously saved image into the currently active image.

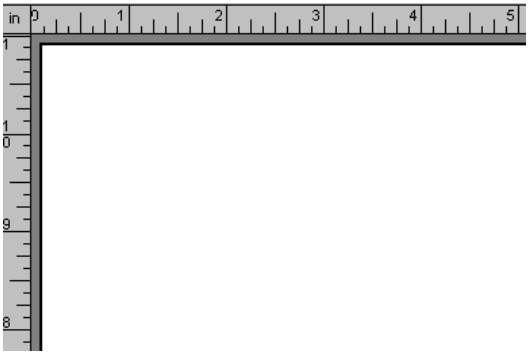
Toolbox Pull-Down Menu
This pull-down menu turns On or Off any of the toolboxes found within the program.



The Toolbox Pull-Down Menu

A checkmark beside an entry indicates that the Toolbox is currently displayed on the screen. The diagram above shows the program defaults

Rulers



To turn on the Rulers, either choose the Rulers command from the **Layout** Menu or press CTRL R. To turn them off choose Rulers again from the **Layout** Menu or press b R again.

The rulers show the current ruler scale settings of the document. To change these settings, either choose Rulers/Scale from the **Layout** Menu or, if the rulers are on, double-click on either the unit of measure readout, horizontal or vertical ruler.

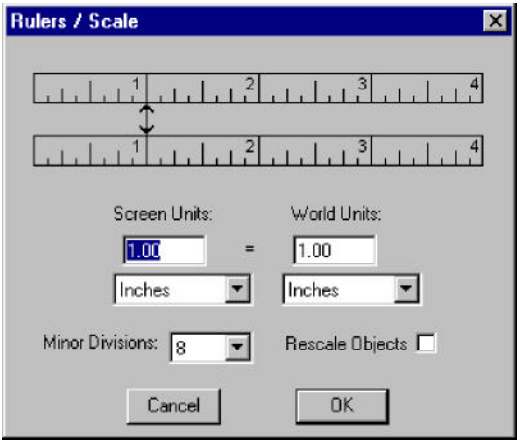
The vertical ruler does not start at 0 at the top, but at the bottom. This is because the Origin point is in the lower left.

The horizontal and vertical rulers also change if you change the document's paper size, origin point, or if you click on the Actual Size or Fit to Window buttons (or their menu equivalents) in the Menu Bar.

Unit of Measure Readout

To the left of the horizontal ruler and at the top of the vertical ruler, is the Unit of Measure Readout which indicates the current screen unit of measure.

Rulers/Scale...



The Rulers/Scale Dialog Box

The *Rulers/Scale Dialog* box is used for setting the scale to work in when creating a design. **KeyCAD Pro** supports all of the scales used in today's engineering fields. It is also used to set the unit of measure and the number of divisions a unit on the ruler will be divided into.

Scale factors are very useful when working on precise designs that are larger than the paper needed to print the design. Working with scaled drawings, requires knowledge of how a design proportionally fits with respect to the real world coordinates.

The program defaults to 1" (in.) on the screen equals 1" (in.) in the real world and 16 ruler divisions. Once the scale and ruler settings are changed they stay at the new setting until they are changed again. Below are some example scales that are commonly used.

1/16 in. = 1 ft.
1/8 in. = 1 ft.
1/4 in. = 1 ft.
1/2 in. = 1 ft.

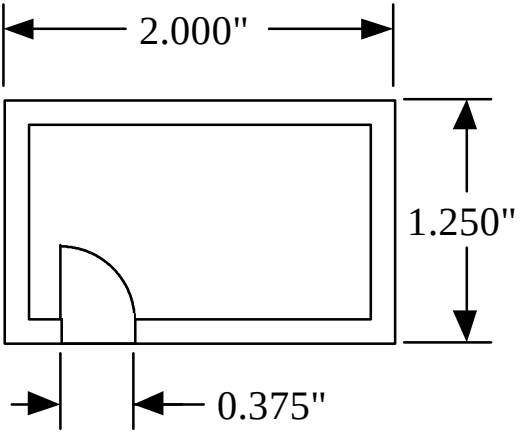
These scales are more commonly used in Architectural designs. Since most Architectural designs are very large in real life, a small scale factor is needed to permit the design to fit on a reasonable size of paper.

KeyCAD Pro *Rulers/Scale Dialog* supports both English and Metric scales.

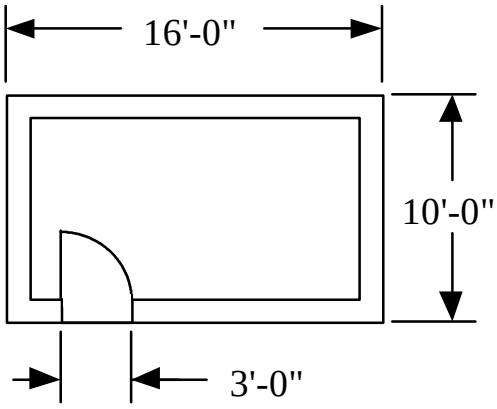
Example

Below is a simple room presented in two different scales to assist in the comprehension of how scales can benefit a design.

1. Create the room **1" = 1" scale factor**. Notice that the size of the room is very small in comparison to the real world size of the same room.



2. Now create the same room but use **1/8" = 1' scale factor**, to show the real life measurements.



Setting the Drawing Scale

Set the *Drawing Scale* at the begining of the design, when a new file is first started.

When setting the scale factor consider setting the Display options to display the proper units of measurement.

1. Pull down the **Layout** menu and select *Rulers/Scale* or, if the rulers are on, double click on either the unit of measure readout, horizontal or vertical ruler. A dialog box like the one shown on page 8-10 appears asking the user to set the desired scale, unit of measure, and number of ruler divisions. At the top of the dialog box there is a sample view of what the rulers will look like if they are turned on.

2. Type the desired screen unit in the edit field under the Screen Units: heading.

3. Type the desired world unit in the edit field under the World Units: heading.

4. Choose the desired unit of measure for the screen scale from the pop-up list box below the screen scale edit field.

5. Choose the desired unit of measure for the world scale from the pop-up list box below the world scale edit field.
6. Choose the desired number of ruler divisions from the Minor Divisions pop-up list box.

❒ To rescale all of the objects in the drawing to the new settings, make sure the box next to Rescale Objects is turned on. To turn it on, click once on the box, an X appears in the box to indicate that it is on. If it is on (X in the box) and you want to turn it off, click once on the box and the X disappears.

The readouts at the bottom of the screen change accordingly to reflect any changes.

The Sample Ruler

There are two sample rulers displayed at the top of the dialog box. The top ruler shows the screen unit of measure (inches, millimeters or centimeters). The bottom ruler shows what the rulers will look like if they are turned on. The lines on the rulers are called "tick marks" and vary in height depending on whether there is a major or minor division of the unit of measure.

Between the two sample rulers is an arrow that shows how many screen units make up a screen unit of measure (inch, millimeter, centimeter).

