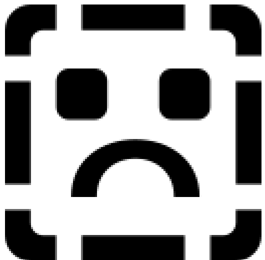
MiniCad Commands

MiniCad Palettes



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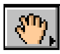
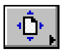
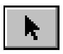
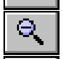
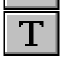
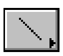
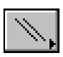
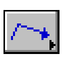
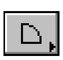
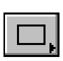
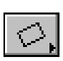
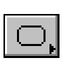
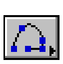
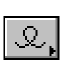
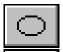
The tools on the **2D Tool** palette are used to create 2D objects and to perform editing operations upon them. The palette also contains tools for manipulating the drawing within the viewing area.

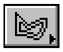
The **2D Tool** palette is opened using the **Tools** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette. It contains the following tools:

Tool	Description
	The **Pan** tool is used to move the view of the active drawing.
	The **Move Page** tool is used to move the drawing around the drawing page. This tool is on the **Pan** tool's pop-out menu.
	The **Selection** tool is used to select a 2D object in the drawing window.
	The **Zoom In** tool magnifies the view of the drawing.
	The **Zoom Out** tool reduces the view of the drawing.
	The **Text** tool is used to place text anywhere on a drawing.
	The **Line** tool is used to create constrained and unconstrained single lines.
	The **Double Line** tool is used to create constrained and unconstrained double lines. This tool is on the **Line** tool's pop-out menu.
	The **Leader Line** tool is used to create a 2-segment, leader line. This tool is on the **Line** tool's pop-out menu.
	The **Arc** tool draws circular arcs of any degree, at any angle.
	The **Quarter Arc** tool creates arcs that are one quarter of an ellipse, with an angle limited to 90°. This tool is on the **Arc** tool's pop-out menu.
	The **Rectangle** tool is used to create a rectangle or a square.
	The **Rotated Rectangle** tool is used to create a rectangle at any angle. This tool is on the **Rectangle** tool's pop-out menu.
	The **Rounded Rectangle** tool is used to create rectangles with rounded corners. This tool is on the **Rectangle** tool's pop-out menu.
	The **Polyline** tool is used to create a connected series of arcs, curves and lines that are treated as a single entity.
	The **Freehand Line** tool produces a polygon along a path that is specified by dragging the mouse. This tool is on the **Polyline** tool's pop-out menu.

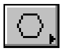


The **Ellipse** tool is used to draw circles and ellipses.

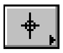
The **2D Polygon** tool is used to create polygonal shapes.



The **Double Line Polygon** tool is used to create double line polygons. This tool is on the **Polygon** tool's pop-out menu.



The **Regular Polygon** tool is used to create a closed polygon whose sides are equal in length. This tool is on the **Polygon** tool's pop-out menu.



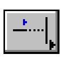
The **Insert Symbol** tool is used to place symbols in the drawing in 2D or Plan projection.

The **Symbol Paint** tool is used to place multiple copies of the active symbol in the drawing by dragging the mouse. This tool is on the **Insert Symbol** tool's pop-out menu.

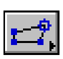
The **2D Locus** tool is used to place a 2D reference point (represented by a diagonal cross) on the drawing. This tool is on the **Insert Symbol** tool's pop-out menu.

The **Trim** tool is used to remove a portion of an object using another object as a cutting tool.

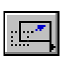
The **Offset** tool is used to create a new object whose outline is at a constant distance from the selected object. This tool is on the **Trim** tool's pop-out menu.



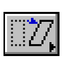
The **Extend** tool lengthens or "extends" one object to meet another object. This tool is on the **Trim** tool's pop-out menu.



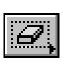
The **Reshape** tool is used to edit polygons and polylines, including freehand lines.



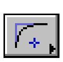
The **Resize** tool is used to scale an object. This tool is on the **Reshape** tool's pop-out menu.



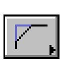
The **Shear** tool skews all of the vertices of a rectangle, polygon or ellipse. This tool is on the **Reshape** tool's pop-out menu.



The **Clip** tool is used to cut objects. It can cut lines, arcs, rectangles, ellipses, polygons, and polylines. This tool is on the **Reshape** tool's pop-out menu.

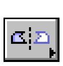


The **Fillet** tool is used to places an arc (fillet) between two objects, making the arc tangent to each object at the fillet's end points.

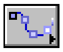
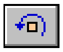
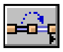
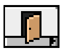
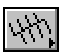
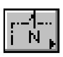
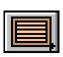
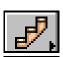
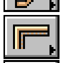
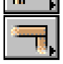
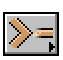


The **Chamfer** tool is used to place a line between two linear objects, at a specified distance from the intersection of the lines or objects. This tool is on the **Fillet** tool's pop-out menu.

The **Eyedropper** tool is used to "pick up" attributes from one object and apply them to another.

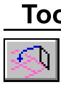


The **Mirror** tool is used to make a mirror image of a

	selected object.
	The **Duplicate Along Path** tool is used to replicate an object (or group of objects) at regular intervals along a path. This tool is on the **Mirror** tool's pop-out menu.
	The **Rotate 2D** tool is used to rotate a selected object.
	The **Duplicate Symbol in Wall** tool is used to replicate the active symbol (typically a door or window) at a specified interval in a wall.
	The **Fancy Door** tool is used to create a door in a front elevation. The door can have raised panels, rectangular windows, and a semicircular top window.
	The **Number Stamp** tool is used to place a marker with an identifying number on the drawing. The number automatically increments each time the tool is used.
	The **Parking Space** tool is used to create evenly-spaced lines for parking lots.
	The **Property Bounds** tool is used to create a polygon based on bearing and distance values. The boundary line can be made of line segments or curves.
	The **Revision Cloud** tool on the on the **2D Tool** palette is used to create a revision cloud.
	The **Shutters** tool is used to create window shutters in front elevation.
	The **Straight Stair** tool is used to create stairs.
	The **Wall** tool is used to create walls.
	The **Butt-Join Wall** tool is used to join walls with corners butted together instead of mitered.
	The **Y-Join Wall** tool is used to join multiple walls that come together at odd angles and cannot be joined using the other wall-joining tools.

The tools on the **3D Tool** palette are used to create 3D objects and to perform editing operations upon them. The palette also contains tools for manipulating the 3D drawing within the viewing area.

The **3D Tool** palette is opened using the **Tools** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette. It contains the following tools:

Tool	Description
	The **Align Plane** tool allows 3D objects to be aligned to the working plane in both rotation and translation.
	The **Flyover** tool is used to move the view of the drawing by dragging the mouse. It changes the angle

of view, while constraining the viewer to an upright position.

The **3D Locus** tool is used to place a 3D reference point (represented by a triad) on the drawing.

The **3D Mirror** tool is used to make a mirror image of a selected object in 3D space.

The **Extruded Polygon** tool is used to extrude objects in 3D without drawing them in 2D first.

The **3D Polygon** tool is used to create 3D polygons without drawing them in 2D first.

The **Extruded Rectangle** tool creates 3D rectangles without drawing them in 2D first.

The **3D Reshape** tool is used to edit extrudes, walls, roofs and 3D polygons.

The **Rotate 3D** tool is used to rotate a selected object in 3D space.

The **Rotate View** tool is used to change a model's angle of view.

The **3D Selection** tool is used to select objects in 3D orthogonal views.

The **Set Working Plane** tool is used to set the working plane based on either a currently extant object's plane or three points in space specified by the user.

The **Insert 3D Symbol** tool is used to insert symbols in 3D orthogonal projections.

The **Translate View** tool moves the 3D projection around the page.

The **Walkthrough** tool is used to visualize an architectural model.

The **Zoom In** tool magnifies the view of the drawing.

The **Zoom Out** tool reduces the view of the drawing.

Any object with a surface can be added to another object using the **Add Surface** command on the **Tool** menu.

The result of an **Add Surface** operation is a single polygon that is a combination of two or more objects.

Any combination of the following objects can be added together:

- Rectangles
- Rounded rectangles
- Ellipses/circles
- Quarter arcs
- Full arcs
- All types of polygons (regular, freehand and polyline)

Any 2D filled object that is not locked, is not in a group or is not a symbol, can be added to another object. More than two objects can be added together simultaneously.

When surfaces are added, the object in front is added to the object in back. If the objects have fill patterns, the new object has the fill pattern of the rearmost object.

If the objects being added are open, they are converted to closed polygons then added together.

For related information, see the following topics:

Combine into Surface

The **Align Layer Views** command on the **Organize** menu changes the view and projection of all layers in a drawing to match the view and the projection of the active layer.

To use this command, place the active layer in the desired view and projection. Then choose the **Align Layer Views** command.

For related information, see the following topics:

Projection

The **Align to Grid** command on the **Tool** menu moves objects that were created with the reference grid disabled, or were moved off the grid, back into alignment with the grid. This tool can also be used to realign objects to the grid after the grid is changed.

The upper left corner of each object is aligned to the grid. The grid is controlled with the **Set Grid** command on the **Page** menu.

To align objects to the reference grid, select the object you want to align to the grid and choose the **Align to Grid** command on the **Tool** menu.

The **Align Objects** command on the **Tool** menu is used to align selected objects to each other. When the **Align Objects** command is chosen, the **Align Objects** dialog box opens.

If only 2D objects are selected, alignment is restricted to the *x* and *y* axes. If 3D objects are selected, the *z* axis can also be used for alignment.

The dialog box is divided by row. The top buttons are used to select the alignment of the *x* axes, and the next row, the *y* axes. If the objects selected are 3D, the bottom row aligns the *z* axes. Any combination of alignments of the *x, y,* and *z* axes is possible.

When aligning objects in 3D, the current view determines the direction of the alignments.

For related information, see the following topics:

Align Layer Views

Align to Grid

The **Attributes** command on the **Window** menu is used to open the **Attribute** palette. The **Attribute** palette controls attributes such as pen color, pen pattern, marker style and line weight.

For additional information, see the following topics:

Attribute Palette

The **Cam Construction** command on the **Mechanical** menu draws a flat plate cam using the parameters specified in the **Cam Construction** dialog box.

Note: This command is only available when the **Mechanical Overlay** is active. If the **Mechanical** menu is not listed on the menu bar, select the **Mechanical Overlay** using the **Overlays** command on the **File** menu.

The cam is specified by the radius of the prime, or base circle (P.C.R.), maximum follower displacement (Max), and radius of the follower (Size). The cam profile is described by constructing a follower displacement curve.

To construct the curve, click the mouse in the area of the diagram to be described. A black bar highlights the ordinate of the portion of the diagram selected. For the initial point, click anywhere in the diagram.

Next, specify the type of follower motion for that portion of the diagram. The types of motions allowed are:

- Uniform
- Harmonic
- Parabolic
- Cycloidal
- Trapezoidal
- Modified Sine

If a portion of the diagram is a dwell, the type of motion has no effect.

Click the **New** button and specify the angle and displacement for the selected portion of the diagram. The angle specified is the angle from 0° (0° being vertical).

The displacement is measured from the prime circle radius. If the displacement for any portion of the diagram is the same as the previous section, a dwell is assumed.

A few seconds after the data is entered, if no further action is taken, the displacement curve for the selected area is drawn. The displacement curve also is drawn if the mouse is clicked anywhere in the diagram. Successive points of the displacement curve are similarly described until the diagram is complete.

After the diagram is complete, any point of the curve can be changed by clicking on the vertical line at the point and changing the angle or displacement. The remainder of the diagram is automatically changed to reflect the modification.

Any point can be deleted by clicking on the vertical line at the point and clicking the **Delete** button. If the type of motion is to be changed, click anywhere in the portion of the diagram to be changed, and re-specify the type of motion using the pop-up menu.

After the diagram is complete, the cam is drawn by clicking the **OK** button or pressing **Enter**.

A few important things to remember when using the **Cam Construction** command are:

- The displacement curve and prime, or base, circle radius represents the motion of the follower.
- Be sure to click in the area of the displacement diagram in which you want a new point to be positioned before clicking the **New** button.
- To change an existing point, click on the vertical line at the point and type in the new angle or displacement.
- To delete a point, click on the vertical line at the point and click the **Delete** button.

The **Classes** command on the **Organize** menu is used to create a "class."

The use of classes is a powerful way to organize the elements of your drawing. Classes are similar to layers in many ways, but they can work across layers, which often makes them more useful.

For example, you can put objects, like lighting fixtures, into a class called "Fixtures," and the fixtures can be used on many layers in the drawing. You can make the lighting fixtures in the drawing visible or invisible by changing the visibility of the Fixtures class.

Classes also allow intricate detail to be included with objects or symbols. The intricate details can be placed in a class. The class can be made invisible, reducing clutter on the drawing and significantly improving speed of screen redraws. The class can be made visible when details are required. You can create multiple detail classes to display increasing levels of detail.

Class information can be used with worksheets, and can be used as search criteria for the database. Class information can be returned into formulas, where the data can be calculated and manipulated.

All drawings have a minimum of two classes, **Dimension** and **None**, with **None** being the default class. These classes can be renamed, but they cannot be deleted.

Additional classes can be created or deleted. If a class is deleted, the objects in the class are assigned the default class, **None**. If you import a symbol from another drawing, any classes that have been assigned to the symbol are imported too.

A single drawing can contain up to 256 classes (including **Dimension** and **None**).

Objects are always drawn in the active class, with the exception of dimensions. All dimensions are placed in the **Dimension** class regardless of the active class.

For additional information, see the following topics:

Creating Classes
Assigning a Class
Worksheets

Classes are created with the **Classes** command on the **Organize** menu which opens the **Classes** dialog box.

This dialog box has the following options:

Normal

Selecting **Normal** makes the class fully visible. Objects in a normal class can remain invisible if the object is "owned" by another object that is in an invisible class.

Invisible

Selecting **Invisible** makes the class invisible. Objects that are "owned" by an invisible-class object are hidden, even if the owned object itself is in a class that is visible.

Rename

Allows you to rename an existing class. Renaming a class has no effect on objects in that class, or on their visibility.

New

Creates a new class in your drawing. Each class in a drawing must have a unique name that can be no longer than 20 characters.

Remove

Removes the selected class from the drawing. A dialog box warns that this permanently removes the class from the drawing. Any objects in the deleted class are put in the default class, **None**.

For related information, see the following topics:

Assigning a Class

Classes

Object Info Palette

All objects are drawn in the active class. The name of the active class is always displayed in the mode bar, in the header of the **Class** pop-up menu.

The active class is always visible, regardless of what the visibility setting for that class is. This allows you to display (or print) an invisible class by making it the active class.

The active class is changed by choosing a different class in the **Object Info Palette**. You can also change classes by selecting it from the **Class** menu on the **Data Display** bar while no objects are selected.

After an object is drawn, its class can be changed by selecting it and choosing a different class name in the **Object Info** palette or in the **Class** menu on the **Data Display** bar.

For related information, see the following topics:

Creating Classes

Classes

Object Info Palette

The **Clear** command on the **Edit** menu removes the selected object(s) from the drawing.

The **Clear** command has the same effect as pressing the **Delete** key while an object is selected.

Unlike the **Cut** command, **Clear** does not place the deleted objects on the Clipboard. They can only be retrieved with the **Undo** command.

For related information, see the following topics:

Cut
Undo

Clipping a surface takes any number of objects and creates a new polygon by removing overlapping areas of the two surfaces. Clipping is done with the **Clip Surface** command on the **Tool** menu.

When two objects are clipped, the object in front remains on the drawing, and the area beneath it is removed. Depending on the objects used, the clipped object may be converted into a different object type. For instance, if a hole is cut in a rectangle, the rectangle is converted to a polyline figure.

Any combination of the following objects can be clipped:

- Rectangles
- Rounded rectangles
- Ellipses/circles
- Quarter arcs
- Full arcs
- All types of polygons (regular, freehand, and polylines).

Other things to remember about clipping:

- Any object that can be filled and that isn't in a group can be clipped.
- You cannot clip with a symbol.
- If open objects are clipped, they are converted to closed polygons first.

To clip objects, arrange the objects in the proper order (if necessary, use the **Send** command to place the objects in the proper order). The **Clip Surface** command takes the topmost object and uses it to clip the selected objects below it. Once arranged, select all objects and choose **Clip Surface** on the **Tool** menu.

For related information, see the following topics:

Clip
Intersect Surface

The **Close** command on the **File** menu closes the active document.

The **Close** command has the same effect as clicking in the close box in the drawing window.

If there are any unsaved changes in the document, you will be prompted to save them before closing the document.

For related information, see the following topics:

Open

The **Combine into Surface** command on the **Tool** menu, creates a new object that is the combination of objects from which it is created. Any combination of objects can be used as long as they form a closed curve (their perimeter must be unbroken).

This tool produces a brand new object as a result. The original objects are not changed in any way.

To combine surfaces:

1. Select both objects.
2. Choose **Combine into Surface** on the **Tool** menu.

   The **Paint Bucket** cursor appears.
3. Place the **Paint Bucket** cursor inside the area you want to combine and click.
4. A polygon is created with the current fill pattern.

Troubleshooting tip! If you have objects that won't combine with this tool, the selected objects are probably not closed (a single pixel gap can prevent the new surface from being formed). Use the **Object Info Palette** to check the polygons and polylines in your selection. Click the **Close Object** button on each. If the command still fails, zoom-in and carefully examine the objects for a gap.

For related information, see the following topics:

Add Surface

The **Constraint** palette opens when the **Constraints** command is chosen from the **Window** menu.

The **Constraint Palette** displays 2D constraints with 2D tools, and 3D constraints with 3D tools.

Like all palettes, the **Constraint** palette can be moved around the screen by dragging its title bar. The palette can be closed by clicking its close box.

Most objects, both 2D and 3D, can be converted into individual lines with the **Convert to Lines** command on the **Tool** menu. For example, if you convert a rectangle to lines, it becomes four separate lines.

When converting circles, the accuracy of the conversion depends on the "conversion resolution," which is set with the **Preferences** command. The conversion resolution specifies the number of line segments per 360° of arc.

The sides of rounded rectangles are converted to lines and the corners become one quarter of the conversion resolution.

Arcs with fill patterns are converted with radius lines.

Polygons have one line per segment (i.e., one line less then the total number of vertices).

To convert an object to lines, select the object and choose **Convert to Lines** on the **Tool** menu. The object converts to individual lines. If you want to keep the original object intact, and convert a copy of the object, use the **Convert Copy To Lines** command.

If the selected object is a 3D object, a dialog box opens with two options:

- **Convert Object To Lines** produces a 2D object made of individual lines, displaying all lines in the original 3D object. The view of the object is taken from the current projection. The 3D object appears in wireframe with all lines visible.
- **Hidden Line Rendering** produces a hidden line rendering, taken from the current projection. A hidden line rendering removes all objects obscured by the top-most objects. The resulting objects are 2D, in a 3D view, made of lines with no surface. Therefore, the objects can have no color or shading.

For related information, see the following topics:

Convert Copy To Lines
Convert to Polygons
Convert to Mesh

Convert Copy to Lines works like the **Convert to Lines** command except that it creates a copy of the selected object. The original object is unchanged.

Note: The **Convert Copy to Lines** command is located on the alternate **Tool** menu. This menu is activated by, pressing the **Control** key while selecting the **Tool** menu from the menu bar.

When the **Convert Copy to Lines** command is chosen, a copy of the selected object, converted to lines, is placed on top of the original. If the selected objects are 3D, copies of the hidden line rendering are left selected on top of the original.

For additional information, see the following topics:

Convert to Lines

The **Convert to Mesh** command on the **3D** menu is used to convert a 3D polygon or extrude to a mesh object.

When a 3D object is converted to a mesh, each individual vertex of the mesh object can be edited. Multiple vertices can be selected and edited together.

A 3D object that has been converted to mesh can be converted to a wireframe again. To convert the mesh object back to wireframe, select all of its vertices and group the selection.

After converting to mesh, the object can be edited with the **Object Info Palette**. In addition, the vertices can be edited with the 2D tools. Any vertex, or multiple selection of vertices, can be moved with the **Move** command on the **Tool** menu, or dragged to a new location with the mouse.

The individual polygons that make up the mesh can be edited. Because the polygons are grouped objects, the group must be entered with the **Edit Group** command before it can be edited.

For related information, see the following topics:

Convert to Lines
Convert to Polygons

Any object that has a surface can be converted into a polygon with the **Convert to Polygon** command on the **Tool** menu.

When converting circles, the accuracy of the conversion depends on the "conversion resolution," which is set with the **Preferences** command. The conversion resolution specifies the number of line segments per 360° of arc.

The corners of rounded rectangles are one quarter of the conversion resolution.

Arcs with fill patterns are converted with radius lines.

To convert an object to a polygon, select the object and choose **Convert to Polygons** on the **Tool** menu. The object converts to 2d polygons. If you want to keep the original object intact, and convert a copy of the object, use the **Convert Copy To Polygons** command.

If the object being converted is 3D, the **Convert to Polygons** command opens a dialog box with the following options:

- **Convert To Polygons** converts the 3D object to 2D polygons. Each polygon can be individually selected and edited, deleted, or moved. However, in this mode polygons may not be stacked correctly--polygons that were beneath others in the 3D object may be on top after conversion. The **Send** command on the **Tool** menu can be used to reorder them.
- **Render to Remove Hidden Surfaces** renders the object into individual lines and 2D polygons. The polygons are sorted so that objects in the foreground obscure objects in the background. The result is a hidden surface rendering of the original 3D object. After conversion, the individual lines or polygons that make up the rendering can be selected, individually edited, deleted or moved. 2D polygons can have color or fill. This option takes somewhat longer than a **Convert to Polygons**.

For related information, see the following topics:

Convert Copy To Polygons
Convert to 3D Polygons
Convert to Mesh
Convert to Lines

Convert Copy to Polygons works like the **Convert to Polygons** command except that it creates a copy of the selected object, and converts the copy to polygons. The original object is unchanged.

Note: The **Convert Copy to Polygons** command is located on the alternate **Tool** menu. This menu is activated by, pressing the **Control** key while selecting the **Tool** menu from the menu bar.

When this command is chosen, a copy of the selected object converted to polygons is placed on top of the original. If the object is 3D, a dialog box offers the option of converting a copy or rendering a copy of the 3D polygon.

For additional information, see the following topics:

Convert to Polygons

Any selected 2D line or surface object, including polylines, polygons, ellipses or rectangles, can be converted into a 3D polygon with the **Convert to 3D Polygons** command on the **3D** menu. After conversion, the new 3D polygon can be rotated and manipulated with the 3D tools and commands.

A 3D polygon has a *z* dimension but no thickness. Thus, the object has a location in 3D space and projects appropriately in a 3D projection.

Surface items are shown in wireframe after being converted into 3D. They show a surface only when rendered.

Multiple selections can be converted to 3D polygons.

Groups of 2D objects cannot be converted to 3D polygons. Grouped items must be ungrouped, or the group must be entered with the **Edit Group** command, before conversion.

For related information, see the following topics:

Convert to Polygons
Convert to Mesh
Convert to Lines

The **Copy** command on the **Edit** menu copies selected objects or text to the **Clipboard**. The original objects remain on the drawing.

If the **Show/Snap/Modify** option is set with the **Layer Options** command, you can select multiple objects on different layers and copy them to the Clipboard. When pasted, all objects are placed on the active layer.

If objects are on the Clipboard, be careful not to inadvertently switch applications. MiniCad is a high resolution program. The resolution is maintained with objects copied to the Clipboard as long as MiniCad remains the active application. If the application is switched, the contents of the Clipboard may be converted to 72 dpi.

MiniCad objects pasted into other applications from the Clipboard will lose detail. You can use the **Export** command to bring high resolution objects into other applications without any loss of detail.

For related information, see the following topics:

Cut
Paste
Duplicate
Duplicate Array
Duplicate Along Path

Layers created with MiniCad are independent entities; that is, they can have their own z and Δz, scale, projection, and render status. As such, they can be thought of as "slices" of a single drawing--separate, independent pieces of a drawing.

The layers can be linked together and viewed as a group with the **Create Layer Link** command on the **3D** menu. The layer link is created on the active layer.

Because layering is spatial in nature, you must assemble the pieces of the drawing by manipulating the z and Δz of the individual layers (a process that can be done automatically when you create the layers).

If you want to rotate these pieces together in precise alignment, they must be linked. For example, if you create a building's walls on one layer and a proposed roof on another, you have to link the layers in order to rotate the assembled model.

A linked layer maintains more than registration with the layers to which it is linked. Any changes made to the original layers are updated on the linked layer with every screen redraw. The dynamic link between layers occurs only in one direction--from the linked layer to the layer to which it is linked. This way, you can add detail to a linked model without any unwanted changes to the original.

To create a layer link, make the layer that displays the multiple layers active. In many cases it is desirable to create separate "model" layers to display layer links.

When the proper layer is active, choose the **Create Layer Link** command on the **3D** menu. A dialog box opens with all layers except the active layer displayed in a scrolling list. Select the layers to which you want to link. You can **Shift-Drag** or **Shift-Click** to make multiple selections.

The dialog box has the following options:

Link

Creates the model from the layers selected. Double-clicking the selected layers in the scrolling list has the same effect as clicking the **Link** button.

Project 2D Objects

When selected, 2D objects on the layer, such as dimensions, are projected on the linked layer.

Cancel

Returns you to the drawing without any changes.

Additional layers can be added to the layer link by choosing the **Create Layer Link** command and selecting the additional layer or layers. The new layers are added to the link according to the layer's z and Δz settings.

For related information, see the following topics:

General Information about Layers

Layers

Layer Options

Layers can be deleted from a layer link. When created, layers in the layer link are locked, keeping them in registration. If a layer link is selected, it displays gray handles showing it is locked. To delete a layer from a layer link, select the layer and choose the **Unlock** command on the **Edit** menu. The handles display in black. Select the layer to be deleted and press the **Delete** key. The original layer is unaffected.

Layer Links and Elevations

Layer links can be used to create elevations. A useful technique is to create a layer link and switch the model layer to the front view, or right side, whichever is appropriate.

Select the entire drawing and choose the **Unlock** command. With the entire model still selected, choose the **Convert to Lines** command on the **Tool** menu. When the dialog box opens, click on the **Render** button to produce a hidden line rendering. The resulting object is a 2D representation of the model. While the object is no longer dynamically linked to the drawing layers, it is quite small and fast. It is easy to edit and add text.