As the values for the screen units, world units, screen unit of measure and number of ruler divisions are changed, the bottom ruler is updated to show what effect changes will have on the rulers. As the values for the screen unit of measure and number of divisions are changed, the top ruler is updated. As the values of screen units, world units and screen unit of measure are changed, the arrow between the rulers is updated.

Screen Units

In the edit field below the heading Screen Units: the desired screen unit can be entered. Below the edit field, there is a pop-up list box where the screen unit of measure is chosen. Cchoose from Inches, Millimeters, or Centimeters.

NOTE: Millimeters are shown on the sample rulers in groups of 20 millimeters instead of individual millimeters. Each tick mark on the rulers is 2 millimeters.

World Units

In the edit field below the heading World Units: the desired world unit can be entered. Below the edit field, there is a pop-up list box where the world unit of measure is chosen. Cchoose from Inches, Feet, Yards, Miles, Millimeters, Centimeters, Decimeters, or Meters.

Minor Divisons

The Minor Divisions list box allows enables the user to change the number of divisions that a unit of measure will be divided into. The number of divisions are indicated by tick marks. If inches are chosen, the possible number of divisions listed are 1-16. If millimeters are chosen, the Minor Divisions list box is hidden and the program uses 5 divisions. To unhide the list box, chose either inches or centimeters. If centimeters are chosen, the possible number of divisions listed are 1-5.

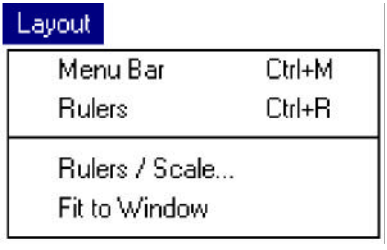
Rescale Objects

The *Rescale Objects* box can be selected to automatically update any previously drawn objects. If the *Rescale Objects* is not selected, then only the objects drawn from this point take on the new scale factor. This allows multiple scale factors within one document.

Fit to Window

This command provides an overview of the entire document by fitting it to the current window size. This command is useful for designs that cover several pages. The view can be changed to see how the whole design looks, and ensure a proper fit on the selected paper size. **KeyCAD Pro** reduces or enlarges the size of a design to permit viewing of it in its entirety within the current window size.

The *Fit to Window* command can be selected under the **Layout** menu.



Fit to Window Menu command

KeyCAD Pro also allows the option to use the *Fit to Window* button on the Menu Bar. To activate the Menu Bar, either pull down the **Layout** Menu or press CTRL M.

Axis Lines

The *Axis Lines* command turns On or Off the Axis lines on the screen. The axis lines represent where the X & Y axis lines meet. This is considered the Absolute reference point of the design. When the program is first started the axis lines appear in the lower left-hand corner of the screen. The axis lines can be moved by selecting the *Move Origin* under the **Options** Menu.

X-Axis runs horizontal or (left to right).

Y-Axis runs vertical or (up and down).

To access the *Axis Lines* command pull down the **Layout** Menu. A check mark beside the name indicates that the axis lines are on.

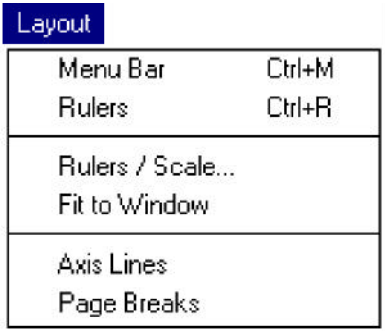
KeyCAD Pro offers the option of using the Axis Lines button found on the Menu Bar. The *Menu Bar* command can be activated under the **Layout** Menu, or press CTRL M. Click on the button, to turn the lines On or Off, like a toggle switch.

Page Breaks

This command permits viewing of how a drawing divides into pages for printing purposes. It shows the page breaks of any file (document). A page break appears as dashed lines on the screen. The design may extend over the page break lines. The dashed lines *DO NOT* print; they represent where **KeyCAD Pro** divides the drawing into printed pages. The page break lines are in accordance with the page size and orientation selected in the *Page Setup Dialog* box.

Page break lines cannot be moved or adjusted on the page where they appear on the design. **KeyCAD Pro** automatically sets the page break positions based upon the settings chosen in the *Paper Size* and *Page Setup Dialog* boxes.

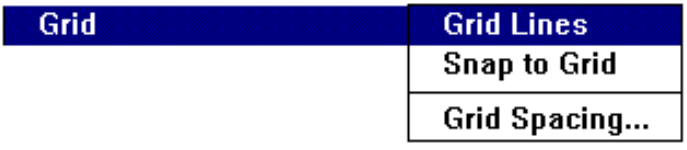
- ❒ To display Page Breaks choose the *Page Break* command from the **Layout** Menu.



- To hide the page breaks, choose the *Page Breaks* command again and the lines disappear. The *Page Breaks* command works as a toggle switch.

Grid

The *Grid* options are accessed through the *Grid* pop-up menu, which is found under the **Layout** menu. Drag the mouse cursor on top of the word Grid and a pop-up menu appears as shown below. Within this menu the user can choose to turn the *Grid lines* On or Off, *Snap To Grid*, and set the *Grid Spacing*.



The Grid Pop- Up Menu

Grid Lines

Grid lines are evenly spaced on the screen and can be used as reference points when creating a design. These lines assist in accurately positioning and adjusting the size of objects. The space between each grid line can be set to a specific distance. To change the grid spacing, select the Grid Spacing command; a Dialog box appears which allows the grid spacing to be set.

The Grid Lines option works like a toggle switch, and the check mark beside the option means that the grid lines are On. Select the Grid Lines option to turn them Off and the check mark disappears.

To turn the Grid Lines On or Off click on the Grid Lines button found on the Menu Bar. The Menu bar appears under the title bar of the window. The Menu Bar can be turned On or Off from the **Layout** menu or by pressing CTRL+M.

Snap To Grid

To ensure that objects drawn conform exactly to the Grid, turn On the Snap to Grid option. When the Snap to Grid option is turned On, **KeyCAD Pro** restricts the sizing, placement, and drawing of objects to the spacing of the grid.

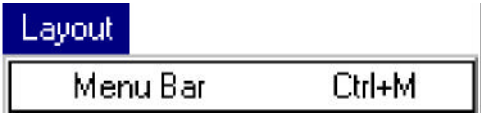
Objects only change size in increments of the grid spacing when the Snap to Grid option is activated. Objects cannot move to positions between the grid points. The boundary lines of the object can be moved to align only with the grid lines.

Turning on the Snap to Grid

To turn on the Snap to Grid option select it from the Grid pop-up menu found under the **Layout** menu. The option works like a toggle switch, and when the Snap to Grid is On, a check mark appears beside the name.

Alternatively turn the Snap to Grid On or Off by clicking on the Snap To Grid button found on the Menu Bar. Turn the Menu Bar On or Off by selecting the command under the **Layout** menu or by pressing CTRL+M.

For more details concerning the Menu Bar refer to the *MenuBar* section of this chapter.