Layer Links and Printing

Layer links can be used to create layers specifically for printing. For example, multiple layers of a floorplan can be arranged with the layers next to each other instead of stacked into a model. Layers can be printed displaying all layers in the drawing on a single sheet. This can be useful when used with classes, as different elements of the drawing can be displayed as appropriate.

Layer Links and Multiple Views

Links can be made to the same layer multiple times to give a multiple view effect. Any combination of views of the original can be linked, and the changes made to the original are updated in the multi-views every time the screen is redrawn.

For additional information, see the following topics:

Create Layer Link

Layers

Layer Options

The **Create Symbol** command on the **Organize** menu is used to create a symbol.

Symbols are an efficient way of displaying repeating items because only one copy of the item is stored in the drawing's **Symbol** library. No matter how many times the symbol is displayed in the drawing, only the single object in the library is stored in the drawing. Any edits to a symbol in the library affect all instances of the symbol placed in the drawing.

Symbols are stored in the active file, but, through the **Resource Palette**, they can be imported from external libraries or from other drawings.

Symbols can have a 2D component, a 3D component, or both. If the symbol has both, it is known as a "hybrid" symbol. A hybrid symbol appears correctly in any projection. That is, when in 2D/Plan projection, the 2D component is displayed. When in 3D orthogonal projection, the 3D component is displayed.

If the symbol is only one type, it always displays as drawn.

For additional information, see the following topics:

Creating Symbols
Creating Hybrid Symbols

To create a symbol, first create the objects, text, or group that are to become the symbol.

Then, select the objects and choose the **Create Symbol** command on the **Organize** menu. The **Create Symbol** dialog box opens.

Enter the symbol's name in the **Name** field.

Select one of the following options to define the symbol's "insertion point." The "insertion point" is the point within the symbol that, when the symbol is placed, will register exactly with whatever point you select in the drawing:

- **Plan Projection Center.** This option places the insertion point at the center of the symbol (as determined by its bounding box). When creating a 3D symbol in Top/Plan projection, the plan projection center is the center of the object on the ground plane. If the projection is isometric, this button changes to **3D Object Center**.
- **Next Mouse Click.** This option places the insertion point at the next mouse click. The snap constraints and data display bar can be used to set the insertion point.

If the symbol created is 3D, its *z* dimension is also determined at creation. However, the *z* dimension for a 3D symbol can be changed after it is placed (with the **3D Reshape** tool) without changing the *z* values of all symbols in the drawing.

When finished, click OK.

The symbol is removed from the drawing and placed in the library.

A symbol is placed in a drawing by first selecting it in the **Resource Palette**, then inserting it with the **Insert 2D Symbol** or **Insert 3D Symbol**.

For additional information, see the following topics:

Creating Hybrid Symbols
Symbols and Walls
Create Symbol

Converting a 3D symbol into a hybrid symbol is easy. First, create the symbol as you normally would (see **Creating Symbols**).

Once the symbol is created, double-click its name in the **Resource** palette and, if the symbol is 3D, click the **3D Edit** button.

When the **Edit** window opens, rotate the symbol to the orientation of the intended 2D view (usually top view). Draw the 2D component over the existing 3D symbol, using the existing symbol for reference. When finished, click the **Exit Symbol** button.

The symbol is now a hybrid. The 2D components display in Top/Plan, and the 3D components display in all 3D projections. The 2D components can be edited now by clicking on the **2D Edit** button.

The process of converting a 2D symbol to a hybrid is the same.

For additional information, see the following topics:

Symbols and Walls

Create Symbol

The **Custom Selection** command creates a macro that searches across the drawing for every instance of the criteria you select. Objects that meet the criteria are selected.

The macro is created by making selections from a dialog box. The macro is named and placed in a floating palette. Double-clicking the macro name in the palette, automatically selects everything in the drawing that meets the search criteria.

To create a custom selection macro choose the **Custom Selection** command on the **Organize** menu.

In the **Search Criteria** dialog box, set the search criteria then click **OK.** See **The Custom Selection Search Criteria dialog box** for description of its options.

A dialog box opens for the command's name. Enter a unique name for the macro command. When you click **OK**, the command is put in the active command palette. If no command palettes are open on the drawing, one is created.

Macro commands and command palettes can also be created and edited with the **Resource Palette**.

For related information, see the following topics:

Custom Tools/Attributes
Custom Visibility
Resource Palette

The **Search Criteria** dialog box is used to define the criteria used for a custom selection.

When an option presents a list of choices:

- Multiple selections can be made using the **Shift-Drag** or **Shift-Click** mouse technique.
- Multiple selections can be unselected by clicking the **Remove** button.

The **Search Criteria** dialog box has the following options:

Preset
Sets the default settings for the search criteria.

Edit
Edits the search criteria in the edit window.

Select
When clicked, the objects that meet the criteria are selected. Any items on the drawing already selected remain selected.

Deselect
When clicked, the objects that meet the criteria are deselected.

Select only
When clicked, only the objects that meet the criteria are selected.

Visible
When clicked, the search is limited to visible objects.

Selected
When clicked, the search is limited to selected objects.

All
When clicked, all objects in the drawing are subject to the search.

Layer
A search criteria can be limited to a particular layer or layers. To set the layer search criteria, click the **Layer** button. When clicked, a list of all layers in the drawing is displayed.

Class
The **Class** option is used to include instances of a particular class or classes. When clicked, a list of classes in the drawing is displayed. Select classes that are to be included in the custom selection.

Type
A search can be conducted for any of the object types used in MiniCad. To search for a object type, click on the **Type** button. When clicked, a list of MiniCad's object types is displayed. Select any of the types to be included in the search.

Name
You can search the drawing to select named objects. To include a named object in a search criteria, click the **Name** button. When clicked, a list of named objects is displayed. Select any of the names to be included in the search. .

Symbol
The custom command can select symbols placed on the drawing. To include a symbol in the search criteria, click on the **Symbol** button. When clicked, a list of symbols in the drawing is displayed. Select any of the symbols to be included in the search. Symbols can be located in folders nested within other folders. You can find them by looking at the path list at the top of the scrolling list of symbols.

Record/field

The custom selection command can search the drawing for instances of a specific record or fields within the record. In addition, certain operations can be performed on the record or field to further refine the search.

When you click the **Rec/Fld** button the **Record Field** dialog box is opened. Select the records and/or fields for which a search is to be conducted and double-click. After making all selections, click the **Done** button.

You can click the **Delete** button in the **Record Field** dialog to remove a specific record or field from the list. This will not remove the record or field from the drawing.

For more information about this option, see **The Record Field dialog box**.

Fill Pattern

The search can be limited to instances of a specific fill pattern. Click the **FilPat** button. Select the fill patterns for which a search is to be conducted and double-click. Edited fill patterns retain the ID number of the original fill pattern so the search criteria remains the same no matter how many times you edit the pattern. After making all selections, click the **Done** button.

You can click the **Delete** button to remove a selected fill pattern from the list.

Line Weight

The search to can be limited to instances of specific line weights or thickness. Click the **LineWgt** button. Select the line weights for which a search is to be conducted and double-click. It is possible to have more line weights on your drawing than the five selections available on the attribute palette. If there are more than five different line weights on the drawing, click on the **Set Thickness** line in the selection box. Type the desired line weight to be searched directly into the dialog box. After making all selections, click the **Done** button.

You can click the **Delete** button to remove a selected line weight from the list.

Line Style

The search can include instances of a specific dash style. Click the **LineStyl** button. The selection list displays all available dash styles. Select the dash styles for which a search is to be conducted and double-click. After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected dash style from the list.

Arrow Shape

The search can include instances of specific arrowheads and line markers. Click the **Arw Shape** button. There are seven available "slots" for different arrowhead shapes. Select the particular slot for which a search is to be conducted and double-click. After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected arrowhead or line marker from the list.

Arrow Size

The search can include instances of specific arrowheads sizes. Click the **Arw Size** button. A scrolling list of the markers on the drawing is displayed. Select the particular arrow size for which a search is to be conducted and double-click. After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected arrowhead or line marker from the list.

Fill/Pen Foreground/Background

The search can include instances of a specific color. MiniCad isolates the foreground color from the background color and also separates the pen (lines and text) and fill (surfaces). To search by color, click one of the color buttons (**FillFore**, **FillBack**, **PenFore**, **PenBack**). The color palette appears. Select the colors to search for, and double-click. After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected color from the list.

The **Record Field** dialog box has the following options

No operators
When selected, no operation is performed on the search criteria.

Less than (<)
When selected, the search is successful only if the search criteria is less than the value in the limit value field.

Greater than (>)
When selected, the search is successful only if the search criteria is greater than the value in the limit value field.

Equal (=)
When selected, the search is successful only if the search criteria is equal to the value in the limit value field.

Less than or equal (<=)
When selected, the search is successful only if the search criteria is less than or equal to the value in the limit value field.

Greater than or equal (=>)
When selected, the search is successful only if the search criteria is greater than or equal to the value in the limit value field.

Not equal (<>)
When selected, the search is successful only if the search criteria is not equal to the value in the limit value field.

Enter a Limit Value
When using the greater than and less than operators, enter the limit value here.

Record search only
When checked, the search is limited to records.

Different combinations of tools and attributes can be configured to form custom tools. These custom tools can be named and placed into palettes and accessed with a double-click on the name in the palette. Custom tools are created with the **Custom Tools/Attributes** command on the **Organize** menu.

Custom tools are created by changing the default attributes in the drawing as required for the special tool. The desired tool, if any, is selected. Finally, when all attributes and tools are properly selected, the **Custom Tool/Attribute** command is chosen from the **Organize** menu.

When the command is chosen, the **Custom Tools/Attributes** dialog box opens. The specific selections made in this dialog box control the command's action when it is selected. Any or all attributes can be selected from the dialog box. The two buttons labeled **All On** and **All Off** can be used to select or deselect all attributes at once.

See **The Custom Tools/Attributes dialog box** for a description of its options.

When the **OK** button is clicked, a dialog box opens for the command's name. Enter a unique name for the macro command.

The command is put in the active command palette, if one is open. If no command palettes have been created before, one is created.

Macro commands and command palettes can also be created and edited with the **Resource Palette**.

For related information, see the following topics:

Custom Selection

Custom Visibility

Resource Palette

The **Custom Tools/Attributes** dialog box is used to define the criteria used for a custom selection. It has the following options:

Pen
The pen color controls the color of lines, hatches, and text. You have individual control over foreground color, background color, and pattern. Text color is controlled by the pen color. Generally, the line and text color are controlled by the foreground color. All pen attributes can be selected by clicking the **Pen** checkbox.

Fill
The fill color determines the surface color and pattern of objects. You have individual control over foreground color, background color, and pattern. Generally, the line and text color is controlled by the background color. All fill attributes can be selected by clicking the **Fill** checkbox.

Line
The **Line** section of the dialog box controls whether a line's weight (thickness), style (dash patterns), markers (arrowheads), and marker size becomes part of the tool's attribute. All line attributes can be selected by clicking the **Line** checkbox.

Text
Select any combination of text attributes. All text attributes can be selected by clicking the **Text** checkbox.

Class
If selected, the current class is included as a tool attribute

Layer
If selected, the current layer is included as a tool attribute

Symbol
Select any symbol to be used in combination with the selected attributes.

Tool
If selected, the active tool can be used in combination with the selected attributes.

Constraint
If selected, the current snap constraint(s) is selected as an attribute.

Macro commands and command palettes can be created and edited with the **Resource Palette**.

The **Custom Visibility** command on the **Organize** menu is used to create commands that search across the drawing for every instance of specific criteria. Objects that match the criteria can be made visible or invisible by double-clicking the macro's name on the command palette. These commands, or macros, can control the visibility of any combination of objects on the drawing.

To create a custom visibility command, choose **Custom Visibility** from the **Organize** menu. All selections for the command are made from the **Search Criteria** dialog box. Set the search criteria by clicking the appropriate buttons.

See **The Custom Visibility Search Criteria dialog box** for description of its options.

When done, click OK. A dialog box opens to enter the command's name. All commands must be given a unique name. The command is put in the active command palette if one is open. If no command palette exists in the drawing, one is created.

Macro commands and command palettes can also be created and edited with the **Resource Palette**.

For related information, see the following topics:

Custom Tools/Attributes
Custom Selection
Resource Palette

The **Search Criteria** dialog box is used to define the criteria used for a custom selection.

When an option presents a list of choices:

- Multiple selections can be made using the **Shift-Drag** or **Shift-Click** mouse technique.
- Multiple selections can be unselected by clicking the **Remove** button.

The **Search Criteria** dialog box has the following options:

Preset
Sets the default settings for the search criteria.

Edit
Changes the settings for the search criteria.

Show
When selected, the objects that meet the criteria are made visible.

Hide
When selected, the objects that meet the criteria are made invisible.

Show only
When selected, only the objects that meet the criteria are made visible. All other objects are hidden.

Visible
When selected, the search is limited to visible objects that meet the search criteria.

Selected
When selected, the search is limited to selected objects that meet the criteria.

All
When selected, all objects in the drawing are subject to the search.

Layer
A search criteria can be limited to a particular layer or layers. To set the layer search criteria, click the **Layer** button. When clicked, a list of all layers in the drawing is displayed.

Class
The search can include instances of a particular class or classes. This is done by clicking the **Class** button. When clicked, a list of classes in the drawing is displayed. Select any classes to be included in the search.

Type
Any of the object types can be made invisible as a group. To search for an object type, click on the **Type** button. When clicked, a list of MiniCad's object types is displayed. Select any of the types to be included in the search.

Name
You can search the drawing to select named objects. To include a named object in a search criteria, click the **Name** button. When clicked, a list of named objects is displayed. Select any of the names to be included in the search.

Symbol
The custom command can control the visibility of symbols placed on the drawing. To include a symbol in the search criteria, click on the **Symbol** button. When clicked, a list of symbols in the drawing is displayed. Select any of these symbols to be included in the search. Symbols can be located in folders nested within other folders. You can find them by looking at the path list at the top of the scrolling list of

symbols.

Record/Field

The custom selection command can search the drawing for instances of a specific record or fields within the record. In addition, certain operations can be performed on the record or field to further refine the search. To do this, click the **Rec/Fld** button

In the **Record Field** dialog box, select the records and/or fields for which a search is to be conducted and double-click. After making all selections, click the **Done** button.

You can click the **Delete** button in the **Record Field** dialog to remove a specific record or field from the list. This will not remove the record or field from the drawing.

For more information about this option, see **The Record Field dialog box**.

Fill Pattern

The search can be limited to instances of a specific fill pattern. To do this, click the **FilPat** button. Select the fill patterns for which a search is to be conducted and double-click. Edited fill patterns retain the I.D. number of the original fill pattern, so the search criteria remains the same no matter how many times you edit the pattern.

After making all selections, click the **Done** button.

You can click the **Delete** button to remove a selected fill pattern from the list.

Line Weight

The search can be limited to instances of specific line weights or thickness. To do this, click the **LineWgt** button. Select the line weights for which a search is to be conducted and double-click.

It is possible to have more line weights on your drawing than the five selections available on the attribute palette. If there are more than five different line weights on the drawing, click on the **Set Thickness** line in the selection box. Type the desired line weight to be searched directly into the dialog box.

After making all selections, click the **Done** button.

You can click the **Delete** button to remove a selected line weight from the list.

Line Style

The search can include instances of a specific dash style. To do this, click the **LineStyl** button. The selection list displays all available dash styles. Select the dash styles for which a search is to be conducted and double-click.

After making all selections, click the **Done** button.

You can click the **Delete** button to remove a selected dash style from the list.

Arrow Shape

The search can include instances of specific arrowheads and line markers. Click the **Arw Shape** button. There are seven available "slots" for different arrowhead shapes. Select the particular slot for which a search is to be conducted and double-click.

After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected arrowhead or line marker from the list.

Arrow Size

The search can include instances of specific arrowheads sizes. Click the **Arw Size** button. A scrolling list of the markers on the drawing is displayed. Select the particular arrow size for which a search is to be conducted and double-click.

After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected arrowhead or line marker from the list.

Fill/Pen Foreground/Background

The search can include instances of a specific color. MiniCad isolates the foreground color from the background color and also separates the pen (lines and text) and fill (surfaces). To search by color, click one of the color buttons (**FillFore**, **FillBack**, **PenFore**, **PenBack**). The color palette appears. Select the colors to search for, and double-click.

After making the selections, click the **Done** button.

You can click the **Delete** button to remove a selected color from the list.

The **Cut** command on the **Edit** menu removes the selected objects or text from the drawing and places it in the Clipboard.

If the **Show/Snap/Modify** option is set with the **Layer Options** command, you can select multiple objects on different layers and cut them to the Clipboard. When pasted, the objects are placed on the active layer.

If objects are on the Clipboard, be careful not to inadvertently switch applications. MiniCad is a high resolution program. The resolution is maintained with objects copied to the Clipboard as long as MiniCad remains the active application. If the application is switched, the contents of the Clipboard may be converted to 72 dpi.

MiniCad objects pasted into other applications from the Clipboard will lose detail. You can use the **Export** command to bring high resolution objects into other applications without any loss of detail.

Note: Do not confuse the **Cut** command with the **Delete** key or the **Clear** command. These two methods do not copy the deleted objects on the Clipboard.

For related information, see the following topics:

Paste

Copy

Clear

The **Cut 2D Section** command on the **3D** menu produces a 2D object that is a "slice" of a 3D object. The section can be cut through a single layer or through a layer link of multiple layers.

To cut a 2D section, choose the layer that is to be sectioned. Orient the drawing to the appropriate view for the section.

Next, choose the **Cut 2D Section** command. Drag the cursor along the line where the section is to be cut and click the mouse on one side of the section line.

The specified section is copied to a new layer, and the drawing is automatically changed to that layer. The new layer is given the default layer name, which you can change using the **Layers** command.

The original layer, and the objects contained on it, remain unchanged. There is no link between the original object and the new section.

If the image is reversed after a section, use the **2D Mirror** tool to correct the orientation.

For related information, see the following topics:

Cut 3D Section

The **Cut 3D Section** tool on the **3D** menu cuts a section through a 3D model. Sections can be cut through a linked layer model and through 3D symbols.

When chosen, this command allows you to set the plane used to cut the section. After cutting the plane, you are asked to set the side of the model to be cut away. The sectioned model is drawn in a new, untitled layer. The sectioned model has no links to the original model nor to that model's original layers.

As a 3D object, the new section can be rotated to any view, or viewed through any projection. For this reason, you must be aware of the implications of adding 2D information to your 3D section. Because dimension information or text annotations are 2D objects, these elements cannot be attached to the sectioned model if the model is rotated.

For related information, see the following topics:

Cut 2D Section

All open documents are listed on the **Window** menu. The selected document is identified with a check mark next to its name in the menu. A document is made active by selecting its name from the menu.

When the **Calculate Terrain Model** command is chosen with 3D loci or polygons selected on the active layer, the **Digital Terrain Model** dialog box opens.

Note: This command is only available when the **Terrain Modeler Overlay** is active. If the **DTM** menu is not listed on the menu bar, select the **Terrain Modeler Overlay** using the **Overlay** command on the **File** menu.

To learn more about the DTM, see **General Information About the Digital Terrain Modeler** and **Importing Data into the Digital Terrain Modeler**.

The **Digital Terrain Model** dialog box has the following fields and buttons:

Name
Unique name of the DTM. This is the name of the layer that contains the terrain data. If the layer name is changed, the DTM is deleted.

2D To
Name of the layer where the 2D portion of the terrain model is output. If the layer doesn't exist, it is created.

3D To
Name of the layer where the 3D portion of the terrain model is output. If the layer doesn't exist, it is created.

Current Data
The lowest and highest points in the current selection of loci or polygon vertices, and the difference between them (the range). If a DTM exists on the layer, the field returns the lowest and highest points in the DTM and their range.

Contour Interval field
Enter the desired contour interval. This value must be greater then one, but it doesn't need to be an integer.

Major Multiple
The **Major Multiple** value is the number of contour lines per index number. The major multiple must be an integer.

Datum Elevation
Used for calculating the volume between the irregular surface of the DTM and the horizontal datum plane. The elevation can be any number.

Base Elevation field
The contour is based on the value in the **Base Elevation** field. This value is usually zero, but it can be any number.

Minimum Elevation field
The value in the **Minimum Elevation** field is the lowest point in the terrain model. By default the **Minimum Elevation** represents the lowest 3D loci or 3D polygon vertex on the layer. In a "cut and fill" DTM, the minimum elevation is typically less then the maximum cut (usually a negative number).

Maximum Elevation field
The value in the maximum elevation field is the highest point in the terrain model. By default the maximum elevation represents the highest 3D loci or 3D polygon vertex on the layer. In a "cut and fill" DTM, the maximum elevation is typically greater then the maximum fill depth.

2D Contours

When selected, 2D contours are created.

Surface 3D
When selected, 3D triangular polygons are created. This creates a 3D terrain model with a multifaceted surface. The model can be rendered as a solid. 3D polygons cannot be smoothed.

Contour Labels
When selected, contour labels are created at the index 2D contours. The index contours are placed at the major multiples, where the index contours meet the hull (i.e., the edge of the DTM).

Triangles (2D)
When selected, 2D triangular polygons are created.

3D Contours
When selected, 3D contours are created using 3D polygons. 3D polygons cannot be smoothed, but the 3D polygons retain the elevations that they represent.

Hull
When selected, a closed 2D polygon is created representing the hull, or boundary of the DTM.

Smooth 2D Contours
When selected, the 2D contours are smoothed with Bezier splines at creation and extra vertices are created as required. The contour can be smoothed after it is created, but vertices are not added.

Group Objects
When selected, like objects are grouped upon creation. For example, all 2D contour lines or 3D triangles are grouped together. Contour labels are grouped with 2D contour lines.

Use Breaklines
When selected, 3D polygons are used as breaklines in the DTM. Breaklines force linear interpolation along each leg of each polygon.

Always Recalculate
When selected, the DTM automatically recalculates whenever the dialog box is changed. It is usually desirable to always recalculate. When selected, less RAM is required. However, because the DTM is recreated each time it is recalculated, you may wish to disable this option in a complex DTM.

For related information, see the following topics:

Display Area
New Cut & Fill Model
Display Cut & Fill-To Datum

The area of a terrain model can be calculated with the **Display Area** command. When the **Display Area** command is chosen, the area of the terrain model is displayed in a dialog box.

Note: This command is only available when the **Terrain Modeler Overlay** is active. If the **DTM** menu is not listed on the menu bar, select the **Terrain Modeler Overlay** using the **Overlay** command on the **File** menu.

The area is calculated by taking the projection of the model on a horizontal plane. The unit of measure is the same as the drawing's current unit of measure, cubed. The command does not calculate the sum of the areas of the sloped surfaces.

A layer with an existing terrain model must be active in order to calculate the display area of the model.

To learn more about the DTM, see **General Information About the Digital Terrain Modeler** and **Importing Data into the Digital Terrain Modeler**.

For related information, see the following topics:

Calculate Terrain Model

New Cut & Fill Model

Display Cut & Fill-To Datum

The **Display Cut and Fill to Datum** command calculates and displays (in a dialog box) the total cut and total fill to get from the surface of the model to a datum plane.

Note: This command is only available when the **Terrain Modeler Overlay** is active. If the **DTM** menu is not listed on the menu bar, select the **Terrain Modeler Overlay** using the **Overlay** command on the **File** menu.

The elevation of the plane is indicated in the dialog box which appears when you choose to create or change a model.

The plane may be above, below, or intersecting the model. If it is above, then there will only be fill; if it is below, then there will only be cut; if it intersects, then there will be both cut and fill.

If the site is balanced, then the cut and fill volumes will be identical; this would indicate that soil cut from high parts would be exactly enough to fill low parts, creating a level surface at the datum elevation.

Bulking and compaction factors are not used, and there is no way to take into account the stripping of topsoil (other than adjusting the original data upon which the model was based).

A layer with an existing terrain model must be active in order to calculate the volume of the cut and fill to datum.

To calculate the cut and fill necessary to get from one model to another, you must first create a cut and fill model, then calculate its cut and fill volumes to get to a datum plane of elevation zero.

To learn more about the DTM, see **General Information About the Digital Terrain Modeler** and **Importing Data into the Digital Terrain Modeler**.

For related information, see the following topics:

Display Area
Calculate Terrain Model
New Cut & Fill Model

The cut and fill between two terrain models can be calculated. There must be at least two existing terrain models in the drawing in order to calculate the cut and fill.

Note: This command is only available when the **Terrain Modeler Overlay** is active. If the **DTM** menu is not listed on the menu bar, select the **Terrain Modeler Overlay** using the **Overlay** command on the **File** menu.

The name given to the model will be the name of the active layer. This named can be changed. The names of the 2D and 3D output layers can also be edited.

This command is similar to the **Calculate Terrain Model** command. But instead of using 3D loci and/or 3D polygons as its input, the **New Cut & Fill** command uses two existing models.

One of the models must be totally enclosed by the other (they may have common boundaries as well). The enclosing model typically represents existing terrain, and the enclosed model typically represents proposed terrain. The calculations assume going from the enclosing model to the enclosed model.

The "enclosing" model is specified in the first pop-up menu in the **Digital Terrain Model** dialog box. The "enclosed" model is specified in the second. None of the data in the enclosed model may extend beyond the boundaries of the enclosing model.

The new model created by this command represents the difference in elevation between these two models.

Where the new model's elevation is zero, the two chosen models are at the same elevation.

Where the enclosed model is at a higher elevation, the elevation in the new model is positive. By calculating the cut and fill to a datum plane of zero elevation on the new model, the cut and fill volumes to get from the enclosing model to the enclosed model will be displayed.

The border of the new model will be the same as that of the enclosed model. A cookie cutter approach is used, meaning that vertical walls are assumed at the edge of the new model to represent the transition from the enclosing model to the enclosed model.

To learn more about the DTM, see **General Information About the Digital Terrain Modeler** and **Importing Data into the Digital Terrain Modeler**.

For related information, see the following topics:

Display Area
Calculate Terrain Model
Display Cut & Fill-To Datum

The Digital Terrain Modeler (DTM) reads 3D loci and 3D polygons to create the terrain models. 3D loci are typically created by importing 3D point data provided by surveyors.

3D polygons are typically created by digitizing an existing contour map, or importing a DXF file containing 3D polylines which represent contour lines. The module uses each vertex of each 3D polygon, along with their individual elevations, so 3D polygons are not assumed to be, and don't have to be, horizontal.

The module uses the Triangulated Irregular Network (TIN) method. The data may be scattered and in clumps; it need not be organized in any rectangular grid. The terrain modeler will use all the data points, without doing approximations. A network of triangles is created from the points. These triangles then form the terrain model. Interpolation is performed within the triangles to calculate the threading of contour lines.

Multiple terrain models can be used. For example, two terrain models can be created--one representing existing terrain, and another representing the proposed terrain. It is then possible to calculate the cut and fill volumes from the terrain to a datum plane. If two models exist, then the terrain modeler can create a new model which represents the cut and fill needed to get from one model to the other.

The model can represent terrain, hydrological measurements, temperature, financial data, and a wide range of other types of information.

Three layers are used per model. The original data resides in one layer, the 2D results are put into a second layer, and the 3D results are put into a third layer. If the name of the model is "existing", then the name of the 2D layer will be "existing_2D", and the name of the 3D layer will be "existing_3D". You may change the names which will be assigned to the output layers.

Once you have created a model, you can edit and delete the standard graphic objects which were created to represent it. You can also change and delete the model as a whole. This assumes that you do not change the names of the three layers associated with it. To perform an operation on a model, you must select one of the three layers associated with it.

Terrain model information is saved along with a drawing after each DTM-related command. If you save your drawing and open it again, the original internal DTM data remains.

For related information, see the following topics:

Importing Data into the Digital Terrain Modeler

Display Area

Calculate Terrain Model

New Cut & Fill Model

Display Cut & Fill-To Datum

3D point data may be imported and converted into 3D loci. The data must be in a text (ASCII) file in the same folder as the MiniCad application.

The text file must contain the data in tab-delimited y, x, z (northing, easting, elevation) format. There should be no other characters, such as headings, point numbers, or descriptions, in the file.

Tab-delimited files have a format such as:

```
123 <Tab> 456 <Tab> 78 <CR>
```

This is an example of a *y,x,z* format, which represents northing, easting, and elevation (*n,e,z*). It is identical in appearance to an *x,y,z* (*e,n,z*) file. The only difference is that the first two columns are reversed.

There is a **Tab** character after each of the first two numbers, and a carriage return at the end of the line.

The most common formats are *y,x,z*; *x,y,z*; and #,*n,e,z,d*. The last format consists of five items per line: point number, northing, easting, elevation, and description. This kind of data can be entered with a macro found in the DTM Toolkit. This macro reads #,*n,e,z,d* files, stripping the first and last items from each line.

Configure the units and scale before importing data into the drawing. The units in the file should match the data in the file from which the data is imported. The **Display as Fractions** button on the **Units** dialog box should be deselected.

Scale must be set before importing data. Inspect the data to be imported and choose an approximate scale. Set the scale to one that will allow all of your data to fit in the drawing (consider mainly the horizontal coordinates).

For instance, if your northing (*y*) values range from 1100 to 1300 feet, and your easting (*x*) values range from 2500 to 2800 feet, then the *x* and *y* ranges will be 300 feet and 200 feet, respectively.

If you have set your units to feet, and you want the data to fit in one page, you may want to set the scale to 1"=50', which gives a paper scale of 1:600. In most cases you will want the scale to apply to all layers.

You may want to create a layer specifically for one model's data. Give it a name such as "existing", "proposal 1", etc. The data will be imported into the active layer as 3D loci.

Now you are ready to import the data. On the **Resource Palette**, select the appropriate macro from those listed in the **DTM Toolki**t, then click in the **Run** button.

The macro will present a standard dialog box in which you should select the data file you want to import. The data file must be in the MiniCad application folder.

Once you have imported your data you will probably need to bring it into view. The easiest way is to choose the **Normal Scale** command on the **Page** menu. If asked whether to center on off-screen objects, click **Yes**.

When the data is imported, choose the **Calculate Terrain Model** command

Terrain data also can be imported as DXF files. The contour lines are represented as polylines in DXF, and are converted into polygons in MiniCad.

If the polylines are only 2D, the elevations represented by the contour lines are not imported.

If a 3D terrain model is desired, the 2D polylines must be converted to 3D polygons and moved to the proper *z* value for the respective elevations.

For related information, see the following topics:

General Information About the Digital Terrain Modeler

Display Area

Calculate Terrain Model

New Cut & Fill Model
Display Cut & Fill-To Datum

The **Drawing Size** command on the **Page** menu controls the number of pages used for printing.

The size of the paper is controlled by the settings made with the **Print Setup** command.

Since different printers have different printable areas on a page, switching printers will affect the print area. Be aware that MiniCad doesn't determine the print area--this is done by the printer.

The drawing area used by MiniCad, and the portion of the drawing area that is printed, are different.

A gray border in the drawing represents the overall size of the printable area of the drawing. The actual number of pages needed to print the drawing at this size is shown within the border. If only a portion of a page is to be used for tiling, only that portion is shown.

The **Drawing Size** dialog has the following options:

Edit Window
The print area can be enlarged by dragging the small rectangle in the upper left corner of the edit window to select the number of pieces of paper desired.

Size
The **Size** pop-up menu is used to select standard paper sizes, including technical sheet sizes used by plotters. A custom page size can be specified using the **Height** and **Width** fields.

MiniCad automatically tiles the drawing across the required number of printer pages to match the paper size you specify.

Width and Height
The actual print area for the selected printer (times the number of pages selected) is shown in the height and width fields.

Custom page sizes can he entered directly in the fields. Some printers allow for enlargement or reduction of the print area. These settings, made with the **Print Setup** command, will affect the dimension shown in the **Height** and **Width** fields.

Inch/millimeter
Choose a unit of measure by clicking one of the buttons.

Show Page Breaks
Select this checkbox to show the page breaks.

The **Duplicate** command on the **Edit** menu makes an exact copy of a selected object.

When an object is duplicated, the copy can be offset from the original. The offset is controlled with the **Preferences** command on the **File** menu.

If the duplicated object is moved immediately after it is created, subsequent duplicates will be placed at the same offset. If **Offset Duplications** is deselected, the copy is placed on top of the original.

Note: unlike the **Copy** command, the **Duplicate** command does not place a copy of the object on the Clipboard.

For related information, see the following topics:

Copy
Duplicate Array
Duplicate Along Path

With the **Duplicate Array** command, it is possible to make multiple copies of the object, resize the duplicate, and rotate or move the duplicate.

Note: The **Duplicate Array** command is located on the alternate **Edit** menu. This menu is activated by, pressing the **Control** key while selecting the **Edit** menu from the menu bar.

Selecting the **Duplicate Array** command opens the **Duplicate Array** dialog box.

The possible array types are: Linear, Circular, and Rectangular.

Objects can be resized as they are duplicated by selecting the **Resize Duplicates** button. The object's *x* and y dimensions are multiplied by the specified factor. Values greater than one enlarge the duplicate. Values less than one reduce the duplicate.

Duplicated objects can be rotated from the original by selecting the **Rotate Duplicates** option and entering the degree of rotation in the **Angle** field.

For additional information, see the following topics:

Creating Linear Arrays
Creating Rectangular Arrays
Circular Arrays

For related information, see the following topics:

Copy
Duplicate Along Path
Sweep

To create a linear array with the **Duplicate Array** command, select the **Linear Array** button in the **Duplicate Array** dialog box. Then, type the number of copies in the **Copies** field, and enter the *x* and *y* distance from the center of the original to the center of the copy.

When setting the offset for the direction of the duplication, a positive *x* value makes the offset move to the right. A negative *x* value moves the duplicate to the left.

A positive *y* moves the duplicate upward. A negative *y* moves the duplicate downward.

Clicking the **Next Mouse** button lets you set the offset with the mouse.

To create a Rectangular array with the **Duplicate Array** command, select the **Rectangular Array** button in the **Duplicate Array** dialog box. In this mode, duplicates are placed in rows and columns.

The **Copies** field changes to **Row** and **Columns**. Type in the desired values. The *x* offset determines the distance from center-to-center of individual columns.

When setting the offset for the direction of the duplication, a positive *x* value makes the offset move to the right. A negative *x* value moves the duplicate to the left. A positive *y* moves the duplicate upward. A negative *y* moves the duplicate downward.

Clicking the **Next Mouse** button lets you set the offset with the mouse.

To create a Circular array with the **Duplicate Array** command, select the **Circular Array** button in the **Duplicate Array** dialog box. Enter the copies and angle values.

In the **Center** field, enter the radius for the arc angle with the *x* and *y* fields. The **Next Mouse** option allows the radius for the arc angle to be set with a mouse click.

Object attributes, such as fill pattern, line thickness, dash style, arrow head and the color palette can be edited with the **Edit Attributes** command on the **Edit** menu.

To edit an attribute, select it from the **Edit Attributes'** submenu.

For additional information, see the following topics:

Editing Patterns
Editing Line Thickness
Editing Dash Styles
Editing Arrow Heads/Line Markers
Editing the Color Palette

Thirty six of the patterns on the **Attribute Palette** can be edited. These patterns are shown on the bottom half of the pattern menu.

These patterns are edited using the **Edit Attributes** command's **Pattern** option. When **Patterns** is chosen from the **Edit Attributes** submenu, the editable patterns are displayed on the pop-up menu.

To edit a pattern, select a pattern from the pop-up menu. The full pattern is displayed in the right-hand window. A close-up bitmap version of the pattern is displayed in an editing window on the left.

In the editing window you can change the pattern by clicking individual bits off or on. Your changes are reflected in the full size window.

Changes to a pattern are global, which means that every instance of that pattern number on your drawing is changed.

Pattern edits are limited to the active drawing. The custom pattern will not be transferred to a new drawing if you copy and paste an object with that pattern applied, and there is no way to move a custom pattern from one drawing to another. However, if a custom pattern is used repeatedly, you may want to edit the pattern and save a blank version of the file with the **Save as Stationery** command. The new pattern will be available to all new documents.

The **Attribute Palette** has five default line thicknesses. All five of these default line weights can be modified.

To edit a line weight, choose the **Edit Attributes** command from the **Edit** menu, then select the **Line Thickness** option. The **Preferred Line Thickness** dialog box opens. The values in the edit fields can be displayed in points, mils or millimeters. The current line thickness is displayed in the selected unit of measure.

Changes are made by entering new values in the **New** fields.

When lines are edited, the changes are not global. Lines already in the drawing remain unchanged. To change a line to the new weight, select the object and choose the desired line weight from the **Attribute** palette.

Custom line weights are saved as preferences. They are available in all new drawings. It is not possible to return to the default line weights unless the preferences file is replaced.

The **Dash Styles** command on the **Edit Attributes** menu is used to add a new dash style to the drawing, or to change one that is already there.

MiniCad comes with eight standard dash patterns. These standard dash patterns can be edited, and up to eight additional dash patterns can be added, for a maximum of 16 patterns.

To create a new dash style or edit an existing one, choose the **Edit Attributes** command from the **Edit** menu, then select the **Dash Styles** option.

To edit an existing dash style, select one from the pop-up menu on the dialog box. To make a new dash style, click on the **Add** button.

The dialog box has three sliders to make the pattern. The first one on the left controls the dash. The second one controls the space between the dashes. The slider control on the far right creates two new sliders to make additional dashes and spaces.

If you create a custom dash style and apply it to an object, and copy that object into another file, the new style becomes available in that file. If you copy a custom dash pattern into a file that already contains the maximum sixteen dash patterns, the dash pattern reverts to solid.

The **Edit Dash** dialog box has the following options:

Dash Style #
All of the existing dash styles in the active file are displayed in this menu. Choose the dash style to be edited from this menu. This menu also is useful for creating new dash styles that are similar to existing ones. To create a new style, choose the dash style closest to the new style and click the **Add** button.

Inches / mm buttons
Changes the display for reference purposes as the custom dash style is created. It has no real bearing on the dash style.

Scale with line thickness checkbox
When selected, the gap between lines is scaled to maintain the gap's proportion as the line is made thicker.

Revert
Discards any changes made to the current dash.

Add
Adds a new dash style for editing.

Cancel
Closes the dialog box and returns to the drawing window without any changes.

OK
Accepts the changes and add the new and edited dash styles to the Attribute palette.

MiniCad has seven available line markers. Any of the seven default markers can be edited.

To edit a line marker, choose the **Edit Attributes** command from the **Edit** menu, then select the **Arrow Heads** option.

Each marker occupies a "slot," which is selected from the **Arrow #** pop-up menu. The marker type to be edited is selected with the **Style** pop-up menu. When clicked, the pop-up menu displays the seven slots and the current marker in each.

Arrowhead changes made with the **Edit Attributes** command are "global." That is, the changes affect all MiniCad drawings. These changes are stored in the preferences file.

The **Set Arrow Head Size** dialog box contains the following options:

Style

The marker to be edited is selected with the **Style** pop-up menu. When clicked, the pop-up menu displays the seven default markers.

Angle

Arrow heads and markers can be any angle up to 90°. The angle of the marker is displayed in the **Angle** field. Any changes in the edit window are reflected in this field. The field accepts direct entry of the arrow head angle. In addition, the marker in the edit window reflects any changes to the value in the **Angle** field. Each one of the seven slots can have a different angle on its marker.

Length

The length of the marker is displayed in the **Length** field. Any changes in the edit window are reflected in this field. The field accepts direct entry of the arrow head angle. Each of the seven slots can have a different length marker. The current units for the file is used for the marker length.

Arrow #

MiniCad has seven "slots" for line markers. Each slot can be edited independently. The slot to be edited is selected with the **Marker #** pop-up menu. When clicked, the menu displays the seven slots, and the current marker in each.

MiniCad uses a color palette containing 256 different colors. While the palette is limited to 256 separate colors, the colors within the palette can be completely customized.

To edit the color palette, choose the **Edit Attributes** command from the **Edit** menu, then select the **Color Palette** option.

The **Edit Color Palette** dialog contains the drawing's palette. Each box on the palette represents one of the 256 available "slots" that can be edited. Boxes can be edited individually, or a range of boxes can be edited as a group.

The **Edit Color Palette** dialog box has the following options:

Pick Color

Opens a dialog box in which the drawing's palette is displayed. The range of colors in this palette depends on a number of factors including the computer hardware being used.

To change a color in the palette, click the color in the palette. The selected color is shown. Changes to the color can be made by entering new RGB values in the fields.

Click on the **Add to Custom Colors** button to add the edited color to the color palette.

Import

Color palettes can be imported from any MiniCad 6 file. When the **Import** button is clicked, a dialog box opens. Select the file containing the color palette to be imported, then click **OK**. The color palette of the selected file replaces the color palette in the active file.

Lighten

When clicked, the selected color, or range of colors is lightened. When a color is lightened, the brightness level of the color is adjusted upward about 10 percent.

Darken

When clicked, the selected color, or range of colors is darkened. When a color is darkened, the brightness level of the color is adjusted downward about 10 percent.

RGB Blend

Blends the selected range of colors using the RGB color table.

HSV Blend

Blends the selected range of colors using the HSV color table.

Undo

Returns the color palette to the state it was in when the **Edit Color Palette** dialog box was open (not the color palette's default state).

The **Edit Group** command on the **Organize** menu provides a way to edit a grouped object without ungrouping it. This is useful if you need to preserve the nested character of a group as well as edit within the group.

Note: This command will automatically appear as **Edit Symbol** if a symbol is selected. Remember, when you edit a symbol, the changes are applied to all instances of that symbol in the drawing.

The **Edit Group** command is required to make changes to hybrid objects, such as roofs, floors, and walls. Although these objects are created without using the group command, they are special grouped objects, and need to be entered before they can be edited.

Important! Although it is possible to ungroup a hybrid object, you can never regroup it into a hybrid object again. For this reason, hybrid objects should always be edited with the **Edit Group** command.

When editing a hybrid object, you must display the component you want to edit before you select the **Edit Group** command. Once **Edit Group** has been selected, you cannot change between 2D and 3D projections.

Each time the **Edit Group** command is chosen, you are taken another level deeper into the group. It is possible to hide the rest of the drawing while editing by choosing the **Preferences** command and deselecting the **Show Other Objects While in Groups** checkbox. This is useful because it reduces screen clutter. Conversely, showing the entire drawing helps align objects in the group with the rest of the drawing.

When all edits have been made, you can return to the main drawing window by selecting the **Exit Group** command on the **Organize** menu, or by clicking the **Exit Group** button in the mode bar. Each level of the group you have entered requires a separate **Exit Group** command.

If the object has many levels of grouping, the **Top Level** command on the **Organize** menu returns to the main drawing level in one step.

The **Engineering Properties** command on the **Mechanical** menu calculates and displays in a data window the area, perimeter, centroid, and moments of inertia about both the centroid and any user specified point, and the radii of gyration of any selected 2D object.

Note: This command is only available when the **Mechanical Overlay** is active. If the **Mechanical** menu is not listed on the menu bar, select **Mechanical Overlays** using the **Overlays** command on the **File** menu.

Only one object at a time can be selected for its properties information. However, an object and a locus point can be selected together.

The **Exit Group** command reverses the action of the **Edit Group** command. in order to return you to the main drawing, this command must be chosen as many times as the **Edit Group** command was chosen.

When in **Edit Group** mode, this operation can also be performed using the **Exit Group** button on the mode bar . If you have entered a group that is many levels deep, you may want to use the **Top Level** command on the **Organize** menu to quickly return to the main drawing.

MiniCad supports a range of export formats that allow objects and data created in MiniCad to be manipulated in other programs.

Exporting in a different format does not change the original file. It creates a new file in the format specified.

A dialog box allows you to name the exported file and save it to the disk and folder of your choice. The following formats are supported:

Export Pict
Export EPSF
Export DXF
Export Text Format
Export Worksheet
Export Database

Exports a MiniCad file as a PICT document. A loss of resolution may occur during a PICT export.
Files exported as PICT carry object-oriented attributes such as thin lines, fill patterns, and object colors.

Files exported in EPSF format can be read by many other graphic and page layout programs. MiniCad exports EPSF files at high resolution with full accuracy.

All drawing elements, except color bitmaps, appear in the EPSF file as you see them on screen, including 3D objects and grayed layers. Color bitmap objects are not supported and do not appear in the EPSF file.

MiniCad EPSF files are exported in Illustrator '88 format. They can be edited in any program that supports that format, including Illustrator 5.0 by Adobe.

Files exported in DXF can be read by other CAD programs and printed at a service bureau. DXF is the de facto standard in the DOS CAD marketplace.

Files saved in DXF can be opened in AutoCAD. DXF exports can be used to transfer a MiniCad 3D model into a rendering program, such as Specular's Infini-D or into a modeling program, such as FormZ by autodessys.

The DXF translator in MiniCad writes to DXF 12 format.

MiniCad reads all 2D and 3D entities in the DXF export file. All visible objects in visible layers are exported. Export of 2D or 3D information depends on the projection chosen at export. That is, if a layer is in 2D/Plan projection, the DXF export is in 2D. If the layer is in a 3D projection, it is exported in 3D.

To export a file in DXF format, choose **Export** from the **File** menu and **DXF** on the submenu. Name the exported file and choose a location to place the file in.

The **Export DXF** dialog box contains the following options:

Export as DXF Layers

This menu contains four choices for converting different MiniCad structures into AutoCAD layers.

The default converts MiniCad layers into AutoCAD layers. This is the most straightforward conversion process.

You also can use MiniCad classes, which can actually mimic the AutoCAD layer structure more accurately. The pen foreground and background also can be used to generate the AutoCAD layers.

Export 2D Objects Only/Export 2d & 3d Objects

Choose one of these options. All AutoCAD drawings contain 3D information. However, MiniCad can create objects that are only 2D.

If a drawing contains objects drawn exclusively with the 2D drawing tools, you can choose to export as 2D only. The resulting file is significantly smaller.

Select 2D & 3D to export the complete file geometry.

Decompose 3D Symbols into 3D Polys

Some programs that import DXF block information do not handle MiniCad symbols correctly. Selecting this checkbox converts all 3D symbols into grouped 3D polygons. The object looks like the symbol, but imports correctly. The object does not convert into a DXF block.

MiniCad files can be exported as text. All objects and attributes are exported with the file.

MiniCad text files can be imported into any open MiniCad file. This can be used to repair a damaged file. It also can be used to import files from older versions of MiniCad.

Text exports also can be used to create start-up files. The use of start-up files is an obsolete way to change the defaults on a new file when it is opened. This method has been replaced by stationery files in newer versions of MiniCad. Start-up files are retained in MiniCad to maintain compatibility with older versions of the program.

The MiniCad worksheets can be exported in comma delimited, tab delimited, merge, DIF, or SYLK formats.

The files exported in any of these formats can be read by other programs such as Microsoft's Excel or Claris' FileMaker. Files saved in tab delimited format can be opened as a table in Microsoft Word.

All or part of a worksheet can be exported. Note that the different export formats support different aspects of the original worksheet. Generally, exporting the values in a file rather than the formulas has more predictable results. It is a good idea to experiment with the different formats and check the results in the target program.

To export the contents of a worksheet, you must open the worksheet first. If you want to export just part of a worksheet, select the rows you wish to export. If multiple worksheets are open, make the worksheet to be exported the active worksheet.

Next, choose **Worksheet** from the submenu of the **Export** command. The **Export Worksheet** dialog box opens.

This dialog box has the following additional options:

All rows/Selected Rows Only

Select one of these choices. If only part of the worksheet is exported, you must select the rows before the **Export** command is chosen.

Format

Select one of the formats from the drop-down list.

Comma

Each cell in the exported worksheet is separated with a comma. The file is exported by rows.

Tab

Each cell in the exported worksheet is separated with a tab character. The file is exported by rows. Exporting worksheets as tab delimited files usually gives the best results.

Merge

Used mainly in text-processing programs to produce mail merge documents. It is quite useful for placing different information in the same location in multiple documents.

DIF

Used for placing worksheet information in database programs.

SYLK

Used to export worksheet formulas, not just the results. However, since the syntax used to build the formulas in a spreadsheet differs from program to program, check the resulting file to be sure it contains the proper values when the formulas are calculated.

Database records and fields can be exported. To export a database record or field, choose **Export** from the **File** menu and **Database** on the submenu. The exported file can be read by other programs such as Microsoft's Excel, or Claris' FileMaker.

When you select the **Database** command, a dialog box opens. Select a record from the scrolling list. Next, select the fields you want to export. Upon clicking **OK**, you are asked to name the exported database and choose a location to store it.

Choose the desired format from the **Format** menu:

Comma
When a database worksheet is exported, each subrow cell in the worksheet is separated with a comma. The file is exported by subrows.

Tab
When a database worksheet is exported, each subrow cell in the worksheet is separated with a tab stop. The file is exported by subrows.

Merge
Used mainly in text processing programs to produce mail merge documents. It is quite useful for placing different information in the same location in multiple documents.

DIF
Used for placing database worksheet information in standard database programs.

SYLK
Used for placing database worksheet information in standard spreadsheet programs.

Extrude is the process that gives a *z* dimension to a 2D object, turning a 2D object into a 3D object. Objects are extruded with the **Extrude** command on the **3D** menu.

Note: The default *z* for an extruded object is the active layer's Δz. If you want to set a different *z* than this, use the **Extrude...** command.

If an object with a fill is extruded, it extrudes as a wireframe solid. (The fill is not displayed until the object is rendered.)

When extruding, the *z* axis is always the extruded axis, and that axis is determined by the current view. For example, if you extrude in **Top/Plan** view, the *z* is the "height" of an object. But, if you extrude from the **Front** view, the *z* is the "depth" of the object.

The following objects can be extruded:

- Rectangles (including rounded rectangles)
- ovals/circles
- arcs and quarter arcs
- polygons
- freehand lines
- polylines.

Groups cannot be extruded. However, multiple selections can. When extruded, multiple selections become a group of 3D objects. They can be ungrouped with the **Ungroup** command and edited separately.

In addition, groups can be entered with the **Edit Group** command and edited without ungrouping. The only way text can be extruded is by first converting it to polylines with the **Truetype to Polyline** command.

A line is extruded as a flat plane. All other objects are extruded as wireframe solids.

To extrude an object, select it and choose the **Extrude** command.

To reshape an extrusion, select it and open the **Object Info Palette**. Enter the new values for the Δx, Δy and Δz and click **OK**.

You also can reshape an extruded object with the **3D Reshape** tool on the **3D Tool** palette.

For related information, see the following topics:

Multiple Extrude
Extruded Polygon
Extruded Rectangle
Sweep

The **Extrude...** command is identical to the **Extrude** command except that instead of using the layer Δz for the extrusion it opens a dialog box where the extrusion is set.

Note: The **Extrude...** command is located on the alternate **3D** menu. This menu is activated by, pressing the **Control** key while selecting the **3D** menu from the menu bar.

The **Fit to Objects** command has two forms. It will perform differently depending on whether an object is selected or not.

- **If no objects are selected on the drawing,** the window zooms-out to show all objects on the drawing, even if they are outside the print boundary. This differs from the **Fit to Window** command which zooms-out to show only those objects that fall within the print boundary. The print boundary is defined by the gray border shown on screen; all objects inside the boundary are printed.
- **If an object is selected on the drawing,** the window zooms-in to show just the selected object.

For related information, see the following topics:

Zoom Out
Zoom In
Fit to Window
Normal Scale

The **Fit to Window** command places the entire drawing, even multiple pages, within the drawing window of the screen. The drawing is displayed with the center of the page(s) at the center of the screen. The **Normal Scale** command or the **Zoom In** tool can then be used to view a detail within the drawing.

You can move quickly from one section of a large drawing to another by selecting **Fit to Window**, selecting an object in another part of the drawing, and choosing **Normal Scale**.

For related information, see the following topics:

Zoom Out
Fit to Objects
Normal Scale

The **Floor** command on the **AEC** menu, greatly simplifies the creation of floors and decks. For example, if the specifications for the floors in a floorplan call for the use of 2 x 10's with a subfloor of 3/4" plywood, the thickness of the floor is then set to 10 3/4" and all subsequent floors have a thickness of 10 3/4" (until the value in the **Thickness** field is changed). This reduces the number of calculations you must perform as you draw.

Note: This tool is only available when the **AEC Overlay** is active. If the **AEC** menu is not listed on the menu bar, select **AEC Overlays** using the **Overlays** command on the **File** menu.