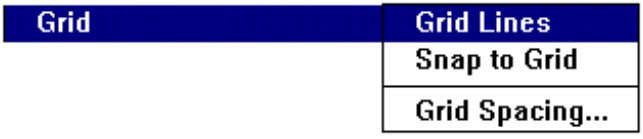


The Menu Bar Command

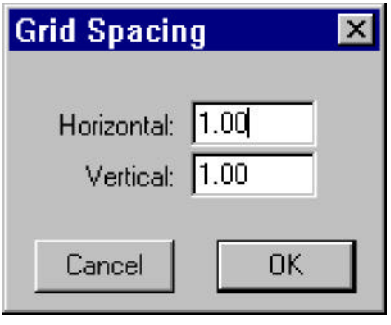
Grid Spacing...

The spacing between the horizontal and vertical grid lines can be set by accessing the *Grid Spacing Dialog box*. The grid spacing can be set for both the Horizontal and Vertical lines.

Access the Grid Spacing Dialog box by selecting the Grid Spacing option found in the Grid pop-up menu, which is found under the **Layout** menu.



With the appearance of the Dialog box, type in the horizontal and vertical distance required. The horizontal and vertical distances do not have to be the same. Once the desired settings have been entered click on the OK button and the Grid lines are automatically updated to reflect the new settings. The Dialog can be cancelled by clicking on the Cancel button, and the program returns to the previous settings.



The Grid Spacing Dialog Box

Align to Grid

The *Align to Grid* command automatically aligns individual or groups of objects to the current grid spacing.

Using the Align to Grid

1. Select the object to align to, by using the Selection tool.
2. Turn on the *Align to Grid* command by selecting it from the **Layout** menu or press CTRL+R.
3. Now, select the End-Point Snap To tool and click on the vertex point within the object to align to the grid. Clicking on the point with the Snap To tool tells the program which point to snap to the grid.
4. After snapping to the desired point, while holding down the mouse button drag the object to the closest grid point and release the mouse button. The object is automatically aligned to the grid point.

NOTE: The Snap To tools must be used when aligning objects to the grid. This is so the program can identify which point of the object is to be aligned to the grid. Any of the Snap To tools can be used when using the Align to Grid command.

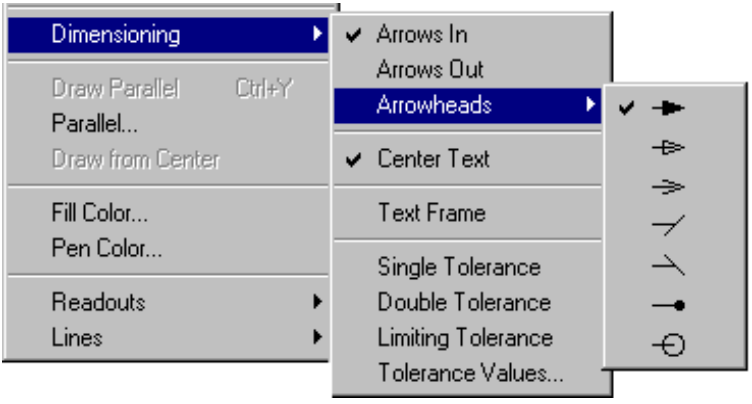
Dimensioning

The Dimensioning pop-up menu allows the user to access several options for the dimension tools. Most options work like a toggle switch, and have check marks beside them to indicate that they are on. These options include Arrows In, Arrows Out, Center Text, Text Frame, Single, Double, Limiting Tolerances.

The Tolerance Values Dialog box can be accessed from the Dimensioning pop-up menu. This Dialog is for setting the upper and lower tolerance values for the Limiting Tolerance option.

The Arrowheads pop-up menu is where the type of arrowhead for the dimension lines is selected.

Using these options enhances the custom appearance of the document and helps in using industry standard methods of drafting.

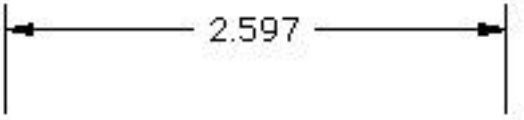


The Dimensioning Pop-up Menu

Dimensioning Options

Arrows In

This option makes the Arrows appear on the inside of the witness lines. The example below displays how the arrows look when using the Arrows In option.

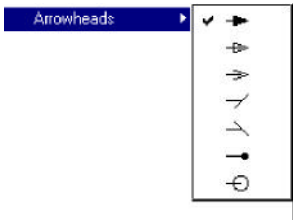


Arrows Out

This option makes the Arrows appear on the outside of the witness lines. The following example displays how the arrows look when using the Arrows Out option.

Arrowheads

The Arrowheads pop-up menu provides the Arrowhead types available for selection with dimensions. The following example displays the available options.



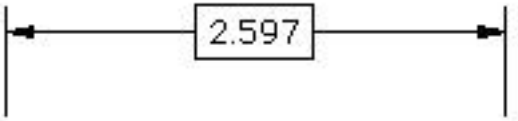
The Arrowhead Pop-up Menu

Center Text

When selecting this option the text that appears with the dimension is automatically centered between the witness lines. This is useful to alter the size of an object that is already dimensioned. The text remains in its original position until the dimension is selected and Center Text is chosen.

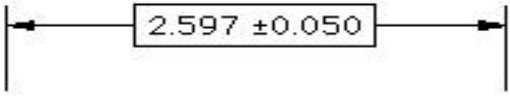
Text Frame

After selecting the Text Frame option, the dimension text automatically has a box drawn around the text. The example below displays how the Text Frame appears.

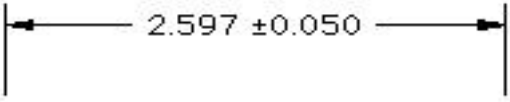


Single Tolerance

This option displays a tolerance value with the dimension text. The Tolerance option can be combined with the Text Frame option as shown in the examples below. To set this tolerance value, choose Tolerance Values from the pop-up menu and set the upper tolerance to the required specifications.



Single Tolerance with Text Frame

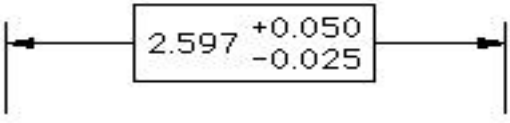


Single Tolerance without Text Frame

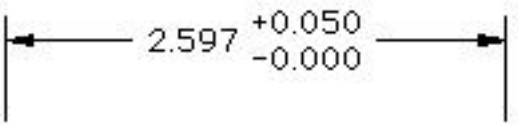
Double Tolerance

This option displays the dimensions with a double tolerance value, i.e. an upper and lower tolerance value. To set these upper and lower values, access the Tolerance Values Dialog box. The Double Tolerance option can be displayed with or without a text frame. The following examples show how the options appear with a dimension.