The **Floor** command produces a hybrid 2D/3D object from a 2D original. A floor can be of any shape or thickness, but it must be flat. A floor must always be created parallel to the ground plane.

The **Floor** command also simplifies the creation of stairwells, lightwells, and balconies.

For additional information, see the following topics:

Making Floors
Editing Floors
Calculating Floor Area and Perimeter

To create a floor, draw the primitive from which the floor is created, using any combination of the 2D surface tools (e.g., Rectangle, Ellipse, or Polygon). Use the **Add Surface** command to combine different objects into a single object.

Holes for a chimney, lightwell, or stairwell can be cut in the primitive using the **Clipping** tool. Holes also can be made by drawing an object the size of the desired hole and positioning it on the primitive. Select both objects and choose the **Clip Surface** command.

When the primitive is finished, select it and choose the **Floor** command on the **AEC** menu. The **Create Floor** dialog box opens. There are two fields in the dialog box, one for the floor's *z* height and the other for its thickness.

Bottom Z

The **Bottom Z** dimension of a floor is the distance from the bottom of the floor to the bottom of the layer. If the floor is in its own layer, enter 0 in the **Bottom Z** field.

Generally, a floor should be placed in its own layer. If a floor is in the same layer as the walls that surround it, door and window symbols must be adjusted for the floor's thickness. And, since floors vary in depth, symbols cannot be placed at a uniform *z* height.

Thickness

The **Thickness** field is used to specify the total dimension (joists, subflooring, and flooring) for the floor.

For related information, see the following topics

Editing Floors

Calculating Floor Area and Perimeter

A floor is a group of objects. Thus, the **Edit Group** command must be used to edit a floor. After the floor has been entered, the original 2D primitive is displayed in the edit window.

The **Add Surface** command can be used to combine new 2D components to the primitive. Holes for a chimney, lightwell, or stairwell can be cut in the primitive using the **Clip** tool.

When finished with the edits, click on the **Exit Group** button in the mode bar or choose the **Exit Group** command in the **Organize** menu. The floor reappears with all of the changes.

For related information, see the following topics:

Floor
Making Floors
Calculating Floor Area and Perimeter

The area and perimeter can easily be calculated for the floor. One method is to select the floor, and choose the **Edit Group** command. The area and perimeter of the primitive will be displayed in the **Object Info Palette**.

Another method of determining area and perimeter is to create a worksheet. The area can be calculated using the AREA function on the SLAB object type.

For related information, see the following topics:

Floor

Making Floors

Editing Floors

The **Font** command on the **Text** menu displays a list of all fonts installed in the system. The currently selected font is identified with a checkmark next to its name.

A single text block can have multiple fonts. To set the default font, deselect the **Text** tool and choose a font from the **Fonts** submenu.

To change the font of a string of characters within a text block, select the characters with the **Text** tool, then choose a font with the **Font** command.

Text is drawn in the current pen color. To make text in a different color than the current color, change the pen foreground color in the **Attribute Palette**.

A block of text also has a fill color. To insert the block of text onto an object that has a fill, change the text block background to match the object's background color.

The text within a text block can be many different colors, but the background must be a single color.

For related information, see the following topics:

Size
Style
Text
Editing Text

The **Size** command on the **Text** menu, opens a submenu of available point sizes for the fonts installed in the system. A single text block can have multiple font sizes.

Outline fonts, like Type 1 PostScript or TrueType, are available in all point sizes. Bitmapped fonts come in a limited number of supported sizes. Only the supported sizes will display correctly. The supported sizes are displayed in the submenu in outline form.

The **Set Size** command opens a dialog box that allows you to enter a font size that is not on the submenu.

To set the default font size, deselect the **Text** tool and select a size with the **Size** command.

To change the font size of a string of characters within a text block, select the characters with the **Text** tool, then choose a size with the **Size** command.

To change the font size of an entire text block, select the block and select a size with the **Size** command.

For related information, see the following topics:

Font
Style
Text
Editing Text

A text block can have multiple font styles (e.g., italic, bold), which can be selected using the **Style** command on the **Text** menu.

To set the default font style, deselect the **Text** tool and choose a font from the **Style** submenu.

To change the font style of a string of characters within a text block, select the characters with the **Text** tool, then choose a style with the **Style** command.

For related information, see the following topics:

Size
Font
Text
Editing Text

The **Grid Creator** command is part of the **Cross Stitch** toolkit. It is designed to work with the **Symbol Paint** tool and the **Cross Stitch** stationery supplied with the **Cross Stitch** toolkit. This toolkit simplifies the creation of cross stitch patterns.

The **Grid Creator** command creates a tenth-of-an-inch spaced grid with the number of squares specified. The grid is made of lines.

Note: This is not the grid used to align objects in a standard drawing. That grid is created and controlled with the **Set Grid** command.

The snap grid must be active, with the current snap set to one-tenth of an inch, for the symbols to fill the grid correctly. The **Symbol Paint** tool, which is used to apply a pattern to the cross stitch grid, automatically scales symbols to the current snap grid size.

Sometimes it is convenient to treat two or more objects as a single object. The **Group** command on the **Organize** menu takes multiple selected objects and turns them into a group.

As a group, the multiple objects act as a single object with a single set of selection handles. Any combination of objects, symbols and text can be grouped.

Selected groups can be combined into another group. Although a group has a single set of selection handles, objects within the group retain their original snap points.

Groups are returned to single objects with the **Ungroup** command on the **Organize** menu. Multiple **Ungroup** commands are needed to completely ungroup nested groups.

Objects within groups can be edited without ungrouping using the **Edit Group** command.

A group is created by selecting all objects to be grouped. Objects may be selected using a marquee, **Control**-**Marquee**, clicking, or **Shift**-**Clicking**. After the objects are selected, choose the **Group** command to group them.

Guides are objects used to align different components of a drawing. Guides work with the snap constraints found on the **Constraint** palette. They also can be used for visual alignment.

There are five different **Guides** commands used to create, display, select, or remove guide lines from the drawing:

For additional information, see the following topics:

Make Guides
Select Guides
Show guides
Hide Guides
Delete All Guides

The **Make Guides** command on the **<u>Guides</u>** command's submenu, creates a guide object from any selected object on the drawing.

Any object on the drawing can be converted into a guide. By default, the guide object is the same color as the reference grid.

The appearance of a guide or grid is determined by the monitor. On a color monitor, they are light blue, whereas on a black and white monitor, they appear as a dotted line.

The **Select Guides** command on the **<u>Guides</u>** command's submenu selects all guides on the drawing. When selected, the guide's color can be changed by changing the pen color on the **Attribute** palette.

The **Show Guides** command on the **Guides** command's submenu displays all guide objects on the drawing.

The **Hide Guides** command on the **Guides** command's submenu suppresses all guide objects on the drawing. This can help reduce screen clutter.

The **Delete All Guides** command on the **Guides** command's submenu deletes all guide objects on the drawing.

The **Hatch** command on the **Tool** menu applies a predefined line pattern to a surface. Hatch patterns are traditionally used to represent specific materials or to show the profile of a section.

MiniCad includes numerous library files which contain predefined hatch patterns. Additional, custom hatch patterns can be created. Hatches are imported and edited with the **Resource Palette**, but applied to objects with the **Hatch** command.

Hatches are created as a pattern of lines. Because the hatch is comprised of lines, the lines are translated correctly when the file is exported in DXF format. This is an important advantage over standard fill patterns which cannot be translated in DXF. This is also an important consideration if the file will be printed at a service bureau that uses DXF format.

Hatches can also be created in paper scale, to make them appear appropriate in a scaled drawing. This is another major difference with fill patterns, which are always in world scale.

To apply a hatch pattern, select a filled object on the drawing. Choose the **Hatch** command.

In the **Select Hatch** dialog box, select a hatch pattern from the scrolling list. Click **Apply**. The dialog box closes and the cursor changes to a paint can.

Place the cursor within the object to be hatched and click. The hatch draws within the filled object. If multiple objects are selected, the hatch is applied to the specific object in which the cursor is placed.

Hatch patterns are created as separate objects from the objects to which they are applied. Because the hatch and the object are separate, you may want to use the **Group** command to combine them. That way the hatch and the object can be moved as a unit.

If the object and the hatch are grouped, resizing the group can distort the hatch. To correct this distortion, ungroup the object, delete the hatch and reapply it.

For additional information, see the following topics:

The Select Hatch Dialog Box

The **Select Hatch** dialog box is used to select and edit a hatch style. It has the following options:

Hatches

All hatches contained in the active file are listed in the **Hatches** scrolling list. The selected hatch is displayed in the window. Hatches to be applied to objects on the drawing are selected from this list. Hatches are also selected from this list before they are edited, removed, or duplicated.

Remove

Removes the selected hatch from the active file. A dialog box asks you to confirm the deletion.

Duplicate

Duplicates the selected hatch. Duplicating a hatch is useful for making a new hatch similar to an existing one.

Cancel

Closes the dialog box without making any changes.

Edit

Opens the selected hatch in the **Edit Hatch** dialog box. The hatch is displayed in the window on the dialog box. Existing hatches can be renamed by selecting the hatch, clicking the **Edit** button, and typing a new name in the **Name** field.

For more information about the **Edit Hatch** dialog box, see the **Hatches** topic under the **Resource Palette**.

New

Opens the **Edit Hatch** dialog box so that a new hatch pattern can be created. For more information about the **Edit Hatch** dialog box, see the **Hatches** topic under the **Resource Palette**.

Apply

When clicked, the selected hatch in the scrolling list becomes the active hatch. When the **Apply** button is clicked (or the **Enter** key is pressed), the dialog box closes and the cursor changes to a paint bucket.

When the cursor is placed inside the selected object, and the mouse is clicked, the active hatch is applied.

The **Hex Socket Screw** command on the **Mechanical** command's submenu is used to draw the top or side view of a socket cap screw.

Note: This tool is only available when the **Mechanical Overlay** is active. If the **AEC** menu is not listed on the menu bar, select **Mechanical Overlays** using the **Overlays** command on the **File** menu.

The parameters are specified in the **Create Hex Socket Screw** dialog box. It contains the following options:

Head styles
Head styles include cap head, button head, and flat head.

Nom size
Nominal sizes, which are read from data files supplied with the external, are selected from the **Nom Size** pop-up menu.

The data files can be edited in any word processor, allowing custom sizes to be added by the user. The data files for drawing the screws are located in a folder named "External Data," which is in the Externals folder.

Screw
The length of the screw is entered in the **Screw Length** field.

Detail
The threads can be drawn in detail, either Unified National Fine (UNF) or Coarse (UNC), or as dotted lines (No Detail).

In addition there are buttons to select the top or side view of the screw, and buttons to determine whether English or metric measurement is used.

For related information, see the following topics:

Machine Screw

Files can be imported into MiniCad in a number of ways. Before importing a file, determine its file type to identify which import method to use.

If MiniCad does not support the format for the file to be imported, you may be able to do a "Save As" within the original program and save it into a format MiniCad can read.

The following import formats are supported:

Import PICT
Import PICT as picture
Import EPSF
Import DXF
Import Text Format
Import worksheet

Used to import PICT files into MiniCad.

PICT as Picture imports with full resolution. A **PICT as Picture** import is a single graphic object. The only editing possible to an imported **PICT as picture** is resizing. No other editing of the object is possible. Greyscale and color images (including 24 bit) retain their quality when they are imported as **PICT as Picture**.

MiniCad imports Encapsulated PostScript Format. EPSF is a high resolution format. To import a file, select the file from the directory dialog box which opens when you choose this command.

The **Data Exchange Format** (DXF) is the de facto standard in the DOS desktop CAD market. MiniCad reads AutoCAD release 12.

Both 2D and 3D entities can be imported through DXF. If 3D entities are detected in the file to be imported, two options are given. One will import 3D objects as 2D, the other imports as 3D.

When importing a DXF file, always start with a blank, unformatted document. If you do not start with a blank unformatted document, the translation may fail (you can set attributes and preferences after the file is successfully imported).

Set the drawing scale to 1:1 prior to import. Units settings need to be made prior to the import process; i.e., if the file that the DXF information originated from was done in meters, the units settings need to be set to meters. If the translator detects a difference in units, a dialog box opens to notify you.

Minor text repositioning can be required after a DXF import because of the inherent differences in the way DOS files handle text.

MiniCad documents that have been exported in text format can be imported into any open document. MiniPascal macro files also can be imported as text.

Text format is not to be confused with text created with a word processing program. Text created in a word processing program can be copied to the Clipboard, and from there, pasted into a MiniCad text block.

MiniCad can import worksheet information that is saved as comma delimited, tab delimited, DIF, or SYLK format. A worksheet must be open, and active, to import the data into the active MiniCad document.

The **Intersect Surfac**e command on the **Tool** menu creates a new object from the overlap (i.e., intersection) of two selected objects.

Any combination of the following objects can be intersected:

- Rectangles
- Rounded rectangles
- Ellipses/circles
- Quarter arcs
- Full arcs
- All types of polygons (regular, freehand, and polylines).

Any object that can be filled and that isn't in a group can be intersected.

When two surfaces are intersected, the object in front is intersected with the object in back. If the objects have fill patterns, the new object has the fill pattern of the rearmost object.

If the objects are open, they are converted to closed polygons before they are intersected.

To intersect objects, arrange them in the desired order. Remember that MiniCad creates a new polygon from the overlapping parts of the objects, and that each object intersects the one below it. If necessary, use the **Send** command to place the objects in the proper order. Select all objects then choose **Intersect Surface**.

For related information, see the following topics:

Clip Surface
Clip
Combine into Surface
Add Surface

The **Join** command on the **Tool** menu takes two segments of a nonparallel line and joins them together. Both single and double lines can be joined.

Walls also can be joined, as can combinations of walls and double lines. The **Join** command extends the lines as required to form the join.

To join two segments of a single line, select both segments and choose the **Join** command.

The result of the join operation depends on the segments selected:

- If two segments of a line join to form a 'T', one segment of the longer line is trimmed away.
- If the lines overlap, the **Join** command trims the excess.

Double lines are joined with some different considerations:

- In order to form a 'T' intersection of two double lines, the two sets of double lines cannot intersect.
- In order to form a cross intersection, the two sets of lines must intersect.
- In order to form a corner with a double line, the outer lines must not intersect, otherwise a "T" is formed.

For related information, see the following topics:

Join and Fillet
Join and Trim
Wall
Butt-Join Wall
Y-Join Wall

The **Join and Fillet** command is used to join selected double lines with a filleted intersection.

Note: The **Join and Fillet** command is located on the alternate **Tool** menu. This menu is activated by, pressing the **Control** key while selecting the **Tool** menu from the menu bar. You must also have a double line selected when you select the **Tool** menu, otherwise the **Join and Fillet** command will not appear.

If a fillet radius has not already been defined, the **Fillet Radius** dialog box opens. Enter the radius in the **Fillet Radius** field and click **OK**.

For related information, see the following topics:

Join
Join and Trim

The **Join and Trim** command is used to join and trim selected single lines.

Note: The **Join and Trim** command is located on the alternate **Tool** menu. This menu is activated by, pressing the **Control** key while selecting the **Tool** menu from the menu bar. You must also have a single line selected when you select the **Tool** menu, otherwise the **Join and Trim** command will not appear.

For related information, see the following topics:

Join
Join and Fillet

Layers are controlled with the **Layers** command on the **Organize** menu or with the **Layer** pop-up menu on the **Data Display** bar.

A description of MiniCad's layers, and their uses can be found under **General Information about Layers**.

When the **Layers** command is selected, the **Layers** dialog box is selected. This dialog box is used to create new layers and edit existing ones.

It contains the following options:

Note: the **Layers Setup** control keys can be used to select many of these options. See the **Layers Setup Control Keys** for a list of these keys.

Name
Displays the name of the selected layer in the scrolling list. A layer can be renamed by selecting the desired layer and typing a new name in the field. Each layer must have a unique name.

New
Adds new layers to the drawing. You are asked to name the new layer; Layer-*n* is entered as a default. Each layer in a drawing must have a unique name.

Remove
Removes the selected layer from the drawing. All objects on the layer are removed from the drawing. You are warned before the command takes effect because this action cannot be undone.

Top
Sends the selected layer to the top of the scrolling list. This affects display and print order of layers in 2D/Plan. Scrolling order has no effect on the *z* dimension of the layers.

Up
Moves the selected layer one layer up the scrolling order. This affects display and print order of layers in 2D/Plan.

Down
Moves the selected layer one layer down the scrolling order. This affects display and print order of layers in 2D/Plan.

Bottom
Sends the selected layer to the bottom of the scrolling order. This affects display and print order of layers in 2D/Plan.

Active layer
Makes the selected layer the active layer and displays it. The active layer is always visible, regardless of its visibility setting.

Normal
Makes the selected layer visible. A solid diamond is displayed in the scrolling list. Visible layers are printed and can be edited from other layers, if desired.

Grayed
Displays 2D layers as a grayed outline when viewed from other layers. A hollow diamond is displayed in the scrolling list. Text and fills are invisible in grayed layers. A grayed layer prints as displayed. 3D layers do not display if they are grayed.

Invisible
Makes the selected layer invisible when viewed from other layers.

Transfer mode

This pop-up menu controls visual attributes that apply to objects drawn on that layer. These settings are most useful on black and white monitors. Transfer modes are unpredictable when used on color monitors. Moreover, except for paint and overlay, these settings are not supported on PostScript printers or on plotters.

For details about this menu, see **The Transfer Mode Pop-up Menu**.

Scale

The **Scale** option is used to assign a scale to the layer. While a drawing can include objects drawn in different scales, each layer can have only one scale. Therefore, in one drawing (but all in different layers), a title block can be in 1:1, a site plan in 1:50, a floor plan in 1/4" and details in 1/2".

When you select the **Scale** option, the **Drawing Scale** dialog box appears. It offers architectural, engineering and enlargement scales.

The value in the **Paper Scale** field will change to reflect changes in the scale. You can type a value directly into the field, if desired.

If you want to set a default scale for all layers in the drawing, select the **All Layers** checkbox. Care must be taken when using this option, as it overrides any settings made in layers setup.

If you want text to be scaled with the drawing, select the **Scale Text** option.

Colors

The **Colors** option allows you to select a color, for both pen and fill, that applies to the entire layer. Use of layer color is a convenient way to differentiate elements on a complex drawing.

Any settings made with the **Colors** option are visible only if the **Use Layer Colors** checkbox is selected in the **Preferences** dialog box. Layer color supersedes any object color attributes. The object attributes remain, but display only when the **Use Layer Colors** checkbox is deselected.

Z

The Layer *z* setting is the distance to the ground plane from the selected layer. This distance is important when using the **Layer Link** command to view multiple layers together.

ΔZ

The **DZ** setting is the default height for 3D objects created in the selected layer. For example, if an object is extruded in a layer with a 8' *z*, the object's *z* value is 8'.

For related information, see the following topics:

Layer Options
Create Layer Link

Paint
Makes objects solid and obscures objects in layers below.

Overlay
Makes objects transparent so they do not obscure objects in layers below.

Invert
Creates a photo negative image on areas where objects overlap each other on different layers.

Erase
Gives objects a transparent effect, but with all foreground patterns displayed as white and all background patterns displayed as "None" (transparent).

Not Paint
Displays the objects as solids and inverts the colors on areas where objects overlap on different layers.

Not Overlay
Gives objects a transparent effect and inverts layer colors

Not Invert
Gives objects a transparent effect and makes overlapping black pixels from other layers turn white and the white pixels transparent.

Not Erase
Gives objects a transparent effect and makes overlapping white pixels from other layers turn black and the black pixels transparent.

The **Layer Setup** dialog box has a unique set of keyboard commands that can be used to select options for the active layer in the **Layers Setup** dialog box. The **Layers Setup** dialog box must be open when these commands are issued.

Layers Control Keys	
Control-A	Activate layer
Control-B	Send layer to bottom level
Control-D	Send layer down one level
Control-G	Gray out layer
Control-I	Make layer invisible
Control-K	Color
Control-M	Normal visibility
Control-N	Create new layer
Control-R	Remove layer
Control-S	Set scale
Control-T	Send layer to top level
Control-U	Send layer up one level

Layers and **Classes** are two different ways MiniCad gives you to organize the elements in your drawing.

In standard 2D CAD, layers are similar to a stack of transparent mylar, where one element is superimposed over another. Layers in 2D CAD have a stacking order, which determines the order in which the layers are displayed and printed.

In MiniCad, layers can have different scales, projections, rendering, and visibility. Any layer in the drawing can be grayed, invisible, or visible.

Because MiniCad is a 3D program, layers have a *z* dimension in addition to the traditional *x* and *y* dimensions. The *z* dimension is measured from the ground plane, and can be either a positive or negative number. This way layers can occupy the same place in space, stacked one atop the other, or be arranged in any order desired.

When working in 3D, layers can be dynamically linked to each other. Layers must be linked before they can be viewed or rotated together. Any number of layers can be linked and edits to one layer are reflected in the linked layer with each screen redraw.

A single layer can be linked repeatedly, giving multiple views of the same layer. Layers are linked with the **Create Layer Link** command on the **3D** menu.

For related information, see the following topics:

Layers
Layer Options
Create Layer Link

The **Layer Options** command on the **Organize** menu controls how layers interact with each other, and how they are seen. The following choices can be made from the **Layer Options** submenu.

Active Only

When selected, only the active layer is displayed. All other layers are hidden. If the file is printed, only the active layer is printed. (MiniCad always prints what is displayed on the screen.)

Gray Others

When selected, all visible 2D layers are grayed, and the active layer is displayed normally. Only the active layer can be edited. 3D layers cannot be grayed.

Show Others

All layers are displayed normally, but only the active layer can be edited.

Show /Snap Others

All layers are displayed normally, but only the active layer can be edited. The active layer snaps to objects on other layers if they are in the same scale and projection. You cannot snap between 3D layers.

Show /Snap/Modify Others

All layers are displayed normally and all visible layers can be edited if they are in the same scale and projection. The active layer snaps to objects on other layers. You cannot snap between 3D layers.

All selection handles in the active layer are black. Selection handles on other layers are hollow and locked objects have grayed handles.

You can snap between layers only if the layers are in the same scale and projection.

For related information, see the following topics:

Create Layer Link
Layers
Layer Options

A text block can be linked to a record that is attached to a symbol. The text appears with the symbol when the symbol is placed. The text can be edited in the **Data** pane of the **Object Info Palette** or with the **Record Format** command.

The **Link Text to Record** command on the **Organize** menu can be used to identify individual symbols. Even though all symbols are identical, linked text allows a unique number to be attached to a symbol on the drawing. A database can be then configured to list of the symbols with their unique numbers.

To link text to a symbol, select 1) a symbol placed on the drawing and, 2) a text block. Choose **Link Text To Records**.

From the **Choose Field** dialog box, select a format to open and a record to attach to the symbol. All instances of the symbol placed thereafter will have the text block attached.

The text block's position relative to the symbol can be edited with the **Edit 2D** button on the **Symbol** dialog box.

To edit the text in the symbol, select the symbol. Open the **Object Info** palette, make the **Data** pane active and select the record to be edited. The text can be edited directly in the edit field.

MiniCad allows you to protect objects from accidental movement, deletion, or editing with the **Lock** command on the **Edit** menu.

Any object that is selected when the **Lock** command is chosen becomes locked.

A locked object has grayed selection handles. A locked object cannot be edited, moved, or deleted.

If a worksheet placed as a graphic on a drawing is locked, it is updated if the underlying worksheet is updated. However, it cannot be moved, deleted, or edited in any other way.

The **Unlock** command on the **Edit** menu can be used to unlock a selected object.

The **Machine Screw** command draws the top or side view of a socket cap screw. This command is located on the **Mechanical** command's submenu on the **Toolkit** menu.

Note: This tool is only available when the **Mechanical Overlay** is active. If the **Mechanical** menu is not listed on the menu bar, select **Mechanical Overlays** using the **Overlays** command on the **File** menu.

The parameters used to create the screw are specified in the **Create Socket Cap Screw** dialog box. It contains the following options:

Head Styles
Head styles include cap head, button head, and flat head.

Nom Size
Nominal sizes, which are read from data files supplied with the external, are selected from the **Nom Size** pop-up menu.

The data files can be edited in any word processor, allowing custom sizes to be added by the user. The data files for drawing the screws are located in a folder named "Data," which in turn was placed in the External folder when MiniCad was installed.

Screw Length
The length of the screw is entered in the **Screw Length** field.

Detail Buttons
The threads can be drawn in detail, either Unified National Fine (UNF) or Coarse (UNC), or as dotted lines (No Detail).

For related information, see the following topics:

Hex Socket Screw

The **Move** command allows for precise movement of an object. This can be quite useful whenever precision is required to move an object or group.

You can use either Cartesian or polar coordinates to move your object. Cartesian moves an object in its *x* and *y* axis; polar moves it in terms of distance and angle.

To use this command, select an object, group, or multiple selection and then choose the **Move** command from the **Tool** menu. Enter the value in the appropriate field for the distance you want the object moved and click **OK**.

For related information, see the following topics:

Nudge
Move 3D
Move Working Plane

The **Move 3D** command on the **Tool** menu allows for precise movement of an object in the *x*, *y*, and *z* axes. This can be useful whenever precision is required to move an object or group through 3D space.

If the working plane has been changed, the *i*, *j* and *k* axes control the movement of the object.

To use this command, select an object, group, or multiple selection and select the choose the **Move 3D** command.

Enter the value in the appropriate field for the distance you want the object moved and click **OK**.

For related information, see the following topics:

Move
Move Working Plane

The **Move Working Plane** command on the **3D** menu is the most accurate way to move a working plane.

When this command is chosen, the **Move Working Plane** dialog box opens. This dialog box allows the choice between the *x, y, z* or the *i, j, k* axes, and has fields for specifying the movement.

The working plane is a user definable plane used while working in 3D. Multiple working planes can be configured in a MiniCad drawing. Working planes are created with the **Working Plane** palette.

For related information, see the following topics:

Working Plane Palette
Move 3D

The **Multiple Extrude** command allows multiple objects to be extruded in a linear, stepladder fashion. The **Multiple Extrude** command allows pyramids, spheres and similar objects to be created.

When objects are extruded, the first object created is located at the bottom of the stacking order. This object is extruded to the second object, which is extruded to the third object, and so forth. This continues until the last object is extruded.

The distance between objects is constant throughout the extrude. The *x* and *y* dimensions can change for each object in the stack, which allows the extrude to flare in any direction. A locus can be used to bring the extruded object to a point.

Note: The default *z* for an extruded object is the active layer's Δ*z*. If you want to set a different *z* than this, use the **Multiple Extrude...** command.

The result of a **Multiple Extrude** is a grouped, 3D object. The height of the object can be changed by selecting it and entering a new Δ*z* for the object in the **Object Info Palette**.

If you choose the **Edit Group** command on the **Organize** menu with a selected multiple extrude object, the original 2D objects are displayed in the edit window. Edits to the original objects modify the extrusion.

If the multiple extrusion is converted to a mesh, the individual vertices of the object can be edited.

For related information, see the following topics:

Extrude
Extruded Polygon
Extruded Rectangle

The **Multiple Extrude...** command allows multiple objects to be extruded in a linear, stepladder fashion to a height specified in a dialog box.

Note: The **Multiple Extrude** command is located on the alternate **3D** menu. This menu is activated by, pressing the **Shift** and **Control** keys while selecting the **3D** menu from the menu bar.

The **Multiple Extrude...** command works identically to the standard **Multiple Extrude** command, except that the height of the object is determined at creation using a dialog box instead of using the current layer's Δz setting.

When the **Multiple Extrude...** command is chosen, the **Create Extrude** dialog box opens. The Δx and Δy values for the selected objects are displayed.

Enter the desired Δz in the **Extrusion** field and click **OK**.

For related information, see the following topics:

Extrude
Extruded Polygon
Extruded Rectangle

The **New** command on the **File** menu creates a new MiniCad document. When you choose the **New** command, a new, untitled document opens.

The first new document is titled "Untitled *n.*"

Up to eight MiniCad documents can be opened simultaneously. Each new document opens on top of the previous document.

The file opens with the standard defaults. A specific set of defaults, such as scale or units, can be created to preconfigure a new file.

This is done by saving the desired settings with the **Save as Stationery** command. The file must be saved with the name "Default.sta" and placed in the same folder as the MiniCad application.

All new documents will have the stationery's defaults.

For related information, see the following topics:

Open

To view or edit a drawing at its print size, choose **Normal Scale** on the **Page** menu.

For example, in 1:1 scale, one square inch of screen area displays a one-inch square of the drawing and prints that one-inch square.

In 1:48 scale, one-inch of screen area displays a forty-eight inch square and prints a one inch square representing a forty-eight inch square.

If an object is selected when the **Normal Scale** command is chosen, it is placed in the center of the drawing window.

Since the actual size of the paper can be larger than the screen, it may necessary to pan the screen display to view the entire drawing. This can be done with the **Pan** tool, the scroll bars, or by auto scrolling.

The **Fit to Window** command also is useful when used in conjunction with the **Normal Scale** command.

For related information, see the following topics:

Zoom Out
Zoom In
Fit to Window
Fit to Objects

The **Nudge** command on the **Tools** menu allows the arrow keys to be used to move an object in one-pixel increments.

Normally the arrow keys move the view of the drawing window one-half screen each time the key is pressed. However, when an object is selected and the **Nudge** command is chosen, the arrow keys will move the object. Each time an arrow key is pressed, the object moves one pixel in the direction of the arrow.

The command remains active until the object is deselected.

There are 72 pixels to an inch. A pixel is an absolute distance and is affected by zoom. For example, if the drawing is zoomed out to display the entire page, moving the object by a single pixel moves the object a greater distance than when the drawing is zoomed in to show just the selected object.

For related information, see the following topics:

Move

The **Object Info Palette** opens when the **Object Info** command is chosen from the **Window** menu.

The **Object Info** palette changes as different objects are selected on the drawing. The palette may have to be resized to see all available buttons and fields. The palette is closed by clicking the close box. Like all palettes, the **Object Info** palette can be moved around the screen.

The **Open** command on the **File** menu is used to open an existing MiniCad document.

The **Open** command presents a dialog box that lists all the available files on the current disk. If you click the folder name above the scrolling list, the directory pop-up menu shows the path from the current folder to the hard disk.

Up to eight MiniCad documents can be open simultaneously.

For related information, see the following topics:

New

MiniCad's tools and commands are organized into industry specific combinations (e.g., architectural, mechanical). These different combinations of menus and tool palettes are called **Overlays**.

Overlays are selected from the **Overlay** submenu on the **File** menu. The default overlay is the MiniCad **Standard** Overlay. To change an overlay, choose the desired overlay from the **Overlay** submenu.

The **Paste** command on the **Edit** menu places a copy of the Clipboard's contents onto the active layer of the current drawing.

Normally a pasted object is centered on the drawing window. Use the **Paste in Place** command to keep the pasted objects aligned or in registration.

The Clipboard's contents remain unchanged after a paste, so you can use this command repeatedly to make multiple copies of an item.

You also can use the **Duplicate** or **Duplicate Array** command to make multiple copies of an object.

For related information, see the following topics:

Paste as Picture
Paste in Place
Copy
Cut

The **Paste as Picture** command on the **Edit** menu places a copy of the Clipboard's contents onto the active layer of the current drawing as a picture.

As a picture, the pasted objects cannot be edited. They will draw much faster, and they retain 24 bit color information and PostScript comments when pasted as a picture.

The Clipboard's contents remain unchanged after a paste, so you can use this command repeatedly to make multiple copies of an item.

For related information, see the following topics:

Paste
Paste in Place

The **Paste in Place** command places a copy of the Clipboard's contents on the active layer of the current drawing, at the same *x*, *y* and *z* coordinates as the original object.

The Clipboard's contents remain unchanged after a paste, so you can use this command repeatedly to make multiple copies of an item.

For related information, see the following topics:

Paste

Paste as Picture

Copy

Cut

The **Perspective** submenu on the **3D** menu controls the amount of distortion used to give the impression of perspective in a 3D drawing. The settings you select are retained for the remainder of the current work session.

A 3D drawing is put in perspective with the **Projection** command on the **3D** menu. In addition, some tools, such as the **Walkthrough** tool, automatically put the drawing in perspective.

The three perspective options on the submenu are used to change the perspective view:

- **Narrow Perspective**
- **Normal Perspective**
- **Wide Perspective**

The **Set Perspective** command opens the **Set Perspective** dialog box for the direct entry of the desired perspective value. This command allows full control over the degree of perspective than is allowed with the default choices. The higher the number entered in the **Perspective** field, the narrower the degree of perspective.

MiniCad maintains a preferences file in which certain default settings are set (e.g., conversion resolution, dimension standards, rotation detail, snap radius).

Changes to these defaults are made through the **Preferences** dialog box, which is opened with the **Preferences** command on the **File** menu.

Changes made to this file are saved when you click **OK**. These settings become the defaults until you use this command to change them again.

The following items are controlled with checkboxes on the **Preferences** dialog box:

Click-Click Drawing

Click-Drag is the conventional method of drawing, wherein you click with the mouse, drag it with the button depressed, and release the button to finish the action.

If this option is deselected, MiniCad is in **Click-Click** mode. In this mode, you click the mouse and drag it without holding the mouse button down (i.e., you simply move it). An action is finished by clicking the mouse again.

Click-Click mode is useful if you enter data frequently in the **Data Display** bar. It is also helpful when pens and digitizing tablets are used.

Note: Some tools use a unique combination of clicks and drags, and are not affected by the **Click-Click Drawing** option. These include the **Polygon** and **Wall** tools and some **Arc** and **Dimension** tool modes.

Offset Duplications

When this item is selected, any duplicated object is offset from the original. The offset places the duplicate slightly above and to the right of the original.

If the duplicate is moved immediately after it is duplicated, subsequent duplications are placed at that offset. If **Offset Duplications** is deselected, the duplicate is placed on top of the original.

Screen Cursor

When selected, the cursor displays solid lines that extend in the *x* and *y* axis to the edge of the screen.

Screen Hints

Screen Hints are the interactive text that accompanies the SmartCursor. When selected, search snapping is enabled; when deactivated, search snapping is off.

Screen Hints can be toggled on and off with the **Y** key.

For more information about this option, see **Using Screen Hints and Floating Datum**.

Floating Datum

A datum is a temporary variable placed into memory. A datum is set by holding the cursor on a snap point for a few moments.

When the cursor snaps to a snap point, the location of the cursor is placed into memory as datum. The user can type an offset to the datum to which the cursor snaps. Thus, the user can draw or place objects at exact distances from any snap point.

There can only be one datum point in the memory at a time. If **Floating Datum** is deselected in the **Preferences** dialog box, the origin is set to be the datum and remains constant until either the origin is changed using the **Set Origin** command, or the **Floating Datum** is turned on again.

For more information about this option, see **Using Screen Hints and Floating Datum**

Snap to Loci

When selected, the snap grid works with loci placed with the **2D Locus** or **3D Locus** tool.

Rulers

This preference regulates the display of the rulers. When selected, on-screen rulers are displayed in the MiniCad drawing window.

Scroll Bars

This preference regulates the display of the scroll bars. When selected, standard scroll bars are displayed in the MiniCad drawing window. If deselected, scroll bars are suppressed. The arrow keys or the **Pan** tool must be used to move the view of the drawing window.

No Fill Behind Text

A text block can have a fill pattern. By default, a text block is given the currently selected fill pattern when the text block is created.

When the **No Fill** checkbox is selected, the text block is created without a fill pattern. When a fill pattern is wanted, it can be assigned to the text block using the **Attribute Palette**.

Zoom Line Thickness

This preference keeps line thickness in proportion when the drawing is zoomed. When selected, this preference forces lines to show in proportion to the zoom level. When deselected, lines display exactly as they are going to be printed.

Black & White Only

When selected, the drawing displays in black and white. This setting supersedes the current monitor setting in the display control panel. Using a black and white display speeds up screen redraws.

Use Layer Colors

When selected, this preference displays objects in their layer color instead of their object color (layer color is specified by the **Color** option in the **Layers** command). When **Use Layer Colors** is enabled, the individual object colors are not displayed. However, the object's color is maintained internally, and will display properly once this preference is disabled.

Log Time In Program

MiniCad maintains a log of your work in the program. The log file keeps track of when you open MiniCad, when you open and close individual files, and when you quit MiniCad. This log file is maintained as a text file in the same folder as the MiniCad application. The log file can be opened with any word processor.

Selecting the **Log Time in Program** preference creates the log file if one has not been created yet. If a file has been created, new data is appended the existing log.

An example of the log file contents is shown below:

4/19/97 12:37 PM	Started MiniCad.
4/19/97 12:38 PM	Closed "Untitled 1" without saving.
4/19/97 12:38 PM	Opened "Jones Addition".
4/19/97 4:29 PM	Closed "Jones Addition".
4/19/97 4:36 PM	Quit MiniCad.

Adjust Flipped Text

When selected, this option prevents mirrored text from being flipped. Instead of flipping, text is rotated, which prevents it from being mirrored. If deselected, mirrored text displays backwards.

Show Other Objects While In Groups

When selected, the entire drawing is visible while editing groups with the **Group command**. When deselected, the editing window hides everything but the group being edited. This reduces screen clutter.

Show 3D Axis Labels

When selected, this option displays the *x* and *y* axes of the ground plane, or the *i* and *j* axes of the working plane, in a 3D projection.

Black Background

When selected, the background becomes black. When deselected, the background is white.

Eight Selection Handles

When selected, a selected object is defined by eight selection handles. The handles are located at the corners and mid points of the object's bounding box. When deselected, a selected object is defined with four handles in the corners of its bounding box.

Constrain Angle

The value in the **Constrain Angle** field allows an additional angle constraint to be set. This constraint is in addition to the 0°, 30°, 45°, 60°, and 90° default constraints available in the **Constraint Palette**.

Snap Radius

The **Snap Radius** is the distance the cursor has to be from an object before snapping occurs. The value is specified in screen pixels (typically 72 per inch.)

Conversion Resolution

The **Conversion Resolution** determines the number of line segments placed into curved surfaces when converting to polygons or lines. It is also used in other sequences, such as rotations.

The option contains four choices: **Low**, **Medium**, **High**, and **Very High**. The higher the value, the rounder the curve.

3D Conversion Resolution

The **3D Conversion Resolution** determines the number of segments to be created when an object is converted to 3D.

The option contains four choices: **Low**, **Medium**, **High**, and **Very High**. The higher the value, the rounder the curve. The higher the setting, the rounder the curve. However, the higher the setting, the larger the file and the more RAM needed to render it.

Dimensions

The **Dimension** pop-up menu allows selection of dimension standards. Choosing **Custom** from the menu allows custom dimension standards to be created or edited. For more information about custom dimension standards see the *MiniCad User Manual*.

3D Rotation

The **3D Rotation** buttons are a series of five radio buttons that control the degree of detail maintained in a 3D model as it is rotated. This setting is always within the range allowed by the hardware being used. That is, the faster the computer, the more control these buttons allow.

Save Reminder

MiniCad can automatically remind you to save the active file at a specified interval. If the **Save Reminder** preference is selected, the file automatically saves at the time interval specified in the **Minutes** field.

If a file has not been previously saved, a dialog box opens to allow the file to be named. Subsequent saves are saved to the specified file without opening the dialog box.

When the file is being auto-saved, the mode bar reports the operation's progress. During the brief time that the file is being saved, you cannot make changes to the file.

Each time the cursor snaps to a point, that point is placed in memory. This allows snapping to the intersections of the horizontal and vertical vectors that extend from the snap points.

For example, position the mouse at an object's snap point. After snapping to the point, move the mouse away from the object. A dashed line appears. Position the mouse at another object's snap point. The dashed line extends to the snap point and the word "Intersect" appears when the cursor is snapped to the intersection. The cursor also displays a snap dot.

The first snap point that the cursor touched is now indicated by a small square. The second point has a small circle. Each time the cursor touches a new snap point, it is displayed with a small circle.

When the cursor crosses a horizontal or vertical vector of any of these snap points, the points are displayed as squares.

For additional information, see the following topics:

Screen Hints with Snap to Surface
Angle Vectors
Screen Hints with Snapping Combinations

When the **Snap to Surface** constraint is selected, screen hints displays the word "Surface" to the right of the cursor when the cursor is within the snap distance of the surface of an object.

To get the vector of an angle (thus allowing snapping to an extension of angled lines or surfaces), move the cursor to the surface of the object. After a brief pause, a dotted line representing the vector from the object appears. Up to two objects can be stroked to get a dotted vector of their surface.

Virtually any combination of constraints can have screen hints. It can, however, be necessary to move the cursor around slightly in order to get the particular snap data to appear when several combinations of snaps are in a small area.

The **Print** command, on the **File** menu, prints or plots a copy of the active document. The actual printed output is determined by the selected printer, the settings made with the **Drawing Size** command (one page is the default), the orientation determined with the **Print Setup** command, and the paper selected with the **Print Setup** command.

Worksheets have their own menus and have their own print command. If you want to print a worksheet, use the **Print** command on the worksheet's main menu. However, worksheets that have been placed on the drawing are considered graphic elements of the drawing and are printed with the standard **Print** command.

All objects, layers, and classes that are visible are printed.

The **Print Setup** command on the **File** menu specifies the page size, orientation and resolution of the current document when it is printed. When the command is chosen, the **Print Setup** dialog box appears.

The **High Resolution** check box sets the selected printer to its optimal resolution, as determined by the printer driver software installed on your system.

Deselecting the check box reduces the resolution to screen resolution (72 dpi). Reduce resolution when printing to devices that do not support the system software's high resolution printing routines (certain fax modems, for example).

Select the paper size and page orientation desired. The selected paper size, reduction/enlargement, and orientation are reflected in the gray printer boundary that shows on the screen.

The actual number of pages it takes to print your document is determined by the **Drawing Size** command on the **Page** menu. The size of those pages is determined by the currently selected printer.

For related information, see the following topics:

Print

Projection is the process of mathematically transforming objects in the 3D model into lines on a 2D projector plane so that the viewer has the impression of viewing 3D objects.

MiniCad has seven standard projections, chosen from the **Projection** submenu on the **3D** menu. The active projection is shown with a checkmark.

Top/Plan
Orthogonal
Perspective
Oblique Cavalier 30 & 45
Oblique Cabinet 30 & 45

Top/Plan is used for normal 2D drafting. **Plan** is not really a projection in the traditional drafting sense. It is a special projection used in MiniCad to provide a way to link 2D with 3D. This provides the link that allows layers to print in perfect registration. **Top/Plan** also allows 2D views and 3D projections to be displayed and printed on the same page.

Orthogonal is the traditional isometric projection used by architects and engineers. It is probably the most commonly used projection for 3D CAD.

This projection displays the drawing as an undistorted projection in which the projectors are perpendicular to the projection plane and parallel to each other. An important property of the orthogonal projection is that objects on the projection plane are their exact size regardless of their distance from the plane.

Perspective projection adds distortion to the displayed objects so that some objects appear to be farther from the viewer than others.

Perspective is very useful for determining what a modeled object is going to look like in its final form, because it is distorted on screen the same way it is distorted when seen in the real world. Objects in the distance appear smaller than objects close to the viewer, and parallel lines converge to a vanishing point.

In perspective projection, the projectors converge to a point called the **Center of Projection** behind the projection plane. The distance between this point and the projection plane determines the amount of perspective, like the focal length of a lens.

In a **Cavalier** projection, a vector perpendicular to the projection plane has same length on the page as those parallel to the projection plane. Therefore, for rectangular objects, all dimensions on the page are actual.

An advantage of a **Cavalier** projection is that measurements can be taken from the drawing after it is printed. **Cavalier** is an oblique projection, meaning that the projectors are parallel, as in orthogonal, but they are not perpendicular to the projection plane.

Be aware that the object being observed must be oriented so that its vectors are aligned with the projection plane. This insures that the dimensions are correct. This means that distances are accurate only if the oblique cavalier projection is used with the front, back, top, bottom, left, or right standard views.

Oblique Cabinet projection is very similar to **Oblique Cavalier**, but the object is foreshortened by 50% to achieve a more natural look.

Distances are still easy to measure, but distances of those vectors perpendicular to the projection plane must be multiplied by 2 to get the actual dimension. Cabinet is an oblique projection, meaning that the projectors are parallel, as in orthogonal, but they are not perpendicular to the projection plane.

Like **Oblique Cavalier**, the object being observed must be oriented so that its vectors are aligned with the projection plane. This insures that the dimensions are correct. This means that distances are predictable only if the oblique cabinet projection is used with the front, back, top, bottom, left, or right standard views.

The **Purge Unused Objects** command on the **Edit** menu removes any combination of unused layers, classes, symbols, and record formats from the active drawing.

When chosen, only layers without any objects on them are removed. The command removes any symbol in the library that is not placed on the drawing, as well as any record format not attached to a symbol or object. This command cannot be reversed; a dialog box confirms the choice before the command is executed.

If the symbols are placed in folders in the symbol library, the symbols are removed but the folders remain.

This command allows large, complicated stationery pads to be used without filling the finished file with unnecessary items. An "office standard" stationery pad can be used, with elaborate layer and class configurations. When the drawing is finished, all unused items can be easily removed.

The **Quit** command on the **File** menu closes MiniCad. If any unsaved documents are opened when you choose the **Quit** command, you are asked if you want to save them.

If there are several unsaved documents, you are asked about each document individually.

The **Recover** command is used to repair a damaged drawing file.

Note: The **Recover** command is located on the alternate **File** menu. This menu is activated by, pressing the **Control** key while selecting the **File** menu from the menu bar.

When a file is opened with the **Recover** command, MiniCad attempts to make repairs to the file. Be aware that not all damaged files can be repaired.

If the **Recover** command fails to recover a damaged file, **Export** the file in text format and **Import** the file back in text format.

Rendering is the process by which MiniCad applies a solid surface to a 3D wireframe object. Rendered objects can have a color and/or fill applied to the surface. If desired, a light source be used to shade the rendered object.

To render a drawing that contains 3D objects to show as a solid, choose one of the rendering commands from the **Rendering** submenu on the **3D** menu.

Wireframe
Quick
Solid
Shaded Solid
Shaded - No Lines
Final Shaded Solid
Final Hidden Line

The **Wireframe** command on the **Rendering** submenu produces a wireframe image of the 3D model. All of the object's lines are visible. This is the default for MiniCad models and it is the fastest way to view a 3D model.

The **Quick** command on the **Rendering** submenu is the fastest solid rendering mode, but it does not always render objects accurately. Polygons may be rendered in the wrong order using this method.

Quick render is a shaded render mode. The first time a shaded render command is chosen, the **Sun** cursor appears. Position the cursor where the light source is to originate and click. The light source can be dragged to a new location after it has been placed. If a light source has already been placed, rendering is performed immediately.

The **Solid** command on the **Rendering** submenu renders any object with a fill pattern (which includes the default fill, white) as a solid.

Any objects that lack a fill are shown in wireframe.

Each object, or facet of an object, can have a different fill and/or color. Fill and color are chosen from the **Attribute Palette**.

Objects rendered as a solid frequently look better if a color is used instead of white. A white fill is difficult to see on screen and when printed. Giving the different faces of a 3D object different shades of gray is particularly effective.

The **Shaded Solid** command on the **<u>Rendering</u>** submenu is used to render an object as a solid, with its surfaces shaded to simulate a directional light source.

The first time a shaded render command is chosen, the **Sun** cursor appears. Position the cursor where the light source is to originate and click. The light source can be dragged to a new location after it has been placed. If a light source has already been placed, rendering begins immediately.

The **Shaded - No Lines** command on the **Rendering** submenu creates a rendered, shaded object with color and fill information, but without a wireframe.

It effectively removes all lines from the drawing. The first time a shaded render command is chosen, the **Sun** cursor appears. Position the cursor where the light source is to originate and click. The light source can be dragged to a new location after it has been placed. If a light source has already been placed, rendering begins immediately.

The **Final Shaded Solid** command on the **Rendering** submenu produces a very accurate rendering, with surfaces shaded to simulate a directional light source.

The price of the accuracy is speed. This mode is relatively slow. It is best for making final presentations.

The first time a shaded render command is chosen, the **Sun** cursor appears. Position the cursor where the light source is to originate and click. The light source can be dragged to a new location after it has been placed. If a light source has already been placed, rendering begins immediately.

The **Final Hidden Lines** command produces is the most accurate rendering. It actually renders the object twice. A hidden line rendering does not have color or shading. It is best for plotting and making final presentations.

The **Resource Palette** opens when the **Resources** command is chosen from the Window menu. It is used to access a variety of drawing components, including:

- Symbols
- Worksheets
- Macro Commands
- Hatch Patterns
- Database Record Formats

The **Resource** palette changes as different types of resources are selected; the palette displays different buttons depending on the type and location of the selected resource.

The palette is closed by clicking the close box.

The palette can be stretched to display more resources if desired. Like all palettes, the **Resource** palette can be moved around the screen.

The **Revert to Saved** command removes all changes to the active document since the last time you saved. The last saved version of the document opens in place of the active document.

Revert is used only if recreating a document from the last save is easier than repairing any undesirable changes you made. Because of this, save the document before doing any work you may want to discard.

Because the **Revert to Saved** command cannot be reversed, you will be asked to confirm your request.

The **Roof** command on the **AEC** menu makes hybrid 2D/3D roofs from simple 2D objects. As a hybrid object, it displays as a 2D object in **Top/Plan** view, but displays as a 3D object in an **Orthogonal** view.

Note: This command is only available when the **AEC Overlay** is active. If the **AEC** menu is not listed on the menu bar, select **AEC Overlays** using the **Overlays** command on the **File** menu.

The **Roof** command creates complex roof structures in a few simple steps. You can either create the roof in the same layer as the wall structure, or in a layer of its own. If you create the roof in its own layer, you can use the **Create Layer Link** command to create a model of the walls and roof together.

For additional information, see the following topics:

Making Roofs

Before the roof is drawn, a preliminary step should be taken. Locate the bearing point for the roof. This is the point where the roof sits on the walls underneath it. Draw a temporary reference line extending beyond the walls, past the eaves (if any.) This reference line will help locate the roof axis.

To create a standard gable roof, draw a 2D polygon or rectangle that represents half of the roof. Select the object and choose the **Roof** command.

The **Create Roof** dialog box opens. This dialog box has fields to set the parameters (e.g., slope, rise, thickness) for the roof. Make your selections in this dialog box, then click **OK**.

See **The Create Roof dialog box** for description of the options in this dialog box.

Next, the axis for the roof must be defined. Drag a line using the reference line to guide you. A triangle forms at the axis line. The next click of the mouse defines the side of the axis on which the peak is located. Click the side where the ridge is located. Then select the reference line and delete it. Select the **Mirror/Duplicate** tool and drag a line down the ridge of the roof.

When you release the mouse, the other half of the roof is formed. Refer also to the *MiniCad User Manua*l for more information about creating roofs.

You can cut a hole through the roof for a chimney or dormer, before or after the roof is created. If you want to cut holes before creation, use the **Clip** tool or draw an object in the form of the hole and choose the **Clip Surface** command on the **Tools** menu to make the hole.

If the roof is already created, use the **Edit Group** command on the **Organize** menu to "enter" the roof for editing. Then, use the **Clip** tool or draw an object and choose the **Clip Surface** command to make the hole.

After making all of the changes, click on the **Exit Group** button in the data display bar or choose the **Exit Group** command in the **Organize** menu.

The **Create Roof** dialog box is opened when the **Roof** command on the **AEC** menu is selected. It has the following options:

Axis *Z*

The **Axis *z*** of the roof is the distance of the axis from the base of the layer. If the roof is in its own layer, this value is set to zero.

If the roof is to be placed on top of some walls, select one of the walls and open the **Object Info Palette**. Note the *z* value for the wall and enter that in the **Axis *z*** field. Generally, it is better to put the roof in its own layer and use layer linking to view the layers as a model.

Angle

If you know the roof angle, select the **Angle** button and enter the value in the field.

Rise over Run

If you know the rise/run of the roof, select the **Rise over Run** button and enter the values.

Height at next click

This option allows you to set the peak of the roof with a mouse click, and lets you match an existing condition without a lot of calculations.

Thickness and Miter

The **Thickness and Miter** options allow you to set those attributes for the roof.

While using the **Double Miter** option, you can set a horizontal and vertical miter. If you enter the lumber thickness and one of the miter values, you can have the dialog box calculate the third value by pressing the **Tab** key.