Examples of Double Tolerance options



Double Tolerance with Text Frame

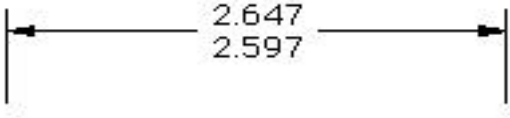


Double Tolerance without Text Frame

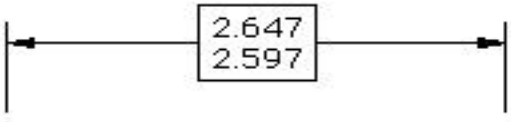
Limiting Tolerance

When choosing this option the dimensions appear with a double dimension value that has added the upper and lower tolerance values to the actual dimension. The examples below illustrate this option. The upper tolerance value is set to 0.050 and the lower tolerance value to 0.000. These values are automatically added to the original dimension. The values can be set in the Tolerance Values Dialog box found within the Dimensioning pop-up menu.

Original dimension



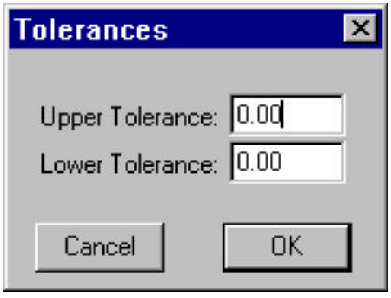
Dimension after selecting the Limiting Tolerance option



Limiting Tolerance with Text Frame

Tolerance Values

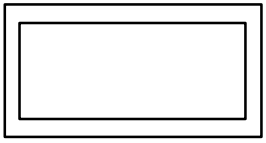
The Tolerance Values Dialog is where the upper and lower tolerance values that are applied to the Single, Double and Limiting Tolerance options are set. The following diagram illustrates the Tolerance Values Dialog box.



The Tolerance Values Dialog box

Draw Parallel

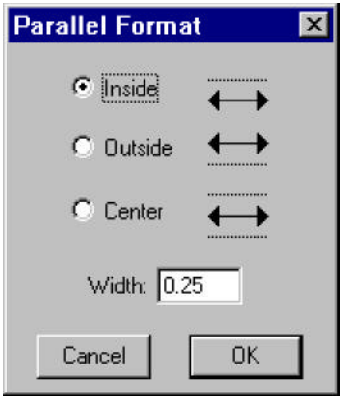
This command draws all objects with a parallel relationship. Select the Draw Parallel command or press CTRL Y before starting to draw the object. The parallel options can be set within the Parallel Dialog box, which is found under the **Layout** menu, and is discussed in the next section.



Draw Parallel Example

Parallel...

The Parallel Dialog box is used to set the parallel options for drawing parallel objects. There is a choice of drawing the parallel to the Inside, Outside, or Center of the original object. The distance that the parallel is to be offset from the original can also be specified.



The Parallel Options Dialog box

Draw from Center

This command sets the Draw tools to always draw from the objects center. When using this command the readouts display half of the objects width and height, due to the fact that it is drawing the object from the center. To draw a circle with a specific radius, select *Draw from Center* and the width and height readouts display the radius.

Fill Color...

The *Fill Color* is the color that is used to fill in an object.The Fill Color can be changed by selecting *Fill Color*... under the **Layout** menu or by clicking on the Fill Color button (paint bucket) in the Menu Bar.This brings up the color picker within the program.

The color picker is used to choose any color or grayscale that is available to the hardware.The current fill color is shown to the right of the Fill Color button in the Menu Bar.The Menu Bar can be turned On or Off from the Layout menu, or by pressing CTRL+M.

KeyCAD Pro provides an easy method to select a fill color that may have previously been chosen for an object within the drawing. Use the Eyedropper tool to conveniently change the fill color in the color picker.


Fill Color Pattern 1


Fill Color Pattern 2

NOTE: If *Fill Color* option under the **Layout** menu does not fill the object with the desire color, make sure the Fill Color Pattern 1, as shown above, is selected in the Fill Patterns Box. You can access the Fill Patterns Box under the **Options** menu. Fill Color Pattern 1 is the first pattern in the third row from the top. Fill Pattern 2 fills the object with white space.

Using the Eyedropper tool (Windows only)

1. Click on the Eyedropper tool, which is found on the Menu Bar.
2. Place the tip of the Eyedropper on top of the color to be selected, and click the mouse button.
3. Notice that the color button changes to the color that was clicked on. This new color stays active until it is changed again.

Pen Color...

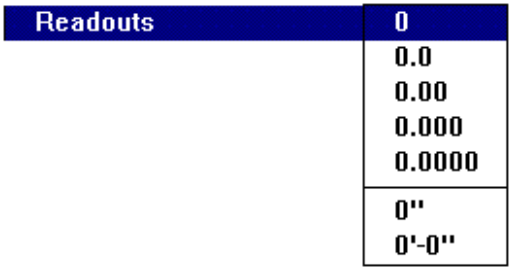
The Pen Color is the color that is used to outline an object. The Pen Color can be changed by selecting *Pen Color...* under the *Layout* menu or by clicking on the Pen Color button in the Menu Bar.

This brings up the color picker within the program. The color picker is used to choose any color or grayscale that is available to the hardware. The current pen color is shown to the right of the Pen Color button in the Menu Bar.

The Menu Bar can be turned On or Off from the *Layout* menu, or by pressing CTRL+M.

Readouts

The Readouts pop-up menu gives access to the options available for the Readouts window. From the pop-up menu the readouts can be set to display certain levels of precision. The readouts can also be set to display feet and inches. A check mark, indicates which option is active.



The Readouts Pop-up Menu

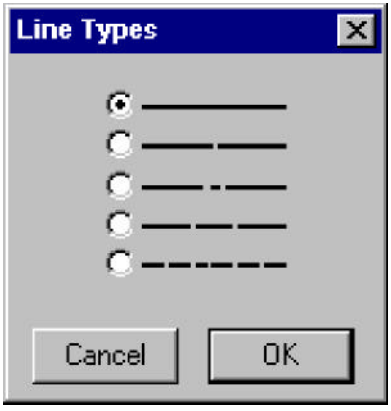
The diagram below shows the Readouts window, which appears at the bottom of the screen. These readouts can be turned On or Off, and can be moved to any location on the screen.

The Readouts Window

Lines

The Lines pop-up menu is used to choose from the different Types and Size of lines. The choice of lines consist of plain lines, centerlines, and different types of dashed lines. The line size applies to all line types.

Lines Pop-up Menu



The Line Type Dialog Box

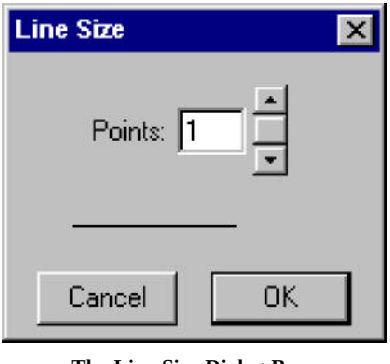
Changing Line types

To change the line type of an object, proceed as follows:

1. Select the object then pull down the **Layout** menu and access the Lines pop-up menu, to select the required line type.
2. Only the selected objects change to the new line type.