For example, if the roof sits on a 2 x 4 plate, enter 3 1/2" in the horizontal field and press the **Tab** key. The vertical field is calculated.

The **Rotate** command on the **Tool** menu allows you to rotate a selected object by degrees, minutes, and seconds (as determined by the accuracy settings in **Units** command) or by a decimal number of degrees.

Text is rotated only in degrees.

Objects also can be rotated around a locus point.

The **Rotate** command's submenu contains the following commands:

Rotate...

The **Rotate...** command lets you rotate a 2D object by any degree.

Rotate 3D

The **Rotate** command lets you rotate a 3D object by any degree. When the **Rotate 3D** command is chosen, the **Rotate Object in 3D** dialog box opens. Select a rotation center, select the axis, and type the value for the degree of rotation into the field. Click **OK**.

See **The Rotate Object in 3D dialog box** for description of the options in this dialog box.

Rotate Right 90

The **Rotate Right 90** command lets you rotate a 2D object 90° right, in one step.

Rotate Left 90

The **Rotate Right 90** command lets you rotate a 2D object 90° left, in one step.

Flip Vertical

The **Flip Vertical** command lets you pivot the object across the horizontal axis (i.e., up/down)

Flip Horizontal

The **Flip Horizontal** command lets you pivot the object across the vertical axis (i.e., left/right).

For related information, see the following topics:

Rotate 2D
2D Mirror
Rotate to Zero
Rotate Working Plane
Rotate View
Walkthrough
Flyover

The **Rotate Object in 3D** dialog box is opened when the **Rotate 3D** command is chosen. It has the following options:

Rotation Center

The 3D object can be rotated about one of three options; the center of the working plane, the center of the object, or any point that is selected with a mouse click.

Rotation Axis

The object can be rotated along either the ground plane (*x, y, z*) or the working plane (*i, j, k*) axes.

Rotation Angle Field

The **Rotation Angle** field is used to enter the desired rotation. The accuracy of the rotation is determined by the **Units** command.

The **Rotate 3D View** command on the **3D** menu allows manipulation of a 3D view. This dialog box permits either interactive or direct input of the *x*, *y*, and *z* coordinates in degrees, minutes and seconds.

This means you can manually move the view in predefined steps, or you can enter a value directly in the rotation fields. The result of the rotation is shown in the thumbnail diagram displayed on the left side of the dialog box.

The **3D Rotation** dialog box has the following fields:

Set By Value
Allows you to see the results of the values entered into the fields below. After entering the desired rotations for the *x*, *y*, and *z* axes, clicking on the **Set by Value** button causes the thumbnail diagram in the dialog box to show the degrees of rotation.

Rotation on X, Y, or Z
Enter the desired degree of rotation in degrees, minutes and seconds.

Absolute
When checked, all rotations set in the dialog box are made from the top (absolute) view. If unchecked, rotations set in the dialog are from the current view.

Increment
Sets the number of degrees each click of the plus or minus buttons moves the viewpoint.

The X, Y, & Z Plus And Minus Buttons
Each click on these buttons moves the selected object by the number of degrees in the increment field.

OK
Accepts the entered rotation. When the screen redraws, the selected object is rotated.

Cancel
Returns you to the drawing without making any changes.

For related information, see the following topics:

Rotate View
Walkthrough
Flyover
Rotate

The **Rotate to Zero** command on the **3D** menu is used to align a 3D object with the world coordinate system. It can be used with any 3D object, including symbols, extrudes, multiple extrudes and sweeps.

This command is useful for quickly reorienting an object to its upright position if you become disoriented while rotating it.

For related information, see the following topics:

Rotate
Rotate 3D

The **Rotate Working Plane** command allows rotation of the working plane with absolute precision.

It is the most accurate way to rotate a working plane. The working plane can be rotated by degrees, minutes, and seconds (as determined by the accuracy settings in the **Units** command), or by a decimal number of degrees.

A working plane must be active to rotate it.

When this command is chosen, the **Rotate Working Plane** dialog box opens. The dialog box allows the choice between the *x, y, z* or the *i, j, k* axes. Select the desired axis and enter the desired rotation in the **Rotation Angle** field.

For related information, see the following topics:

Rotate
Rotate 2D
Rotate 3D
Rotate View

The **Save** command on the **File** menu saves the open document to disk.

The **Save** command updates the document by replacing the version on disk, with the version on the screen.

If the file has been previously saved, it saves without any messages or dialog boxes.

If the document has never been saved before, the **Save** dialog box opens. Enter a name for the file and click **OK**. The file is saved in the active directory. New folders can be made by clicking the **New** button.

For related information, see the following topics:

Save As
Save as Stationery

The **Save As** command saves a copy of the active document under a new name, in a different folder (i.e., directory), or on a different disk. You also can use the **Save As** command to save a new document.

Use the **Save As** command when you want to make a copy of the active drawing. The original document is closed, without saving any changes from the last save. A new file is opened, and the active document is copied into it.

For related information, see the following topics:

Save
Save as Stationery
Save View

Before drawing with MiniCad, most users configure the file's scale, set the grid, set wall thickness, set units, configure classes and create layers. These settings and more can be preconfigured so that new files are created with these settings as defaults.

This is done by saving a preconfigured file as a "Stationery file."

A Stationery file is created with the **Save as Stationery** command on the **File** menu. Any MiniCad drawing file can be saved as stationery. A Stationery file can be also include symbol libraries, macro commands, hatch patterns, record formats and class lists.

If a stationery file is saved with the name "Default.sta" and is stored in the same folder as the MiniCad application, it is used as the default file for all drawings subsequently created with the **New** command.

The stationery file has no effect on existing drawings.

MiniCad allows you to save individual views of your drawing. This feature simplifies the presentation of your drawing. The saved view is given a name, and placed on a command palette. Saved views are actually macro command that are created by arranging the drawing as desired and selecting the **Save View** command.

In addition to saving the "view" of a drawing, the **Save View** command saves class and layer settings, which means you can show a certain "class" by simply clicking names on a list. This is far easier than using the **Classes** command to toggle the visibility of the classes.

When you choose the **Save View** command, the **Save View** dialog box prompts you to name the view and choose certain options.

See **The Save View Dialog Box** for description of the options in this dialog box.

When you click **OK**, the command is put in the active command palette if one is open. If no command palette exists in the drawing, one is created.

Macro commands and command palettes can also be created and edited with the **Resource Palette**.

The **Save View** dialog box is used to save a view to the command palette. It has the following options:

Save Current View

When selected, the view active at the time the command is chosen is saved. The file will switch to this view when the custom view macro is executed.

Save Active Layer

When selected, the active layer is saved with the custom view. The file will switch to the selected layer when the custom view macro is executed.

Save Layer Visibility

This option retains the current layer visibility information with the saved view. For example, you can have a drawing with three layers, one visible, one grayed and one invisible.

When you execute the macro, the drawing changes to these visibility settings, no matter what the layer settings were before the macro command was invoked.

Save Class Visibility

This option retains the class visibility settings for the view you want to save. When you execute the macro, the drawing changes to the specified visibility settings, no matter what the class settings were before the macro command was invoked.

The **Scale** command on the **Organize** is used to set the scale of the active layer.

Scaling is the ability to accurately display, or print, a large drawing in a manageable size (do not confuse scale with the **Zoom In** and **Zoom Out** tools, which only determine how the drawing is displayed).

While a drawing can include objects drawn in different scales, each layer can have only one scale. Therefore, as long as the elements are in separate layers, you could have a title block in 1:1, a site plan in 1:50, a floor plan in 1/4" and details in 1/2".

All new layers automatically assume the scale of the active layer, unless you specify otherwise. Layer scale can be changed at any time.

When you select the **Scale** command, the **Drawing Scale** dialog box appears. It offers architectural, engineering and enlargement scales. The value in the **Paper Scale** field will change to reflect changes in the scale. You can type a value directly into the field, if desired.

The **All Layers checkbox** is used to set the default scale for all layers in a drawing. Care must be used when selecting this checkbox, as it overrides settings made with the **Layers** command.

If you want text to be scaled with the drawing, select the **Scale Text** option.

For related information, see the following topics:

Scale Objects

Resize

The **Scale Objects** command allows you to rescale the *x* and *y* dimension of any selected object or group.

Note: At least one object must be selected in order to use this command.

When **Scale Objects** is chosen on the **Tool** menu, the **Scale Objects** dialog box opens.

You can scale the *x* and *y* dimensions independently. Type the value for the scaling factor in the appropriate fields.

The default scale is 1:1, so any factor entered less than one reduces the object's size. Any factor greater than one enlarges the object (e.g., a scaling factor of 2.5 makes the selected object two and a half times larger).

Rescaling is done from the center of the object.

If the **Scale Text** check box is selected, selected text is scaled to match the entered scaling factor.

If the **Entire Drawing** check box is selected, all visible objects in the drawing are scaled (grayed object are not rescaled).

For related information, see the following topics:

Scale
Resize

The **Select All** command selects all visible objects on the active layer.

Objects that are invisible are not selected.

Objects can be selected in other layers if the **Layer Options** command is set to **Show/Snap/Modify**.

Visible objects on all normal layers are selected. (Setting a layer to **Normal** with the **Layers** command makes it visible.)

For related information, see the following topics:

2D Selection

3D Selection

In MiniCad, every object is drawn in a sequential order with each object being drawn atop the other. This is known as the stacking order. An object maintains its position in the stacking order until it is changed.

There are two ways to move an object through the stacking order. One way is to **Cut** the object to the Clipboard and paste it back into the drawing. The other way is to use one of the **Send** commands. The **Send** submenu on the **Tool** menu contains four different commands.

Objects are moved through the vertical stack by selecting the object and choosing one of the commands on the submenu. The choices are:

Send To Front

When chosen, the selected object is sent to the front (i.e., to the top of the stack). This can also be done with the **Control-F** key combination.

Send To Back

When chosen, the selected object is sent to the back (i.e., to the bottom of the stack). This can also be done with the **Control-B** key combination.

Send Forward

When chosen, the selected object is moved one level forward in the stacking order (i.e., one step up).

Send Backward

When chosen, the selected object is moved one level back in the order (i.e., one step down).

The **Set 3D View** command on the **3D** menu can be used to create a 3D perspective view by dragging a line on a 2D floorplan. The command also can be used while in a 3D projection.

This command lets you specify both the viewpoint and the exact point of interest in both height and location. For example, you could use it to inspect a cornice as viewed from the street, or get an idea of how a kitchen would look as viewed from the kitchen table.

To create a 3D view, choose the **Set 3D View** command. Position the cursor at the viewpoint and drag it in the direction to view. This line determines the actual view. The longer the line drawn, the more accurate the control over the view.

When the mouse is released, a dialog box opens for other parameters of the view.

The **Viewer Height** field sets the height of the viewpoint.

The **Look Toward Height** field determines the height of the point being viewed. One of four perspective choices is made from the pop-up menu. When OK is clicked, the view is created.

Note: If the height of the viewer and the viewpoint is the same, the perspective is quite normal. If however, the two points are different in height, the length of the line drawn with this command controls the distortion in the perspective. This is because the shorter the line, the greater the angle between the two points, and hence the exaggeration of the perspective.

The **Walkthrough** tool can be used to change the view from within the 3D perspective. The perspective can be changed by selecting it with the **Translate View** tool.

The **Set Grid** command on the **Page** menu is used to configure the drawing's grid system.

MiniCad maintains an internal, user-definable grid system. The grid system has two components:

- the invisible, **Snap Grid**, and
- the visible, **Reference Grid**.

These grids can be changed at any time in a drawing by selecting the **Set Grid** command. This command opens the **Set Grid** dialog box.

Note: the **Set Grid** dialog box can also be opened by double-clicking **Snap to Grid** icon on the **Constraint Palette**.

The **Set Grid** dialog box has the following options:

Current Snap Grid
The value you set in the **Current Snap Grid** field sets the **Snap Grid.** This value specifies the smallest unit increment to which the cursor snaps.

Different grid settings help constrain the mouse to the level of detail you desire. This helps maintain a level of precision consistent with the details with which you are working.

Reference Grid
The **Reference Grid** is simply a visual aid and has no physical effect on the drawing. Setting the **Reference Grid** to a multiple of the **Snap Grid** allows you to snap at the **Reference Grid's** intersections.

The display of the **Reference Grid** is affected by the current zoom. When the zoom levels make the grid unwieldy, the grid will be represented as points on the screen.

Show Grid Lines
When selected, the **Reference Grid** will be visible.

Print Grid Lines
When selected, the reference grid will print, and will export with the EPSF and PICT formats.

The origin point in MiniCad is the point on the drawing where the *x* and *y* axis meet. All *x* and *y* values are determined to be positive or negative in relation to this point.

The origin point can be changed. The default origin is the center of the paper as determined in your page setup. If you change the paper size, the origin point changes to the new center point.

Changing a drawing's origin does not change the physical drawing, but it does change the drawing's reference points (the *x* and *y* coordinates).

To set a custom origin point, select **Set Origin**. The cursor changes to a bull's-eye. Click on the drawing at the point you have selected as the new origin point.

Note: The origin point can also be set by dragging the mouse from the upper left corner of the drawing window. Release the mouse when it is at the new origin point.

The **Shallow Symbol to Group** command converts a symbol made up of multiple symbols, into a single group of individual objects rather than decomposing each symbol as a separate group.

Note: The **Shallow Symbol to Group** command is located on the alternate **Organize** menu. This menu is activated by, pressing the **Control** key while selecting the **Organize** menu from the menu bar.

For related information, see the following topics:

Symbol to Group

The **Shift Drawing** command moves the entire drawing, including all its layers and any invisible objects, in one motion.

Note: The **Shift Drawing** command is located on the alternate **Page** menu. This menu is activated by, pressing the **Control** key while selecting the **Page** menu from the menu bar.

To move the drawing:

1. Select the **Shift Drawing** command from the alternate **Page** menu.
2. Click the mouse on the drawing.
3. Move the cursor to the where you want the drawing shifted, and click again.

For related information, see the following topics:

Move Page

The **Smoothing** command on the **Edit** menu is used to change all curves in a selected object to one of the following types:

- Corner Curve
- Bezier Curve
- Cubic Spline
- Arc Radius

Note: This command automatically converts *all* vertices in the object to the selected curve type. If you want to change vertices individually, use the **2D Reshape** tool.

To use the Smoothing command:

1. Select the object you want to smooth.
2. Select the **Smoothing** command from the **Edit** menu.
3. Select the type of curve to which you want all vertices converted. The **No Smoothing** option converts all curves to corner points.

For related information, see the following topics:

Polyline

The **Standard Views** command on the **3D** menu is used to place your drawing in one of MiniCad's standard views.

Standard views simplify the orientation and manipulation of your drawing. The currently selected view will have a check mark next to its name.

The standard views are divided into four basic categories:

Views	Description
Top/Plan	The 2D view. Choosing this view switches to 2D Plan projection.
Top, Bottom **Front, Back** **Right, Left**	3D elevations.
Right Isometric **Left Isometric** **Right Rear Isometric** **Left Rear Isometric**	3D representations of your drawing.
Lower Right Isometric **Lower Left Isometric** **Lower Right Rear Isometric** **Lower Left Rear Isometric**	3D representations of your drawing taken from below the ground plane. These views are useful in mechanical design. When looking at the object from below, the ground plane loses its grid.

Additional views of a drawing can be created with the **Save View** command.

Additionally, any of the view manipulation tools on the **3D Tools** palette can be used to place the drawing in any orientation.

For related information, see the following topics:

Set 3D View
Translate View
Rotate View
Flyover
Walkthrough

The **Structural Shapes** command on the **AEC** and **Mechanical** menus creates profiles various of various structural elements.

Note: This command is only available when the **AEC Overlay** or the **Mechanical Overlay** is active. If the **AEC** or **Mechanical** menu is not listed on the menu bar, select **AEC Overlay** or **Mechanical Overlay** using the **Overlays** command on the **File** menu.

When the command is chosen, the **Create Structural Shape** dialog box opens. The dialog box has two pop-up menus used to select the shape and nominal size of the structural shape.

The **Structural Shape** command uses an editable data file to produce the actual shape. These data files are located in the **External Data** folder in the **Externals** folder.

The **Externals** folder is located in the same folder as the MiniCad application. Complete information on editing these data files is contained in your *MiniCad Toolkit Manual*.

Regardless of the units setting used with the active drawing, the **Structural Shapes** command uses industry standards commonly used in the US. This means all beam calculations are in feet and inches.

The **Sweep** command allows you to create a 3D cylindrical object from a 2D object. A sweep has four basic elements:

- a Centroid,
- an Arc Angle,
- a Segment Angle, and
- the Pitch.

To learn more about these components, see .

A group cannot be swept, however, multiple selections can.

Multiple selections are swept as grouped 3D objects. Afterwards, they can be ungrouped with the **Ungroup** command and edited separately.

When a line is swept, MiniCad creates an outline of a surface. When objects with a surface are swept, a wireframe solid is created.

When the **Edit Group** command is chosen with a swept object, the edit window opens, displaying the original 2D object. Edits to the 2D original will affect the swept object. When the **Exit Group** button is clicked in the mode bar, the edited swept object is displayed.

Converting a swept object to a mesh object allows you to edit individual vertices.

For additional information, see the following topics:

Specifying Sweep Components

For related information, see the following topics:

Extrude
Multiple Extrude
Duplicate Array

To execute a sweep, you must specify the following four basic elements:

Centroid

To sweep an object, you must establish a center of rotation, or centroid. This is done by placing a locus on or near the object to be swept at the centroid. The object and the locus are selected before the **Sweep** command is chosen. The snap functions aids in placement of the locus

If a 2D object is swept without a locus establishing a centroid, the object is swept about itself.

Arc Angle

The arc angle is the degree of the sweep. The **Sweep** command creates an object that is swept a full 360°. This can be changed by opening the **Object Info Palette** and entering a different arc angle.

Segment Angle

The segment angle is the number of segments making up a circle. By default, this is thirty-six, which means there is a segment for every ten degrees of arc.

This number can be changed using the **Object Info Palette** after an object is swept. The default can be changed with the **Preferences** command on the **File** menu.

Pitch

The pitch of a sweep is the degree to which it spirals. This effect is like the coils on a spring.

The pitch is represented in terms of plus or minus height per revolution. This means that if the pitch is one inch, the object rises one inch for every revolution. The default pitch is zero. The pitch can be changed using the **Object Info Palette** after an object is swept.

The **Symbol Edit** command replaces selected symbols placed on the drawing with a different symbol in the symbol library.

Any number of selected symbols can be replaced simultaneously.

The symbol's rotation is maintained when a symbol is replaced. Because symbols are inserted with a specific insertion point; they will rotate consistently as long as their insertion points are consistent.

For example, if symbols are inserted from the object center, they will replace one another appropriately at any angle. However, if some symbols are inserted from the center and others from the upper-left corner, the replacement symbols will be inserted inconsistently.

The command also can be used to precisely move symbols from their current positions on the drawing.

The **Symbol Edit** command can replace a selected symbol placed in a wall with another. Because walls are a special type of object, different rules apply to them:

- Only one symbol at a time can be selected when the symbol is inserted in a wall.
- Hybrid symbols, inserted in a wall, cannot be selected in a 3D view. The symbols must be selected in **Top/Plan** view. However, a selected symbol can be replaced while in a 3D view. This can be useful for experimenting with different doors and windows while in an elevation view.

To replace symbols placed on the drawing:

1. Select the symbols to be replaced (they do not have to be the same kind of symbol).
2. Next, choose the **Symbol Edit** command.
3. The symbol's name and a thumbnail are displayed in the **Symbol Edit** dialog box. If multiple symbols are selected, the **Original** field in the dialog box says **All Selected**.
4. Click the **Replace Symbol** button. (If this checkbox is not selected, the symbols won't be replaced.)
5. Click the **Choose Symbol** button, and select the symbol to replace the ones you've selected.
6. If you want the new symbols inserted at positions other that of the selected symbols, enter new **Y**, **Y** and **Z** position values. Distances are measured from the symbol's insertion point.
7. If the new symbol is to be rotated differently from the original, enter the value in the **Rotation** field.
8. If the selected symbol is in a wall, the new symbol can be rotated differently than the original by clicking on the dialog box. The **Offset** field shows the insertion point from the start of the wall. The **Offset** field determines the height (the Δz) of the selected symbol from the symbol's insertion point.
9. Click **OK** to replace the symbols.

If you edit a symbol directly, all instances of the symbol are changed. The **Symbol to Group** command can be used to convert a symbol to a group, so that you can edit it without making a global change to the symbol.

Note: If the symbol is inserted into a wall or other hybrid object, it must be dragged away from the object before it can be converted to a group. This is because "owned" symbols can't be edited.

After editing, the group can remain as a group, or you can create a new symbol from it.

Note: If a symbol is made up of multiple symbols (i.e. nested symbols), the **Symbol to Group** command converts each constituent symbol into a separate group. If you want to convert this type of symbol into a single group, use the **Shallow Symbol to Group** command.

The **Tablet** command on the **Page** menu is used to activate a digitizing tablet within MiniCad. MiniCad is compatible with digitizing tablets that support the WinTab standard.

To use a tablet, you will need to install the driver supplied by its manufacturer (the driver's installation instructions will be supplied with the tablet).

When the **Tablet** command is selected, the MiniCad searches for the tablet's driver. If one it found, MiniCad determines the size and resolution of the tablet, and displays the tablet on the screen as a large gray rectangle.(If no driver is found, an error message is issued.)

In tablet mode, information from the tablet is scaled directly to the drawing, so that high precision digitization can be performed. Tablets have a resolution of 1000 dots per inch or more. The tablet, when used as a mouse, is only a 72 dots per inch device.

To use the tablet:

1. Set the origin of the drawing to the lower left corner of the tablet.
2. Set the scale of the drawing to the scale of the document being traced. It is best to tape art work to the tablet at this time.
3. Select **Tablet** from the menu. In some versions, you may have to do a screen redraw in order for the rectangle to appear.
4. Look at the page size of the drawing and the tablet rectangle on screen. Are they about the same relative size as a sheet of paper and the tablet you are using? If they are, proper communication has been set up. If not, go back to Step 2.

Note: Two or three attempts can be required to establish communication with the driver.

On the tablet:

- The first available button on the puck is used for mouse clicks.
- The second button is used to toggle between functions.
- The third available button centers the screen on any point on the tablet.

The text alignment commands, **Left**, **Center** and **Right**, on the **Text** menu, control the justification of text within a single text block.

- The **Left** command makes all text in a selected text block left-justified.
- The **Center** command centers all text in a selected text block.
- The **Right** command all the text in a selected text block right-justified.

Justification is set as a default for text by deselecting the text tool and choosing the desired alignment command.

Justification of an existing text block can be changed by selecting the text block and choosing the desired alignment command.

A text block can have only one alignment. To create the illusion of different text alignments, different text blocks, with different alignments, can be created.

The text alignment commands, **Left**, **Center** and **Right**, on the **Text** menu, control the justification of text within a single text block.

- The **Left** command makes all text in a selected text block left-justified.
- The **Center** command centers all text in a selected text block.
- The **Right** command all the text in a selected text block right-justified.

Justification is set as a default for text by deselecting the text tool and choosing the desired alignment command.

Justification of an existing text block can be changed by selecting the text block and choosing the desired alignment command.

A text block can have only one alignment. To create the illusion of different text alignments, different text blocks, with different alignments, can be created.

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- The **Center** command centers all text in a selected text block.
- The **Right** command all the text in a selected text block right-justified.

Justification is set as a default for text by deselecting the text tool and choosing the desired alignment command.

Justification of an existing text block can be changed by selecting the text block and choosing the desired alignment command.

A text block can have only one alignment. To create the illusion of different text alignments, different text blocks, with different alignments, can be created.

The three text case commands, **Lower Case**, **Upper Case** and **Title Caps**, on the **Text** menu specify the case of text.

- **Lower Case** displays selected text in all lower case letters.
- **Upper Case** displays selected text in all upper case letters.
- **Title Caps** makes the first letter of every selected word, uppercase.

Text can have its case changed no matter how it was created.

For example, if a block of text had been created with the **Caps Lock** key down, it can be converted to lower case by selecting it and choosing the **Lower Case** command.

The **Lower Case** tool affects an entire text block.

The three text case commands, **Lower Case**, **Upper Case** and **Title Caps**, on the **Text** menu specify the case of text.

- **Lower Case** displays selected text in all lower case letters.
- **Upper Case** displays selected text in all upper case letters.
- **Title Caps** makes the first letter of every selected word, uppercase.

Text can have its case changed no matter how it was created.

For example, if a block of text had been created with the **Caps Lock** key down, it can be converted to lower case by selecting it and choosing the **Lower Case** command.

The **Upper Case** tool affects an entire text block.

The three text-case commands, **Lower Case**, **Upper Case** and **Title Caps**, on the **Text** menu specify the case of text.

- **Lower Case** displays selected text in all lower case letters.
- **Upper Case** displays selected text in all upper case letters.
- **Title Caps** makes the first letter of every selected word, uppercase.

Text can have its case changed no matter how it was created.

For example, if a block of text had been created with the **Caps Lock** key down, it can be converted to lower case by selecting it and choosing the **Lower Case** command.

The **Title Caps** tool affects an entire text block.

The spacing commands, in the **Text** menu, control how text lines are spaced within a text block. There can be only one line spacing setting in a text block.

- The **Single Space** command spaces lines with a minimal gap between lines.
- The **1-1/2 Space** command spaces text lines with a gap equal to half of the line's height.
- The **Double Space** command spaces lines with a gap equal to that of the line's height.
- The **Other** command opens the **Set Text Size** dialog box, so that you can specify a custom line-spacing value. You can enter the value in points, millimeters, or inches.

If text is selected, changes in the line spacing only affect the selection. Conversely, changes made without anything selected changes the default line spacing.

The **Tools** command on the **Windows** menu is used to open and close the tool palettes.

MiniCad's standard palettes (i.e., **2D**, **3D** and **Dimensioning**) are listed on the **Tools** submenu.

If a palette is already open, a checkmark appears next to its name in the menu.

To toggle a palette off or on, click its name in the list.

For related information, see the following topics:

2D Tools
3D Tools
Standard Dimension Palette

The **Top Level** command on the **Organize** menu returns you to the main drawing window from within an entered group.

This command can be useful when you are working with an object that is many levels deep.