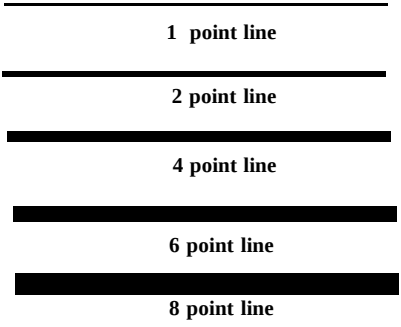
Line Size

This Dialog box is used to select the line thickness that appears when using the Draw tools. The lines are determined by point size. The Dialog box displays the line width after the point size is entered. Use whole numbers such as 1, 2, 3, etc. There is a choice from 1 to 20 on the line size.



The Line Size Dialog Box

The following examples illustrate how different lines sizes look when printed.



1 point line

2 point line

4 point line

6 point line

8 point line

The Options Menu

The **Options** menu is where various settings available to the program are selected. Some of the commands available are filling objects with *Fill Patterns*, *Lock* or *Unlock objects*, *Join* or *Unjoin* objects, *Group* or *Ungroup*, *Move Origin*, and *Actual Size*.

Several of the **Options** menu commands can also be accessed from the *Menu Bar* (refer to *Menu Bar* section) or by using the Control key (Command key on the Macintosh Version).

Menu Bar Buttons


Lock


Unlock

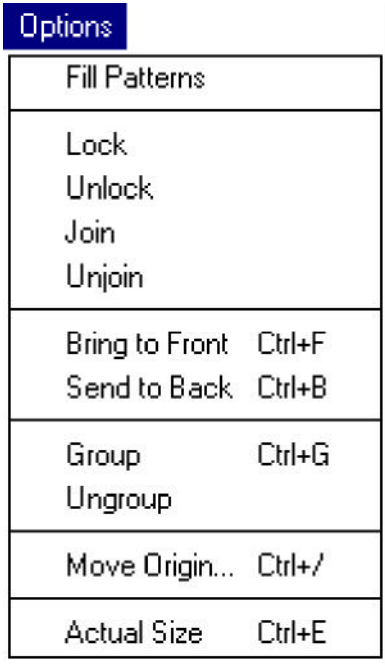

Join


Unjoin


Group


Ungroup

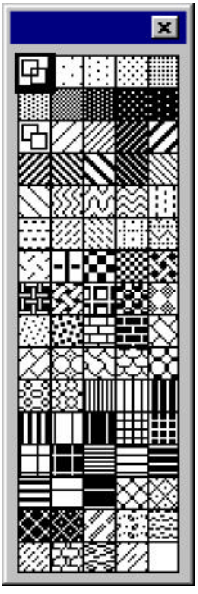
Menu Commands



Fill Patterns

The *Fill Patterns* window gives access to the various fill patterns. A fill can be applied to any object such as polygons, circles, rectangles, squares or multigons. If an object is made up of lines and arcs, the different elements of the object must first be joined together prior to filling it.

To select a fill pattern pull down the **Options** menu and drag the mouse on top of *Fill Patterns*. When the window appears, move the mouse cursor on top of the desired fill and click the mouse button.

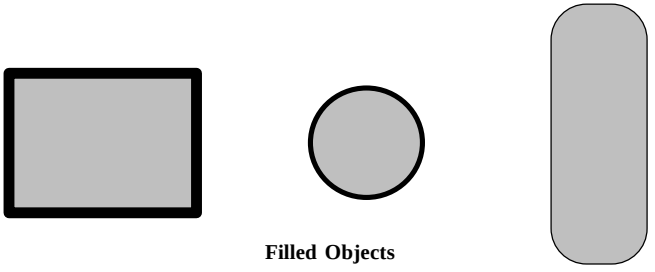


The Fill Pattern Window

Select a fill pattern; on returning to the *Fill Pattern* menu, the selected fill is identified by a white frame. This indicates that a fill is currently selected.

Applying a Fill Pattern to an object

1. Select the object to be filled. More than one object can be selected by holding down SHIFT while clicking on additional objects.
2. Now, select the desired fill from the *Fill Pattern* window.



Filled Objects

When an object is already filled and further edits are done to the size or shape of the object, the fill pattern is automatically updated to reflect the changes.

Lock and Unlock

Objects within the document can be *Locked* to prevent them from being accidentally moved, edited or deleted. When an object is locked it cannot be cut, deleted, moved, rotated, mirrored or flipped. No changes can be made to its' appearance, stacking order (layer) size, alignment to the grid, smooth, unsmooth, or shape of the object. However, the user can copy and duplicate a locked object.

If a locked object is included among objects selected for grouping, **KeyCAD Pro** keeps that object locked even though it is grouped with other objects.

The *Lock* command is found under the **Options** menu; or use the *Lock* and *Unlock* buttons found on the Menu Bar. The Menu Bar is turned On or Off from under the **Layout** Menu.

NOTE: An entire document can be protected by selecting all of the objects in the document and locking them. When locked, none of the objects can be changed.

Locking an Object

1. Select the object by clicking on it with the Selection tool.
2. Choose the *Lock* command from the **Options** menu or click on the *Lock* button found on the Menu Bar.

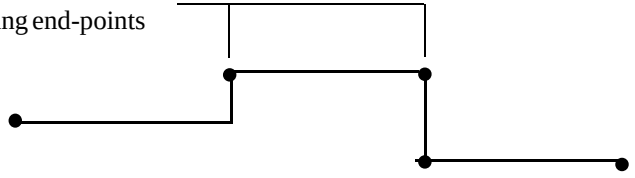
Unlocking an Object

1. Select the object(s) to be *Unlocked*, by clicking on it with the Selection tool.
2. Choose the *Unlock* command from the **Options** menu, or click on the *Unlock* button found on the Menu Bar.

Join and Unjoin

When *Join* is selected from the **Options** menu or the Menu Bar, the program automatically connects all line segments with overlapping end-points. Overlapping means that the end-points are connected by snapping them to each other, (using the *Snap To* tools). The *Join* command combines individual line segments, arcs, polygons, and free-formed splines into one polygon.

Overlapping end-points



The *Join* command turns individual line segments and arcs into polygons. Perform any of the procedures or choose any of the commands to use with open or closed polygons. Joined objects can be resized, reshaped, smoothed or filled.

Join can be selected from the **Options** menu or the Menu Bar. Select objects prior to choosing the *Join* command.

How to Join objects

1. Select the objects to join, by using the Selection tool and holding down SHIFT swhile clicking on the individual objects. (Use the appropriate *Snap To* tool to ensure that the end-points are overlapping).
2. Now, select the *Join* command from the **Options** menu or from the Menu Bar.

Unjoin

When *Unjoin* is selected from the **Options** menu, any selected objects are automatically disconnected from each other. *Unjoin* separates the segments that have been combined into one joined polygon, therefore making them individual line segments. *Unjoin* divides rectangles, rounded rectangles, polygons or circles into individual line segments.

The *Unjoin* command can be selected from the **Options** menu or from the Menu Bar.

How to Unjoin objects

1. Select the object, by using the Selection tool from the Edit toolbox.
2. Then, select the *Unjoin* command from the **Options** menu, or from the Menu Bar.

Bring to Front

The *Bring to Front* command from the **Options** menu, moves any selected object to the front of the stacking order. This command can be used to move an object to the front of the stacking order so that it can be edited. After editing the object move the object to the back of the stacking order. If several objects are selected and then *Bring to Front* is chosen, all selected objects move together in front of the unselected objects. The transposed objects remain in the same stacking order they were prior to the command. Choose the *Bring to Front* command from the **Options** menu, or press CTRL F.

Send to Back

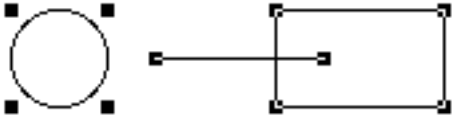
The *Send to Back* command moves any selected object to the last position of the stacking order. An object moved to the back can be partially or totally obscured from view by other objects placed in front. Although obscured, the object remains selected and it can be moved to the front if necessary. The *Send to Back* command can be selected from the **Options** menu, or press CTRL B.

To move an object Front or Back

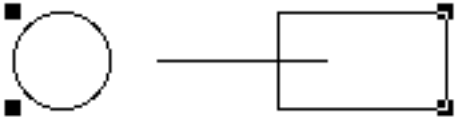
1. Select the object(s). This command also works with grouped objects.
2. Then, select the desired command and the selected object is moved to the *Front* or *Back* depending on which command is selected.

Group and Ungroup

Designs can be made of many different individual objects. A design may need to consolidate, or group, several objects into one object, to make working with a design easier. Once objects are grouped; it enables them to be manipulated as one object.



Ungrouped objects selected



Same objects grouped and selected as a group

Notice in the previous example that the objects have individual boundary points when not grouped. After grouping the objects, they share the same boundary points.

Clicking on a set of grouped objects selects the entire group simultaneously. It is possible when working with grouped objects, to resize, delete or duplicate them. Once a set of objects are grouped, the individual objects that make up the group cannot be changed, unless the objects are first ungrouped.

Objects can only be grouped on the same layer. When objects for grouping are selected, they appear at different positions in the object stacking order. Once these objects are grouped, they take on the stacking order of the foremost grouped object.

Grouping Objects

1. Select the objects to be grouped, by clicking on the individual objects, use the Selection tool and hold down SHIFT. Alternatively a selection rectangle can be drawn around the objects.
2. Choose *Group* from the **Options** menu, or click on the Group button found on the Menu Bar, or press CTRL G. Once the objects are grouped, a display of boundary points are exhibited around the grouped objects.

Ungrouping Objects

To ungroup any grouped objects, to make any changes to the individual objects within the group. Ungrouping of an object breaks that object into the individual objects again. Locked objects cannot be ungrouped. Unlock the objects first, using the *Unlock* command.

1. Select the grouped objects by clicking on them with the Selection tool.
2. Choose *Ungroup* from the **Options** menu, or click on the Ungroup button found on the Menu Bar.

Once grouped objects are Ungrouped, changes can be made to each individual object. **KeyCAD Pro** automatically highlights the selected objects.

Move Origin

The *Move Origin* command is used to move the *Absolute* point of the design. The *Absolute* point is the point from which all absolute coordinates are calculated from. The Absolute coordinates are considered the X and Y coordinates or the horizontal and vertical distance from the zero point. The *Origin* point is where the horizontal and vertical axis lines intersect.

The *Origin* point can be moved to any location on the current document. Moving the *Origin* point affects all layers.

The *Move Origin* command can be accessed from the **Options** menu.

Actual Size

Use the *Actual Size* command to return the current document back to the 100% view. *Actual Size* means the measurement on the screen is the size the document is to be printed.

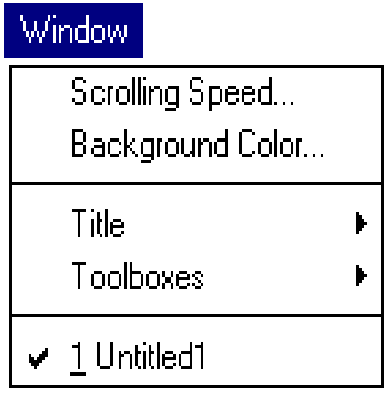
Actual Size is useful for removing any *Zoom In* or *Out* commands that may have been performed. The design returns to 100% viewing.

The *Actual Size* command can be accessed from the **Options** menu, or press CTRL E.

The Windows Menu

The **Windows** menu provides access to various options that affect the characteristics of the drawing window setup. The options are: *Scrolling Speed*, *Background Color*, *Title Options* and *Toolboxes*. The list of currently open files is drawn at the bottom of the menu, selecting a filename makes that document active.

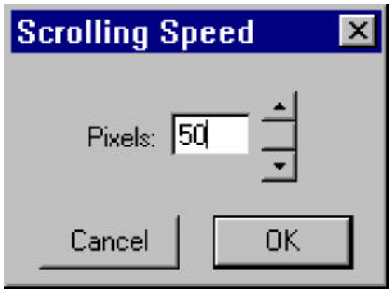
The following diagram shows how the menu appears within the program.



The Windows Menu

Scrolling Speed

The *Scrolling Speed Dialog* box is used to change the speed at which the document scrolls when clicking on the Scroll Arrows. The *Scrolling Speed* is determined by the number of pixels the document moves before it redraws the screen. The Pixel setting can be changed in two different ways; either click on the small arrows next to the numeric display, or highlight the numeric value and type in the new setting. The program defaults to 50 pixels. The higher the number the faster the document scrolls.

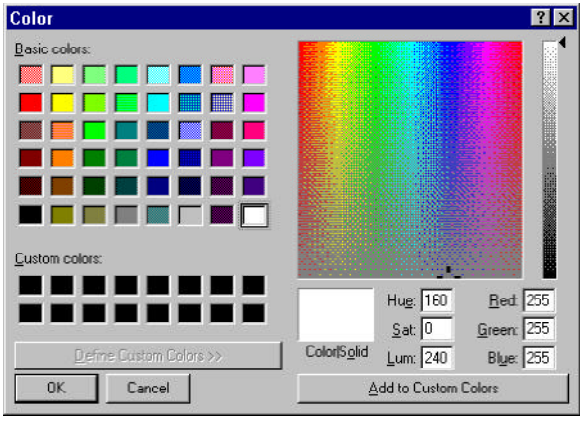


The Scrolling Speed Dialog box

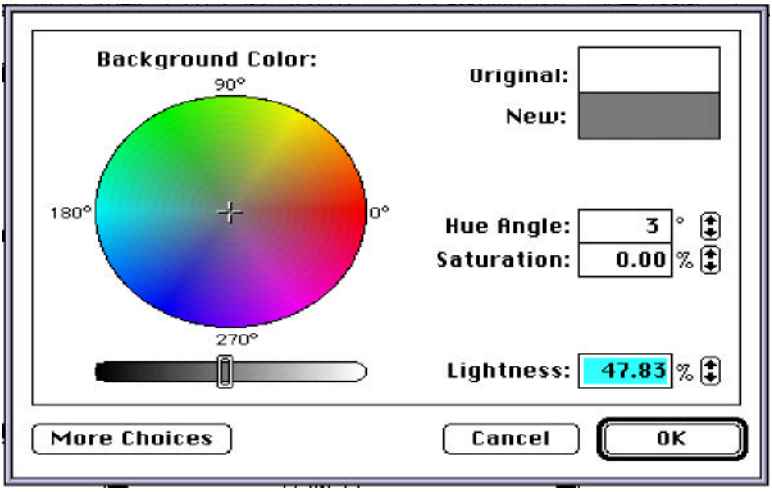
Once the desired speed has been entered, click on the *OK* button and the new settings become effective. Click on the *Cancel* button to cancel any changes made.