For related information, see the following topics:

Edit Group
Exit Group

Bitmap images that are in PICT format can be converted to a polyline or a polygon with the **Trace Bitmap** command.

When you select the **Trace Bitmap**, the **Trace Bitmap** dialog box opens. This dialog allows you to choose the degree of accuracy for the trace. You must set the thin line threshold and the collinearity sensitivity for the bitmapped object you want to convert.

Trace Bitmap can take from several seconds to several hours depending on the size of the bitmap and the settings you select. The progress bar can help you predict the total time needed to complete a given trace.

Control-**Period** cancels the trace and returns you to the drawing.

The **Trim** command on the **Tool** menu is used to cut lines with an object.

Note: This command works only with lines. It will not cut objects that have a surface.

Multiple lines can be trimmed.

Any object can be used as a trim object. The trim object cuts all lines that it intersects. When a line is trimmed, it is cut into segments by the trim object. The trimmed portion is not removed.

To trim a line, position the trim object at the point where you want the lines trimmed. Select the line(s) to be trimmed. (Do not select the objects that will trim the lines.)

Choose the **Trim** command.

Position the pointing-finger cursor on the trim object and click the mouse. The line is cut by the trim object. Selection handles appear at the points where the line(s) and the object had intersected.

For related information, see the following topics:

Clip
Clip Surface
Intersect Surface

The **TrueType to Polyline** command on the **Tool** menu converts any TrueType font into a polyline. After conversion, the polyline is no longer a font and can be edited as any other polyline.

TrueType fonts are defined by Bezier curves and arc points. MiniCad uses these font definitions to create the polyline.

The **Conversion Resolution** setting in the **Preferences** dialog box has no effect on the font's conversion to a polyline.

As a polyline, the text can be converted to 3D and positioned in 3D space. It also can be extruded to give it depth.

Each letter of a text string is converted into a separate polyline object. If a polyline is extruded, the number of segments that make up the extrusion is determined by the **3D Conversion Resolution** as set with the **Preferences** command.

Be aware that setting the conversion resolution too high causes slow screen redraws without improving the quality of the font. Conversely, setting the conversion resolution too low changes the appearance of the font.

The **Undo** command on the **Edit** menu cancels the last change made to the drawing.

Undo works only on the last action taken. When another action is taken, even a mouse click in the window, you lose any chance to undo the preceding action.

You cannot undo an action that does not actually change a drawing (e.g., changing a dialog box setting, selecting a tool, resizing a window, changing zoom level).

For related information, see the following topics:

Revert to Saved

Groups can be broken into their component parts with the **Ungroup** command on the **Organize** menu.

Unless a group is ungrouped, the individual objects in the group cannot be selected, moved, or edited.

Each time the command is chosen, a single level of the group is ungrouped. Because groups can be nested into many levels, multiple "ungroups" can be necessary to ungroup the entire nested group.

Nested groups that have been ungrouped completely are usually difficult to "renest." If you ungroup a nested group completely, and draw a marquee around all ungrouped objects and regroup, a single group will be created.

If you want to edit a nested group, while preserving its nested structure, use the **Edit Group** command.

The **Units** command on the **Page** menu controls the unit type (e.g., mm, decimal feet and inches) in your drawing.

The unit setting is universal. Changing the unit setting changes the units in the entire drawing, including dimensions and worksheets.

Note: When files are imported through DXF, the **Units** command automatically configures the MiniCad environment for the combination of drawing size and units in the imported drawing.

When you select the **Units** command, the **Set Unit** dialog box opened. This dialog contains the following options.

Unit Name

Use the **Unit Name** pop-up menu to select the unit of measure for the file. This is a universal setting for the file. It affects the display of all numbers in the drawing, data display bar, dimensions, and worksheets.

Old Style Feet and Inches is the format used in some earlier versions of MiniCad. It is used only to ensure compatibility with older versions MiniCad. Old files that are in feet and inches should be updated to the new format. To update a file, choose **Feet and Inches** from the menu.

Show Unit Mark

Selecting the **Show Unit Mark** checkbox puts the appropriate mark on dimensions. If **Feet and Inches** is selected, the selection is dimmed.

Display As Fractions

When selected, units less than one display as fractions. If deselected, they display as decimals. This option is dimmed if **Old Style Feet and Inches** is selected in the **Unit Name** pop-up menu.

Round Dimensions To

The **Round Dimensions** pop-up menu controls the display of numbers less than one.

Rounding dimensions only affects how numbers are displayed. It does not affect the stored accuracy of the number. Changing the display accuracy will not cause rounding errors.

Decimal Formats

The **Decimal Formats** menu controls how decimals are displayed. This menu also controls where place-holding zeros are displayed. This setting does not affect stored accuracy. If **Display as Fractions** is selected, this menu is not presented.

Angular Accuracy

The **Angular Accuracy** menu affects the display of angles. Angles can be displayed as decimals or as degrees (0°), degrees with minutes (0° 0'), and degrees with minutes and seconds (0° 0' 0"). This setting does not affect stored accuracy.

Custom

Custom units can be configured in the **Custom Units** dialog box. These custom controls can be useful for some users, but be careful. If you experience any problems with custom units, please call Graphsoft's Technical Support Department.

For more information about this dialog box, see **The Custom Units Dialog Box**.

Accuracy

The stored accuracy of a number can be controlled with these fields:

- **Stored** accuracy is the limit placed on the size of a fraction or a number. A number with only one place right of the decimal has a much greater rounding error than a number with nine places.

The setting in this box defines the smallest unit stored in your drawing, and it determines the overall accuracy of your drawing. When setting a decimal value, use a number that is a power of ten. Custom fractions should be a power of 2 (2, 4, 8,16,.... 4096). There are very few reasons why you would want to reduce the stored accuracy in a MiniCad file.

- The **Units Per Inch** field allows for the use of units not available in the standard units menu. For example, to use decimeters as the base unit in the drawing, enter 0.2540 in this box since there are 0.2540 decimeters to an inch.

Defaults

Clicking the **Defaults** button returns to the standard **Units** dialog box. Any changes revert to the default settings.

The controls for setting custom units are provided in the **Custom Units** dialog box. This dialog has the following options:

Format
Controls whether numbers less than one are displayed as decimals or as fractions.

Display
Custom units can be configured to display numbers in different ways. The display fields are:

Unit Mark
Enter the label, up to twelve characters long, to describe units (e.g. mm).

Sq Unit Mark
Enter the label, up to twelve characters long, to describe square units (e.g.,"sq meters").

Min Unit
This field lets you enter the smallest number to be displayed. While there are no "illegal" values for this field, the minimum decimal unit should be divisible by ten and the minimum fraction should be divisible by two.

Angles
The **Angular Accuracy** pop-up menu affects the display of angles. Angles can be displayed as decimals or as degrees (0°), degrees with minutes (0° 0'), and degrees with minutes and seconds (0° 0' 0"). This setting has no effect on stored accuracy.

Leading 0's:
Controls the display of leading zeros. Select if leading zeros are desired.

Trailing 0's:
Controls the display of trailing zeros. Select if trailing zeros are desired.

The **Unlock** command reverses the **Lock** command on selected locked objects.

After "unlocking" an object, it can be moved, deleted, or edited. Layer links, made with the **Create Layer Link** command, are automatically locked. Before a layer can be moved or deleted on a layer link, it must be unlocked.

To **Unlock** an object, select the locked object and choose **Unlock** from the **Edit** menu.

For related information, see the following topics:

Lock

The **Use Full Screen** command expands the active window to the limits of the screen you are using, and it removes the title bar.

Since the close box is unavailable the **Close** command on the **File** menu must be used to close the file.

If the command is enabled, a checkmark is placed next to the command's name on the menu. Choosing the command when it is enabled returns the window to its normal configuration.

The **Worksheets** command on the **Window** menu is used to open a worksheet that that has been created in the drawing.

A worksheet is opened by clicking its name in the **Worksheets** command's submenu. If the worksheet is already open, a diamond appears next to its name.

For more information about using worksheets in a drawing, see the **Resource Palette**.

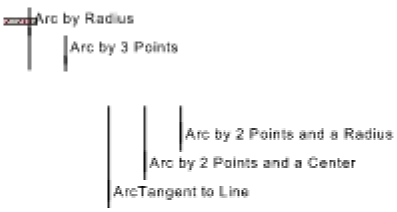
The **Working Planes** command on the **Window** menu is used to open the **Working Plane** palette.

The palette is closed by clicking the close box.

For more information about this palette, see the **Working Plane Palette**.

The **Arc** tool on the **2D Tool** palette draws circular arcs of any degree, at any angle. The area within the arc can have a fill pattern or color, and the arc lines can be of any color (even invisible), thickness or dash pattern.

When the **Arc** tool is selected, the arc icons are displayed in the mode bar. They allow you to define your arc using one of five methods.



If the icon representing the method you want to use is not already selected, click it before beginning to draw your arc.

Note: By double-clicking the **Arc** tool, you can create an arc by entering its parameters directly into the **Create Arc** dialog box.

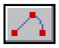
To complete this box, specify the arc's size in the Δx and Δy fields and its sweep in the **Start Angle** and **Arc Angle** fields. In the **Position** fields, enter the arc's XY position, or set the **Next Click** button, and use your mouse to specify the arc's position in the drawing.

For additional information, see the following topics:

Arc By Radius Mode
Arc By Three Points Mode
Arc Tangent To Line Mode
Arc By Two Points and a Center Mode
Arc By Two Points and a Radius Mode
Editing Arcs

For related information, see the following topics:

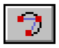
Quarter Arc
Ellipse
Polyline



The **Arc by Radius** button is used to create an arc from a radius that you specify.

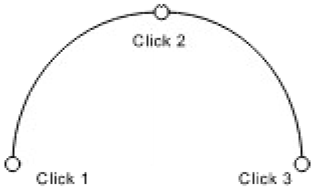
To draw an arc by specifying a radius line:

1. Select the **Arc** tool from the **2D Tool** palette, then select the **Arc by Radius** icon in the mode bar
2. Position the cursor at the arc's center of rotation.
3. Drag the mouse to draw the radius line.
4. When the radius line is the required length, release the mouse button and draw the arc to the desired point by moving the mouse.
5. Click the mouse button to complete the arc.
6. Use the **Data Display** bar to verify the angle of the arc, the cursor's distance from the arc's vertex, its angle, and location on the screen.

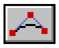


The **Arc by 3 Points** mode creates arcs by selecting three points in the drawing through which the arc must pass.

The first click defines the starting point of the arc. The arc passes through the point created with the second click. The third click defines the end point of the arc.



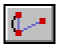
Snap to Objects or **Snap to Intersection** aids in the alignment of the cursor. Use the **Data Display** bar to verify the angle of the arc, and to verify the cursor's distance from the arc vertex, its angle, and location on screen.



The **Arc Tangent to a Line** mode creates an arc that touches, but does not intersect, a line.

To draw an arc tangent to a line:

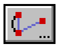
1. Select the **Arc** tool from the **2D Tool** palette, then select the **Arc Tangent to a Line** icon in the mode bar.
2. Place the cursor at the tangent point, then click and hold the mouse button.
3. Drag the mouse to another point to establish a vector for the tangent line, then release the mouse.
4. Move the cursor to the endpoint of the arc and click the mouse button to set the arc.



The **Arc by 2 Points and a Center** mode allows you to set the start and end points of the arc as well as its radius.

To create an arc by two points and a center:

1. Select the **Arc** tool from the **2D Tool** palette, then select the **Arc by 2 Points and a Center** icon in the mode bar.
2. Click the mouse to set the beginning point of the arc.
3. Move (don't drag) the cursor to the end point of the arc.
4. Press and hold the mouse button to set the end point for the arc.
5. With the mouse button still down, drag the cursor to define the arc's radius.
6. When the arc is the right shape, release the mouse button.



The **Arc by 2 Points and a Radius** mode uses the mouse to set the start and end points of an arc, then uses the keyboard to set the arc's radius.

To draw an arc by two points and a radius:

1. Select the **Arc** tool from the **2D Tool** palette, then select the **Arc by 2 Points and a Radius** icon in the mode bar.
2. Click the mouse at the beginning point of the arc.
3. Click the mouse at the end point of the arc.
4. In the **Radius** dialog box, enter the arc's radius. (You may not enter a value less than the minimum shown in the dialog.)

Arcs can be edited with the mouse or with the **Object Info Palette**.

Editing with the Mouse

- To edit an arc with the mouse, select the arc and place the selection cursor on one of the arc's handles. When the resize cursor appears, drag the handle.
- Dragging the center handle edits the arc's radius. Dragging one of the arc's end points edits the sweep.
- The **Data Display** bar will reflect the changes you make to the arc as you move its handles.

Editing with the Object Info palette

- To edit an arc with the **Object Info Palette**, select the arc and enter a new value in the appropriate field in the palette. (If the **Object Info** palette is not currently displayed, select the **Object Info** command.)

The **Quarter Arc** tool on the **2D Tool** palette creates arcs that are one quarter of an ellipse, with an angle limited to 90°.

To create a Quarter Arc:

1. Select the **Quarter Arc** tool from the **2D Tool** palette.
2. Press and hold the mouse button to set the beginning of the arc.
3. Drag the mouse in the direction in which you want the arc to appear.
4. Release the mouse button to set the end of the arc.

The **Data Display** bar displays the ratio of the arc's height to width. A ratio of 1.000, represents a quarter of a circle (i.e., a true arc).

Note: By double-clicking the **Quarter Arc** tool, you can create an arc by entering its parameters directly into the **Create Quarter Arc** dialog box.

To complete this box , specify the arc's size in the Δx and Δy fields, and select the quadrant that you wish to have drawn. In the **Position** fields, enter the arc's XY position, or set the **Next Click** button and use your mouse to specify the arc's position in the drawing.

For additional information, see the following topics:

Editing Quarter Arcs

For related information, see the following topics:

Quarter Arc
Ellipse
Polyline

Quarter Arcs can be edited with the mouse or with the **Object Info** palette.

Important! If you use the mouse to edit a quarter arc, the arc will be converted to a polyline. If you want to maintain the object as a quarter arc, edit it using the **Object Info Palette**.

Editing with the Object Info palette

- To edit a quarter arc with the **Object Info Palette**, select the quarter arc and enter a new value in the appropriate field in the palette. (If the **Object Info** palette is not currently displayed, select the **Object Info** command.)
- Note that the **Object Info** palette treats a circular quarter arc as "arc" and a noncircular quarter arc as a "polyline."

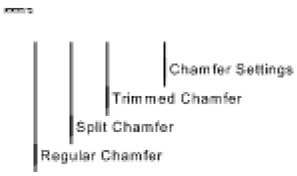
Editing with the Mouse

- To edit a quarter arc with the mouse, select the arc and place the selection cursor on one of the arc's handles. When the resize cursor appears, drag the handle.
- Dragging the center handle edits the arc's radius. Dragging one of the arc's end points edits the sweep.
- As you move the handle, the **Data Display** bar will show the changes to the arc.

The **Chamfer** tool on the **2D Tool** palette is used to place a line between two linear objects, at a specified distance from the intersection of the lines or objects.

The **Chamfer** tool can be used to bevel the sides of rectangles or polygons. It can also be used to chamfer intersecting polylines or line segments (note: parallel lines cannot be chamfered with this tool).

When the **Chamfer** tool is selected, the chamfer icons are displayed in the mode bar. These icons are used to select the type of chamfer you want MiniCad to draw.



To draw a chamfer line:

1. Select the **Chamfer** tool from the **2D Tool** palette.
2. Click the **Chamfer Settings** icon in the mode bar.
3. In the **Chamfer Settings** dialog box, do one of the following:
   - Select **First & Second Line** if you want to define the chamfer in terms of its distance from the intersection of the two lines it will intersect.
   - Select **First Line & Angle** if you want to express the chamfer's position in terms of a specific point and angle on one of the lines.
4. Complete the distance (or distance and angle) information requested in the **Chamfer Settings** dialog, then click **OK.**
5. In the mode bar, select the type of chamfer you want to create.
6. Draw the chamfer line.

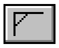
Note: Once you specify values in the **Chamfer Settings** dialog box, you do not have to set them again unless you want to change them. They will automatically be applied to all subsequent chamfer operations.

For additional information, see the following topics:

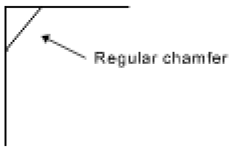
Regular Chamfer
Split Chamfer
Trimmed Chamfer

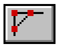
For related information, see the following topics:

Fillet

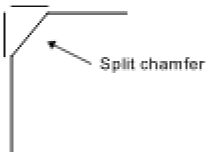


The **Regular Chamfer** mode draws a chamfer as a separate line, but does not affect the object it intersects.

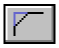




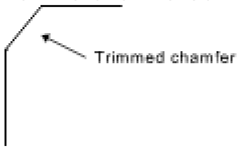
The **Split Chamfer** mode draws a chamfer line between two objects, "splitting off" the lines it intersects.



Note: The **Split Chamfer** mode only works on line segments. Like fillets, the setting is applied to either object independent of the other's type. For example, placing a **Split Chamfer** between a line segment and a polygon will split the line and ignore the polygon.



The **Trimmed Chamfer** mode bevels the edge of a chamfered object by actually changing the endpoints of the chamfered lines.



A trimmed chamfer can be used on lines, polylines, rectangles and polygons. When a rectangle or polygon is chamfered in **Trim** mode, a single object is created with the chamfer as part of the object. If the original object is a rectangle, it becomes a polygon.

The **Clip** tool on the **2D Tool** palette is used to cut objects. It can cut lines, arcs, rectangles, ellipses, polygons, and polylines. This tool is similar to the **Clip Surface** command, but requires fewer steps.

When the **Clip** tool is selected, the clip icons are displayed in the mode bar. These icons are used to select the kind of clipping operation you want to perform.

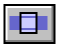
To clip an object:

1. Select the object you want to clip.
2. Select the **Clip** tool from the **2D Tool** palette.
3. In the mode bar, select the type of clip operation you want to perform.
4. Drag a marquee with the cursor. When you release the mouse button, the specified clipping operation is performed.

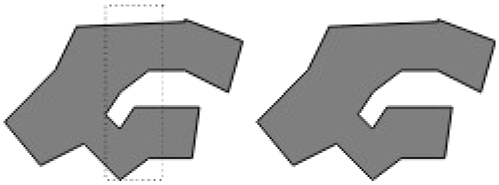
Note: If you clip a full arc or a circle, the resulting object is a polygon (the object is no longer an arc or a circle). If you clip a quarter arc or oval, the result is a polyline.

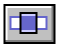
For additional information, see the following topics:

Remove Inside Mode
Remove Outside Mode
Split at Boundaries Mode

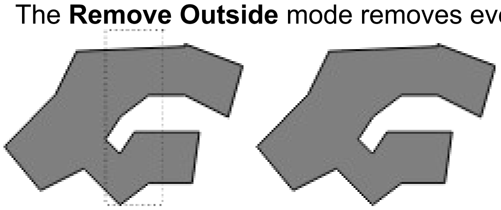


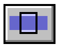
The **Remove Inside** mode removes everything within the marquee.



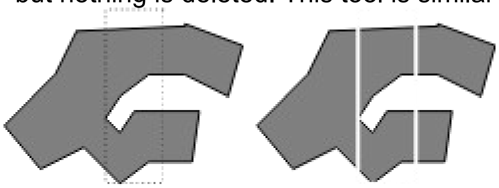


The **Remove Outside** mode removes everything outside of the marquee.





The **Split at Boundaries** mode cuts the object along the marquee's perimeter. The objects are selected but nothing is deleted. This tool is similar to the **Trim** command.



The **Duplicate Along Path** tool on the **2D Palette** is used to replicate an object (or group of objects) at regular intervals along a predefined path, or along a path that you specify with the mouse.

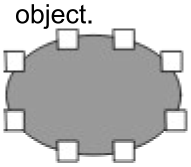
Note: This tool lies beneath the **Mirror 2D** tool. To access the **Duplicate Along Path** tool, press and hold the **Mirror 2D** icon. When the pop-out icons appear, release the mouse over the **Duplicate Along Path** tool.

When the **Duplicate Along Path** tool is selected, the duplicate icons are displayed in the mode bar. These icons are used to specify duplication parameters and to specify which type of path the duplicated objects will be place on.

For additional information, see the following topics:

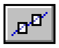
Object Path Duplication
Path Duplication

The **Object Path** mode is used to duplicate an object along a path defined by the perimeter of another object.



To copy an object along a predefined path:

1. Select the object(s) to be copied.
2. Select the **Duplicate Along Path** tool in the **2D Tools** palette.
3. Select the **Object Path Duplication** icon in the mode bar.
4. Select the **Duplicate Preferences** icon in the mode bar.
5. Set the appropriate parameters in the **Duplicate Along Path** dialog box.
   - If you want to create an object at specific intervals, select **Fixed Distance** and specify the distance that will be used to separate one copy from another.
   - If you want to create a certain number of objects, equally spaced on the path, select **Number of Duplicates,** and specify the number of copies that will be created.
   - If you want the copies placed tangent to the object rather than intersecting it, set **Tangent to Path.**
6. Click **OK** to close the dialog box.
7. Click the point at which you want the first copy placed.
8. If you selected **Number of Duplicates** above, the copies will automatically appear around the object when you click the start point.
9. If you selected **Fixed Distance**, drag the cursor along the object for the distance that you want objects copied. Release the mouse to end the operation.



The **Path Duplication** mode is used to duplicate an object along a path defined by the mouse.

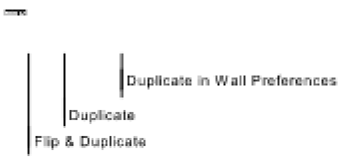
To copy an object along a path drawn with the mouse:

1. Select the object(s) to be copied.
2. Select the **Duplicate Along Path** tool in the **2D Tools** palette.
3. Select the **Path Duplication** icon in the mode bar.
4. Select the **Duplicate Preferences** icon in the mode bar.
5. Select the **Fixed Distance** option and specify the distance that will separate one copy from another.
6. Click **OK** to close the dialog box.
7. Click at the path's start point and at each vertex on the path.
8. Double-click to end the operation.

The **Duplicate Symbol in Wall** tool on the **2D Tool** palette is used to replicate the active symbol (typically a door or window) at a specified interval in a wall.

Note: This tool is only available when the **AEC Overlay** is active. If the **Duplicate Symbol In Wall** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

When the **Duplicate Symbol in Wall** tool is selected, the duplicate icons are displayed in the mode bar. These icons are used to specify duplication parameters, and to specify whether the symbol will be duplicated with or without first flipping it.



To duplicate the active symbol in a wall:

1. Select the **Duplicate Symbol in Wall** tool in the **2D Tools** palette.
2. If you want the symbol to be flipped before it is placed in the drawing, select the **Flip & Duplicate** icon in the mode bar. Otherwise, select the **Duplicate** icon.
3. Select the **Duplicate in Wall Preferences** icon in the mode bar.
4. In the **Duplicate Symbol in Wall** dialog box, enter the distance from the beginning of the wall and the first symbol, or select the **Next Click** button to use the mouse to determine this distance.
5. In the **Spacing** field, enter the distance between the duplicated symbols, or select the **Next Click** button to use the mouse to specify this distance.
6. If you want the symbol duplicated a certain number or times, enter a value in the **Copies** field.
7. Click **OK** to close the dialog box.
8. Click the wall.

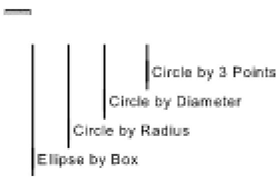
For related information, see the following topics:

Symbols and Walls
Duplicate Along Path

The **Ellipse** tool on the **2D Tool** palette is used to draw circles and ellipses.

The area within the figure can have a fill pattern or color, and the figure's lines can be of any color (even invisible), thickness or dash Pattern.

When the **Ellipse** tool is selected, the ellipse icons are displayed in the mode bar. They allow you to define your figure using one of four methods.



If the icon representing the method you want to use is not already selected, click it before you begin to draw your figure.

Note: By double-clicking the **Ellipse** tool, you can create an ellipse or circle by entering its parameters directly into the **Create Oval** dialog box.

To complete this box, specify the figure's size in the **Δ*x*** and **Δ*y*** fields. In the **Position** fields, enter the figure's XY position, or set the **Next Click** button, and use your mouse to specify the figure's position in the drawing.

For additional information, see the following topics:

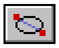
Ellipse by Box Mode

Circle by Radius Mode

Circle by Diameter Mode

Circle by Three Points Mode

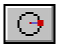
Editing an Ellipse or a Circle



The **Ellipse by Box** mode lets you create a circle or ellipse by drawing a bounding box.

To constrain the figure to a circle, you can:

- Hold the **Shift** key while drawing the bounding box, or
- Select the **Symmetrical Constraint** icon on the **Constraint Palette**.

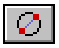


The **Circle by Radius** mode lets you draw a circle from its center point.

To create a circle:

1. Select the **Ellipse** tool in the **2D Tools** palette.
2. Select the **Circle by Radius** icon in the mode bar.
3. Drag the mouse in any direction to create the circle.
4. When the circle is the correct size, release the mouse to set the circle into the drawing.

Note: The **L** field in the **Data Display** reports the length of the radius as you draw the circle. You may also type the radius value directly into the **Data Display** bar.

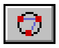


The **Circle by Diameter** mode lets you create a circle by defining its diameter.

To create a circle:

1. Select the **Ellipse** tool in the **2D Tools** palette.
2. Select the **Circle by Diameter** icon in the mode bar.
3. Drag the mouse in any direction to create the circle.
4. When the circle is the correct size, release the mouse to set the circle into the drawing.

Note: The **L** field in the **Data Display** reports the length of the diameter as you draw the circle. You may also type the diameter directly into the **Data Display** bar.



The **Circle by Three Points** mode lets you create a circle by selecting the circle's coordinates with the mouse.

Three points must be selected with the mouse. This tool uses the bull's-eye cursor.

The **L** field in the **Data Display** reports the distance from point to point. The **A** field reports the angle from point to point. You may type these values directly into the **Data Display** bar.

An ellipse (or circle) can be edited with the mouse or with the **Object Info** palette.

Editing with the Mouse

- To edit an ellipse with the mouse, select the figure and place the selection cursor on one of the figure's handles. When the resize cursor appears, drag the handle.
- The **Data Display** bar will reflect the changes you make to the figure as you move its handles.