Background Color

The *Background Color* option is used to change the background color of the document. If there are any objects that have no fill to them, the background color appears through the objects. The background color covers the entire page size of the current document.



The Color Dialog Box for Windows



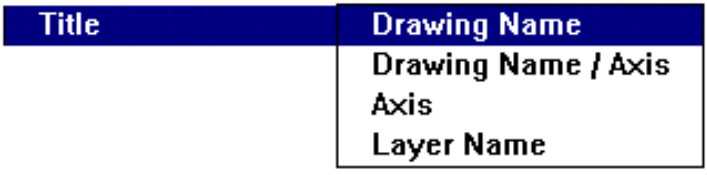
The Color Dialog Box
For Macintosh

When the *Background Color* is selected from the **Windows** menu, a Color Picker Dialog box appears. Choose from any color available to the current video hardware. **KeyCAD Pro** supports up to 16.8 million colors.

The Red Green Blue values of any color can be changed, as well as the *Hue, Saturation*, or *Brightness* of the selected color. Once the color is set, click on the *OK* button, or click on the *Cancel* button to return to the previous background color.

Title Options

KeyCAD Pro offers several different options for titling documents. These options consist of the *Drawing Name, Drawing Name / Axis, Axis* and *Layer Name*.

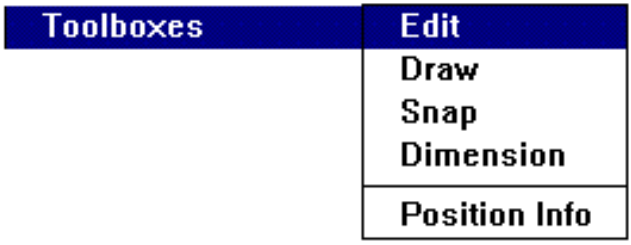


The Title Options Pop-out Menu

Select the *Title Options* by accessing the *Title* pop-out menu under the **Options** menu. The document title can be changed at anytime during the design process. The selected title appears in the middle of the Title Bar at the top of the screen. If the *Layer Name* option is chosen, only the active layer name appears.

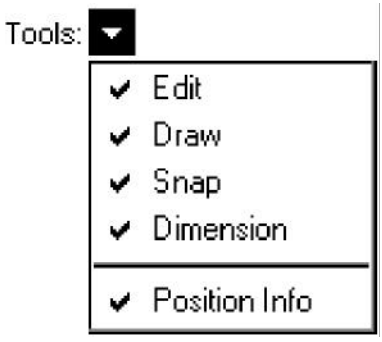
Toolboxes

The toolbox pop-out menu turns On or Off any of the toolboxes, as well as the Readout display at the bottom of the screen. There is access to the Edit toolbox, Draw toolbox, Snap To toolbox, Dimension toolbox and Position Info (readouts). A check mark beside the name indicates that the toolbox is active.



The Toolbox Pop-up Menu

The *Toolboxes* can also be accessed from the Menu Bar. This pull-down menu is designed to turn On and Off the tools. The Menu Bar can be turned on by pressing CTRL M or by pulling down the **Layout** menu.



The Tools Pull-down Menu

The Layers Menu

The **Layers** menu is where the user can control the various layers. **KeyCAD Pro** allows the user to add information to a document in separate layers, which act as transparent overlays. Layers are used to hold or display various aspects of a design. A single layer can contain and display different aspects of a design, **KeyCAD Pro** supports up to 256 layers.

Creating a Layer

To create a new layer in **KeyCAD**, follow the steps below:

1. Choose **Layer Info...** from the **Layers** pull-down menu.
2. Type a new Layer Name in the Layer Name Dialog box.
3. Click on **New**, **OK**.

When the *Layers* option from the **Layers** menu is selected, the new layer name will appear in the drop down box. Click on the newly created layer to make it the currently active layer.

Hiding a Layer

To hide a layer, follow the steps below:

1. Go to the *Layers* option in the **Layers** menu to see the currently activelayer, which is denoted by a check mark. **The currently active layer cannot be hidden**.
2. Now go to Layer Info.. option and click on the layer that you wish to hide.
3. Click on the **Hide** button then on **OK**.

All portions of a drawing within a hidden layer are not dsiplayed on the screen.

Showing a Layer

To display a previously hidden layer, follow the steps below:

1. Choose the **Layer Info...** option from the **Layers** menu.
2. Click on the layer to be displayed.
3. Click on **Show** then **OK**.

All portions of the drawing which were previously **Hidden** will now **Show** on the screen.

An example of the use of Layers would be to draw a floorplan on one layer, adding the windows and doors of the floorplan to a different layers and the dimensions of the floorplan to another layer. Each layer can be separately named for easy access.

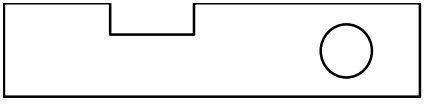
The Layers and Layer Names Pop-up Menus

The Layers Information Dialog box

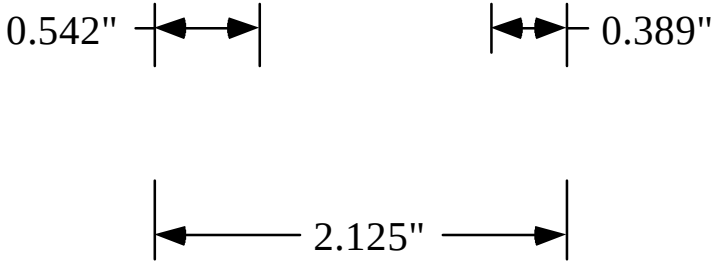
Example

The example below explains how the user can have different information on separate layers. Notice that the separate layers information still pertains to one design and not separate designs.

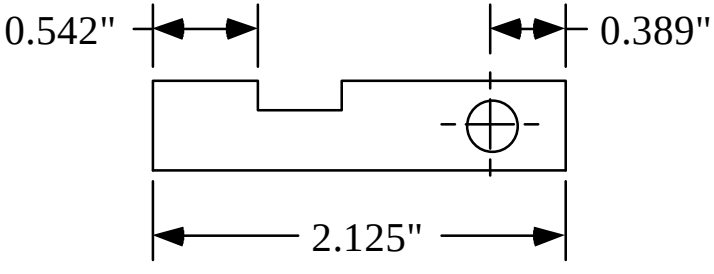
Layer 1: Contains the original design created.



Layer 2: Contains the dimensions of the design on Layer 1, while Layer 1 remains hidden.



Layers 1 and 2: The diagram below shows both layers 1 and 2.



Layer Names Pop-up Menu

Under the **Layers** menu there is access to the Layer Names pop-out menu. This menu displays the names of the layers. The program defaults to one untitled layer. A check mark beside the layer name indicates that the layer is active.

Layers Name Pop-up Menu

Layers Dialog box

Working with the layers provides the option of naming the layer, showing individual layers, showing all layers, hiding layers, hiding all layers, adding new layers, renaming layers, or removing layers. The Dialog lists the names of the layers to help choose the layer to assign any attributes to. A dot beside the layer name indicates that the layer is visible.

The Layers Information Dialog box

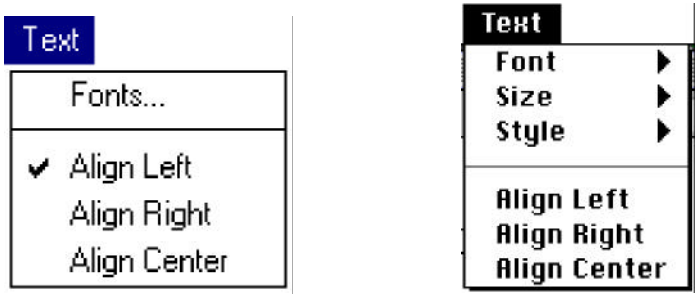
The Text Menu

The **Text** Menu allows the user to edit words, sentences, and paragraphs that are created using the Text tool (found in the Edit toolbox). **KeyCAD Pro** provides the ability to label charts, to display measurements and to give titles to objects and designs. This chapter shows how to change the font, the style, the size or the alignment of a text block in order to place text anywhere in a design.

Four corner points represent the borders of a text block. When a text block first appears on the screen, the default size is about 1.5 inches by 1.5 inches. The size of the text block can be altered by using the Selection tool.

The appearance of text is governed by the selections in the **Text** Menu. To change the appearance, select the text to be changed using the Text tool, then select one of the **Font** and **Text** Menu options. Only the selected or highlighted text is affected by the change.

The initial font style that **KeyCAD Pro** is set to is Arial 12 point. The font style is easily changed using the Text tool and the **Text** menu.



Windows **Macintosh**

The Text Menu

Font

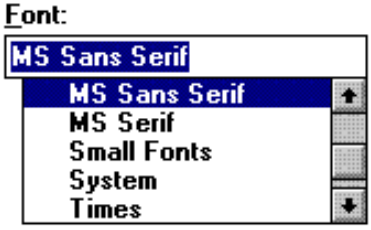
Fonts are different types of lettering styles that are installed within the System. For adding and removing fonts, refer to the Windows User Manual.

How to change Fonts

1. Highlight the text to be changed with the Text tool by clicking the mouse button and dragging the highlight bar across to the end of the desired text. (To highlight individual words, double click on them.) The highlighted area is blackened.
2. Pull down the **Text** Menu at the top of the screen and choose *Fonts*.... Select the desired font, style and size, by clicking on the options within the Font Dialog.
   On the Macintosh version of **KeyCAD Pro** , pull down the **Text** Menu and select **Font**. This will allow you to select the desired font.

Unlike most programs of this nature, **KeyCAD Pro** remembers the last selected font. This means that after any font is changed, all subsequent text is written with or uses the same font style until it is changed again. For example, if some text was changed to Courier with the Text tool and then an object is created, when the Text tool is used again, the Courier font is still chosen.

The Font menu displays a list of all fonts available in the current system. To add and remove fonts from the System refer to the Windows User Manual. **KeyCAD Pro** supports PostScript and TrueType fonts.



The Fonts Menu

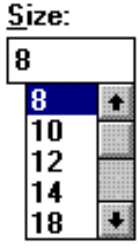
Size

The size of text is regulated by the **Size** function in the **Font** Dialog. The initial or default point size of text is **12**. The size can be raised or lowered depending on the requirements.

How to change Font Size

1. Highlight the desired text by clicking the mouse and dragging it across the words to be changed. (Individual words can be highlighted by double clicking on them.) The highlighted text is blackened.
2. Pull down the **Text** Menu at the top of the screen and choose the Font Dialog. Select the desired font, style and size, by clicking on the options within the Font Dialog.
   On the Macintosh version of **KeyCAD Pro** , pull down the **Text** Menu and select **Size**. This will allow you to select from the font size options for the font selected.

Under the **Size** menu only certain point sizes are shown. The numbers shown are the recommended sizes to receive the best quality text. **KeyCAD Pro** supports other point sizes, however, the text quality may be inferior. Font sizes are determined for each individual font, when the actual font is installed within the System.



The Size Menu

KeyCAD Pro allows point sizes that are larger than 18 points. For larger point sizes highlight the point size above the scroll bar selections. Type in a whole number for the point size and press e or click on the OK button.

Style

Font styles are useful for adding a special touch to text within any document. **KeyCAD Pro** supports several styles such as Regular, **Bold**, ***Bold Italic***, and *Italic*. Additional styles are available depending on the font selected. Any of the styles can be combined with any Font or Font Size. The **KeyCAD Pro** default Style is Regular text. The style can be changed at any time.

How to change Font Style

1. Highlight the text with the Text tool by clicking the mouse and dragging it across the desired text. (Double clicking on words highlights them.) When highlighted, the selected text is blackened.
2. Pull down the **Text** Menu at the top of the screen and choose the Font Dialog. Select the desired font style, by clicking on the option within the Font Style menu. Multiple font styles can be applied to one text block.
   For the Macintosh version of **KeyCAD Pro**, pull down the **Text** Menu and select **Style**. This will allow you to select a style to apply to the selected text.

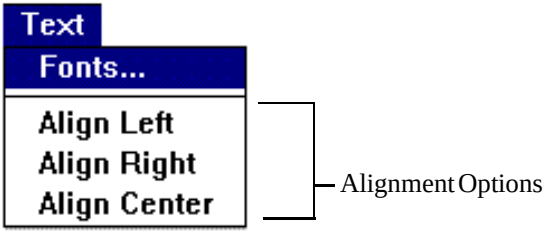
The Style Menu

Align

The last three options under the **Text** menu are *Align Left, Align Right*, and *Align Center*. These three options are used to justify the text inside a text block. This option makes it easier to place text on designs. The default mode is Align Left. The mode can be changed in order to create right or center aligned text, as in headlines or titles.

How to Change Text Alignment

1. Highlight the text using the Text tool by clicking at the beginning of the block and dragging across to the end of the text. (Individual words are highlighted by double clicking on them.) Highlighted text is blackened.
2. Pull down the **Text** menu at the top of the screen and choose the appropriate **Align** options.



The Text Menu

Below are some examples of the alignments and how they appear within a text block.

Text Aligned Left

Text Aligned Right

Text Aligned Center

Font sizes, styles, and align modes can be changed prior to typing the text. Make the desired choices from the **Text** menu at the top of the screen before typing the titles, paragraphs, or captions in the designs.

NOTE: Placing in excess of forty text boxes on the screen may result in distortion of subsequently placed text boxes on the screen.