Editing with the Object Info palette

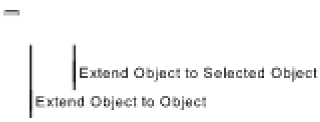
- To edit an ellipse with the **Object Info Palette**, select the figure and type new values into the appropriate fields of the palette. (If the **Object Info** palette is not currently displayed, open it with the **Object Info** command.)

The **Extend** tool on the **2D Tool** palette lengthens or "extends" one object to meet another object. This tool can extend any line, arc, open polygon, or open polyline to any selected line, arc, rectangle, polyline, ellipse, or polygon.

Note: This tool lies beneath the **Trim** tool on the **2D Tool** palette. To access the **Extend** tool**,** press and hold the **Trim** icon. When the pop-out icons appear, release the mouse over the **Extend** tool.

Note: The **Extend** tool will extend an object only if, when lengthened, it intersects the target object.

When the **Extend** tool is selected, the extend icons are displayed in the mode bar. These icons represent the two ways in which you can extend an object.



Extending an Object by Dragging it to the Target

1. Select the **Extend Object to Object** icon in the mode bar.
2. Click on the object to be extended.
3. Position the pointer on the object and drag it to the target.
4. Release the mouse button.
5. The object is extended to intersect the target.

Extending an Object to the Nearest Selected Object

1. Select the target object(s)
2. Select the **Extend Object to Selected Object(s)** icon in the mode bar.
3. Click on the object to be extended.

   The object is extended to intersect the nearest surface of the target.

The **Eyedropper** tool on the **2D Tool** palette is used to "pick up" attributes from one object and apply them to another. It can also be used to change the current default values.

When the **Eyedropper** tool is selected, the eyedropper icons appear in the mode bar. The **Pick up** and **Put Down** icons are used specify which operation will be performed--a "pick up" or an "apply."

The **Preferences** icon is used to specify which attributes are applied to an object. It is also used to specify whether the attributes become the drawing's new default values.

Note: You can use the **Control** key to toggle between **Pick Up Attribute** and **Put Down Attribute** mode while you are using this tool.

For additional information, see the following topics:

Pick Up Attributes
Put Down Attributes
Eyedropper Preferences

The **Pick Up Attributes** mode is used to copy an object's attributes into the eyedropper. It can also be used to automatically set the default attributes from those of an existing object.

Note: All attributes are copied by the **Eyedropper** tool. You may selectively apply these attributes to other objects by selecting them with the **Eyedropper Preferences** button.

To pick up an object's attributes:

1. Select the **Eyedropper** tool in the **2D Tool** palette.
2. Select the **Pick Up Attributes** icon in the mode bar.
3. If you want the attributes that are copied by this tool to automatically become new default values, select the **Eyedropper Preferences** icon in the mode bar and enable the **Pick Up Sets Defaults** option.
4. Click **OK** to close the dialog box.
5. Click the object whose attribute you want to copy.

   If the **Use Sound** option is on, you will hear a "slurping" sound as the attributes are picked up.

The **Put Down Attributes** mode is used to apply attributes from the eyedropper to an object.

Note: Before using this tool you must have already "picked up" attributes from an object. See **Pick Up Attributes**.

To apply attributes with the Eyedropper:

1. Select the **Eyedropper** tool in the **2D Tool** palette.
2. Select the **Eyedropper Preferences** icon in the mode bar.
3. In the **Pick Up/Put Down Filter** dialog box, select the attributes you want to apply, then close the dialog box.
4. Select the **Put Down Attributes** icon in the mode bar.
5. Place the cursor on the object to which you want attributes applied. When you are over an object that can accept the attributes, your cursor will turn to an inverted paint can.
6. Click the mouse.

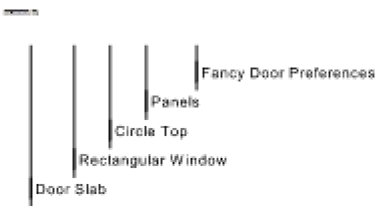
The **Eyedropper Preferences** tool is used to specify which attributes will be applied with the **Put Down Attributes** tool.

When you select this tool, the **Pick Up/Put Down Filter** dialog box is displayed. Select the attributes that you want applied to an object when the **Put Down Attributes** tool is used. Click **OK** to close the dialog box. The choices you've made will remain in effect until you use this tool to change them again.

The **Fancy Door** tool on the **2D Tool** palette is used to create a door in a front elevation. The door can have raised panels, rectangular windows, and a semicircular top window.

Note: This tool is only available when the **AEC Overlay** is active. If the **Fancy Door** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

When the **Fancy Door** tool is selected, the door icons are displayed in the mode bar. These icons are used to specify the door's components and to specify various options.



To create a door using the Fancy Door tool:

1. Select the **Fancy Door** icon.
2. Click the **Fancy Door Preferences** icon in the mode bar.
3. In the **Fancy Door** dialog box, enter a value for the frame's thickness and any bevel for the panels and windows.
4. If the door will have panels, enter the number of rows and columns it will have.
5. Click **OK** to close the dialog box.
6. Click the **Door Slab** icon in the mode bar.
7. Draw the door.

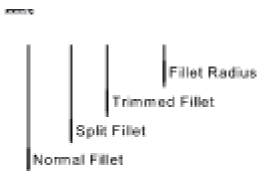
   Refer to the **Data Display** bar if you want to see the size of the door as you are drawing it.
8. If the door will have windows or panels, click the appropriate icon in the mode bar to create them.

Note: The finished door is a 2D object. You may use it in this form, even in a 3D elevation view, however, it won't project in isometric views. In order to have it project properly, it must be converted to a polygon using the **Convert to 3D Polygons** command.

The **Fillet** tool on the **2D Tool** palette is used to places an arc (fillet) between two objects, making the arc tangent to each object at the fillet's end points.

The **Fillet** tool can be used between lines, boxes, polygons, polylines, circles (but not ovals), and arcs. It can also fillet adjacent sides of rectangles, polygons, or polylines. (Parallel or concentric objects cannot be filleted.)

When the **Fillet** tool is selected, the fillet icons are displayed in the mode bar. These icons are used to select the type of fillet you want MiniCad to draw.



To draw a fillet:

1. Select the **Fillet** tool from the **2D Tool** palette.
2. Click the **Fillet Radius** icon in the mode bar.
3. In the **Fillet Settings** dialog box, set the fillet's radius.
4. Click **OK** to close the dialog box.
5. In the mode bar, select the type of fillet you want to create.

   tip! If you are using **Normal** or **Trim** mode, and you want to fillet all vertices of a box, polygon, or polyline, just click the figure and skip the following steps.

6. Position the crosshair cursor over the first object of the pair to be filleted.
7. The cursor will appear as a crosshair cursor only when it is over an object that can be filleted.
8. Drag the cursor to the second object.
9. Release the mouse button.

Note: In some cases (particularly those involving two circular objects), the fillet radius setting may be too small or too large for the fillet requested. When this happens, MiniCad warns you that the fillet cannot be placed, and ignores the command.

The **Freehand Line** tool on the **2D Tool** palette produces a polygon along a path that is specified by dragging the mouse.

Note: This tool creates a polygon, not a polyline. Curves are composed of line segments, whose number and size are determined by the curvature of the shape and the speed of the mouse. A fast-moving mouse dragged in a straight line creates fewer vertices than one moved slowly in a very tight arc.

To create a Freehand Line

1. Select the **Freehand Line** tool from the **2D Tool** palette.
2. Position the cursor at the beginning of the line you want to draw.
3. Press and hold the mouse button while you draw the line.
4. When you are finished drawing the line, release the mouse button.

   Once created, a **Freehand Line** can be reshaped or edited using the **2D Reshape** tool or the **Object Info Palette**.

The **Single Line** tool on the **2D** palette is used to create constrained and unconstrained single lines.

When the **Single Line** tool is selected from the tool palette, the line icons are displayed in the mode bar. These icons allow you to create your line in constrained or unconstrained mode.

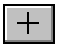
To draw a line:

1. Select the **Single Line** tool from the **2D Tool** palette.
2. In the mode bar, select the type of line you want to draw--constrained or unconstrained.
3. Position the cursor at the beginning of the line.
4. The way in which you draw the line will depend on how you have configured your copy of MiniCad with the **Preferences** command:
   - If it is in **Click-Drag** mode, click and hold the mouse. Drag it to the end of the line, then release the mouse button.
   - If it is in **Click-Click** mode, click the mouse button. Then, move the cursor to the end of the line and click again.

Note: By double-clicking the **Single Line** tool, you can create a line by entering its parameters directly into the **Create Line** dialog box. Or, you can select the **Single Line** tool and enter it's dimensions directly into the **Data Display** bar.

For additional information, see the following topics:

Constrained Lines Mode
Unconstrained Lines Mode
Editing a line



The **Constrained Lines Mode** is used to create lines at angles of 30°, 45°, 90°, and their complements.

Note: Additional angle constraints can be set with the **Preferences** command.

The **Unconstrained Lines** mode is used to draw a line at any angle.

- If you want to constrain a line drawn in **Unconstrained Line** mode, hold the **Shift** key while drawing the line.
- If the **Constrain Angle** constraint is enabled in the **Constraint Palette**, it will override the unconstrained line mode, forcing lines to be constrained.

A line can be edited with the mouse or with the **Object Info** palette.

Editing with the Mouse

- To edit a line, select it, then place the selection cursor on one of its handles and drag it.
- The **Data Display** bar will reflect the changes you make to the line as you move its handle.

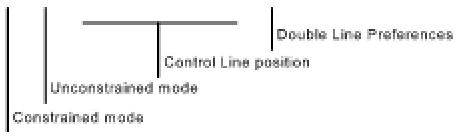
Editing with the Object Info palette

- To edit a line with the **Object Info** palette, select the line and type new values into the appropriate fields of the palette. (If the **Object Info** palette is not currently displayed, open it with the **Object Info** command.)

The **Double Line** tool on the **2D** palette is used to create constrained and unconstrained double lines.

Note: This tool lies beneath the **Single Line** tool on the **2D Tool** palette. To access the **Double Line** tool, press and hold the **Single Line** icon. When the pop-out icons appear, release the mouse over the **Double Line** tool.

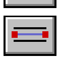
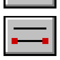
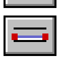
When the **Double Line** tool is selected from the tool palette, the line icons are displayed in the mode bar. They allow you to create your line in constrained or unconstrained mode. They also specify the way in which the line is drawn.



To draw a line:

1. Select the **Double Line** tool from the **2D Tool** palette.
2. Select the **Double Line Preferences** icon in the mode bar, and, in the **Double Line Preferences** dialog box, specify the characteristics of your line (cavity lines are defined with this dialog).
3. In the mode bar, select the control line position. The control line is the invisible line on which the pair is aligned.

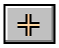
   The control line defines the double-line's length and position. It is the line that is aligned on the drawing's snap points. It is also the line from which the line's thickness is measured (e.g. if the double line is 4" thick, and a center control line is specified, the lines are drawn 2" from either side of the control line).

Select...	If you want...
	The control line to follow the top line of the pair.
	The control line to follow the center of the pair.
	The control line to follow the bottom of the pair.
	The control line to follow a specific offset that you have specified in **Double Line Preferences**.

4. In the mode bar, select the type of line you want to draw--**constrained** or **unconstrained**.
5. Position the cursor at the beginning of the line.
6. The way in which you draw the line will depend on how you have configured your copy of MiniCad with the **Preferences** command:
   - If it is in **Click-Drag** mode, click and hold the mouse. Drag it to the end of the line, then release the mouse button.
   - If it is in **Click-Click** mode, click the mouse button. Then, move the cursor to the end of the line and click again.

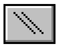
For additional information, see the following topics:

Constrained Double Line Mode
Unconstrained Double Line Mode
Double Line Preferences
Mitering Double Lines



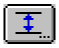
The **Constrained Double Line Mode** is used to create lines at angles of 30°, 45°, 90°, and their complements.

Note: Additional angle constraints can be set with the **Preferences** command.



The **Unconstrained Double Line** mode is used to draw a line at any angle.

- If you want to constrain a line drawn in unconstrained mode, hold the **Shift** key while drawing the line.
- If the **Constrain Angle** constraint is enabled in the **Constraint Palette**, it will override unconstrained line mode, forcing lines to be constrained.



The **Double Line Preferences** icon is used to specify characteristics of a double line. When you select this button, the **Double Line Preferences** dialog box opens.

Note: This dialog will automatically open if you attempt to draw a double line and **Double Line Preferences** have never been set.

This dialog box contains the following options:

Separation Field
The **Separation** field controls the distance between the two lines as they are drawn.

Control Off Field
The **Control Offset** field determines the offset of the control line. This value is used when the **Offset Control Line** option is selected on the mode bar.

Create Lines
Creates a double line.

Create Polygons
Creates a closed polygon instead of a double line.

Create Lines and Polygons
Creates a double line and a polygon.

Cavity Lines
Displays the **Cavity Line** dialog box, which is used to create a cavity line. A cavity line is a line that is automatically drawn between the pair of lines. A double line may have more than one cavity line.

To create a cavity line:

1. Click the **New Cavity** button.
2. Specify the position of the line by entering its offset from the center of the double line.
3. Enter the line's weight (thickness) and dash pattern in the **Line** field.
4. If you want a filled line, select the **Filled Cavity** check box, then enter a value in the **Width** field and select a pattern in the **Fill** field.
5. Repeat these steps for each cavity line you want to create.

Double lines can be drawn with mitered ends. The miters are made by toggling the **Tilde** (~) and **Control** keys in different combinations.

Note: If you want to join double lines, it is easier to use the **Join** command rather than mitering their corners. Corner mitering is primarily provided for compatibility with older versions and for use in exploded details.

The **Control** key controls the starting point of the selected line. The default is no miter.

- The first time the **Control** key is pressed the miter is cut from left to right.
- The next time the **Control** key is pressed, the miter is cut from right to left.
- The next time the **Control** key is pressed, the default (no miter) is selected.
- Each time the **Control** key is pressed, it toggles once through the cycle.

The **Tilde** key controls the end point of the selected line. Each time the **Tilde** key is pressed, it toggles once through the cycle at the end point.

The **Leader Line** tool on the **2D** palette is used to create a 2-segment, leader line. A leader line can be constrained or unconstrained.

Note: This tool lies beneath the **Single Line** tool on the **2D Tool** palette. To access the **Leader Line,** press and hold the **Single Line** icon. When the pop-out icons appear, release the mouse over the **Leader Line** tool.

When the **Leader Line** tool is selected, the leader-line icons appear in the mode bar.

To draw a Leader Line:

1. Select the **Leader Line** tool from the **2D Tool** palette.
2. In the mode bar, select the type of line you want to draw--constrained or unconstrained.

   Note: Both segments of a "constrained" **Leader Line** are restricted to angles of 30°, 45°, 90°, or their complements.
3. Position the cursor at the beginning of the line.
4. Drag the cursor to the end of the first segment.
5. Release the mouse.
6. Position the second segment with the mouse.
7. Click the mouse to set the line in the drawing.

The second segment of the **Leader Line** is automatically given a marker (e.g., an arrowhead). This marker can be changed using the **Attribute Palette**.

The **Locus 2D** tool on the **2D Tool** palette is used to place a 2D reference point (represented by a diagonal cross) on the drawing.

A locus has the following uses:

- It serves as a "point of reference."
- It enables the **Snap to Objects** constraint to snap to a locus.
- It aligns a group of objects to a specific position when used with the **Align Objects** command
- It serves as the center of rotation when included in a group of rotated objects
- When **Snap to Loci** is set, it creates a vertical and horizontal "snap line" from each locus on the active layer to which the cursor snaps.

A locus has no dimension, only a location (the tools on the **Constraint Palette** can help in precisely placing a locus). It can be moved but it cannot be reshaped.

A locus, like any other object, can be cut, copied and pasted through the Clipboard.

For related information, see the following topics:

3D Locus

The **2D Mirror** tool on the **2D Tool** palette is used to make a mirror image of a selected object.

When the **2D Mirror** tool is selected, the mirror icons appear in the mode bar. They determine whether the selected object is flipped (mirrored) or whether a mirror-image copy is made of it.

For additional information, see the following topics:

Mirror Mode

Mirror/Duplicate Mode

For related information, see the following topics:

Rotate

Rotate 2D

3D Mirror

The **Mirror 2D** mode flips the selected object on a specified axis. The axis is used to designate the offset, if any, of the mirrored object.

To mirror an object

1. Select the object you want to flip.
2. Select the **2D Mirror** tool from the **2D Tool** palette.
3. Select the **Mirror** icon in the mode bar.
4. Drag a line to designate the axis around which the object will be flipped.

   Note: By drawing this line outside the object, you can offset the mirrored result from the object's current position.

5. Release the mouse button.

   The object is mirrored.

The **Mirror/Duplicate** mode creates a mirror-image copy of the selected object.

To mirror/duplicate an object:

1. Select the object you want to copy and flip.
2. Select the **2D Mirror** tool from the **2D Tool** palette.
3. Select the **Mirror/Duplicate** icon in the mode bar.
4. Drag a line to designate the axis around which the copied object will be flipped.

   Note: By drawing this line outside the object, you can offset the copy from the original.

5. Release the mouse button.

   The mirrored copy is produced.

The **Move Page** tool on the **2D Tool** palette is used to move the drawing around the drawing page. It is often used to align the drawing within the print boundary.

Note: **Move Page** physically moves drawing on the page. If you just want to move the view of the page, use the **Pan** tool.

Note: This tool lies beneath the **Pan** tool on the **2D Tool** palette. To access the **Move Page,** press and hold the **Pan** icon. When the pop-out icons appear, release the mouse over the **Move Page** tool.

To move the drawing page, choose the **Move Page** tool, then drag the drawing boundary to a new location.

The old location is shown for reference as the drawing is moved.

For related information, see the following topics:

Shift Drawing

The **Number Stamp** tool on the **2D Tool** palette is used to place a marker with an identifying number on the drawing. The number automatically increments each time the tool is used.

Note: This tool is only available when the **AEC Overlay** is active. If the **Number Stamp** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

When the tool is selected, the **Number Stamp** preferences icon appears in the mode bar. This icon is used to specify the stamp's size, shape and content in the **Number Stamp Preferences** dialog box. (You can also open this dialog by double-clicking the **Number Stamp** icon.)

A number stamp can also be attached to a database record. This allows you to use the number stamp to specify objects and symbols in a schedule.

To assign a database record to the **Number Stamp** tool, select the **Number Stamp Preferences** icon and click the **Attach** button. When the dialog box opens, specify the record and fields you want to associate with the stamp, then click **OK**.

Now, each time you use the stamp, the selected record will automatically be attached to the marker.

For related information, see the following topics:

Record Formats

The **Offset** tool on the **2D Tool** palette is used to create a new object whose outline is at a constant distance from the selected object.

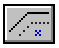
The **Offset** tool can be used with lines, arcs, rectangles, polylines, ovals, and polygons to create concentric figures.

Note: This tool lies beneath the **Trim** tool on the **2D Tool** palette. To access the **Offset** tool**,** press and hold the **Trim** icon. When the pop-out icons appear, release the mouse over the **Offset** tool.

When the **Offset** tool is selected, the offset icons appear in the mode bar. They are used to specify the offset position, and to position the object using one of two methods.

For additional information, see the following topics:

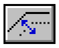
Offset by Distance Mode
Offset to a Point Mode



The **Offset by Distance** mode is used to position the offset at a specified distance from another.

To create an offset by specifying distance:

1. Select an object to offset.
2. Select the **Offset** tool from the **2D Tool** palette.
3. Select the **Offset by Distance** icon in the mode bar.
4. Select the **Offset Preferences** icon in the mode bar.
5. In the **Offset Preferences** dialog box, enter the distance by which the new object will be offset.
6. Click **OK** to close the dialog box.
7. Place the cursor anywhere along the side of the object on which you want the offset object placed.
8. Click the mouse.



The **Offset to a Point Mode** is used to specify the distance between the original object and its offset, using the mouse.

To create an offset by selecting a point:

1. Select an object to offset.
2. Select the **Offset** tool from the **2D Tool** palette.
3. Select the **Offset to a Point** icon in the mode bar.
4. Place the cursor over the position at which you want the offset object drawn.
5. Press and hold the mouse button.

   A shadow of the offset shape appears, which can be moved by dragging the mouse.
6. When the shadow object is in the correct position, release the mouse button.

   The offset object is place in the drawing.

The **Pan** tool is used to move the view of the active drawing. It is useful for fine positioning or making large movements. If is usually a much faster way of moving around a drawing than using the scroll bars.

Note: The **Pan** tool can also be used to redraw the screen. If the **Pan** tool is double-clicked, the screen will redraw. The screen also redraws if the mouse is clicked in the drawing window while the **Pan** tool is selected.

When the **Pan** tool is used, the drawing itself is unchanged-only the view is changed. This is unlike the **Move Page** tool, which physically moves the drawing on the page.

To use the Pan tool:

1. Select the **Pan** tool from the **2D** or **3D Tool** palette.
2. Drag the mouse to move the view of the drawing around the screen.
3. The **Pan** tool will remain active until another tool is selected.

Note: A drawing also can be panned using the arrow keys. When an arrow key is pressed, the drawing view moves half a screen in the direction of the arrow.

The **Parking Space** tool on the **2D Tool** palette is used to create evenly-spaced lines for parking lots.

Note: This tool is only available when the **AEC Overlay** is active. If the **Parking Space** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

To use the Parking Lot tool:

1. Select the **Parking Space** tool from the **2D Tool** palette.
2. Select the **Parking Space Preferences** iconin the mode bar.
3. In the **Parking Space Tool** dialog box, enter the specifications for your parking spaces. If the **Maximum Width** option is enabled, the individual space widths are widened to completely fill a row if the row cannot accommodate another full space.
4. Click **OK** to close the dialog box.
5. Drag the mouse the specify the location of the stripes.

The **2D Polygon** on the **2D Tool** palette is used to create polygonal shapes. A polygon can contain segments of any length or number. It can have as few as two vertices. It can be an open or closed shape. The polygon, whether open or closed, can have a fill.

To create a single line polygon:

1. Select the **2D Polygon** tool from the **2D Tool** palette.
2. Place the cursor at the polygon's starting point.
3. Click the mouse.
4. Move the cursor to the next vertex (hold the **Shift** key if you want to constrain the line).
5. Click the mouse.
6. Repeat the last two steps for each vertex. Double-click the last vertex to end the operation.

Note: By double-clicking the **2D Polygon** tool, you can create a polygon by entering parameters for each vertex directly into the **Create Polygon** dialog box. Enter the parameters for the first vertex, then click **Add** to create a new vertex.

For related information, see the following topics:

Editing a Polygon
3D Polygon
Extruded Polygon

A polygon size can be edited by using the mouse. Its size and shape can be modified with the **2D Reshape** tool or the **Object Info Palette**.

Editing with the Mouse

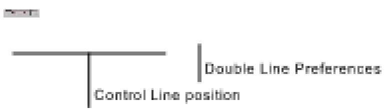
- To edit a polygon's size, select it, then place the selection cursor on one of its handles and drag it to a new position.
- The **Data Display** bar will reflect the changes you make to the polygon as you move its handle.

The **Double Line Polygon** tool on the **2D Tool** palette is used to create double line polygons with the option of having lines only, polygons only, or both lines and polygons.

The polygon can contain segments of any length or number. It can have as few as two vertices. It can be an open or closed shape. The polygon, whether open or closed, can have a fill. Additionally, the space between the double lines can have a fill, or can contain a cavity line.

Note: This tool lies beneath the **2D Polygon** tool on the **2D Tool** palette. To access the **Double Line Polygon** tool, press and hold the **2D Polygon** icon. When the pop-out icons appear, release the mouse over the **Double Line Polygon** tool.

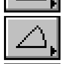
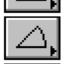
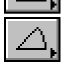
When the **Double Line Polygon** tool is selected from the tool palette, the polygon icons are displayed in the mode bar. These icons are used to specify the way in which double lines are drawn.



To draw a Double Line Polygon:

1. Select the **Double Line Polygon** tool from the **2D Tool** palette.
2. Select the **Double Line Preferences** icon in the mode bar, and, in the **Double Line Preferences** dialog, specify the characteristics of the double line.
3. In the mode bar, select the control line position. The control line is the invisible line on which the pair is aligned.

   The control line defines the double-line segment's length and position. It is the line that is aligned on the drawing's snap points. It is also the line from which the double line's thickness is measured (e.g. if the double line is 4" thick, and a center control line is specified, the lines are drawn 2" from either side of the control line.

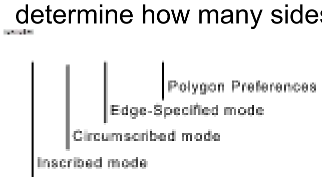
Select...	If you want...
	The control line to follow the top line of the pair.
	The control line to follow the center of the pair.
	The control line to follow the bottom of the pair.
	The control line to follow a specific offset that you have set in **Double Line Preferences**

4. Place the cursor at the polygon's starting point.
5. Click the mouse.
6. Move to the cursor to the next vertex (hold the **Shift** key if you want to constrain the line).
7. Click the mouse.
8. Repeat the last two steps for each vertex. Double-click the last vertex to end the operation.

The **Regular Polygon** tool on the **2D Tool** palette is used to create a closed polygon whose sides are equal in length.

Note: This tool lies beneath the **2D Polygon** tool on the **2D Tool** palette. To access the **Regular Polygon** tool, press and hold the **2D Polygon** icon. When the pop-out icons appear, release the mouse over the **Regular Polygon** tool.

When the **Regular Polygon** tool is selected, the polygon icons appear in the mode bar. These icons determine how many sides the polygon has, and how it is drawn.

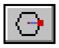


If the icon representing the method you want to use is not already selected, click it before beginning to draw your figure.

tip! The **U** key can be used to toggle among the polygon drawing icons. The **I** key can be used to select the **Polygon Preferences** icon.

For additional information, see the following topics:

Inscribed Mode
Circumscribed Mode
Edge-Specified mode



The **Inscribed** mode lets you draw a regular polygon from center point to vertex.

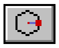
To create an inscribed polygon:

1. Select the **Regular Polygon** tool in the **2D Tools** palette.
2. Click the **Polygon Preferences** icon in the mode bar.
3. In the **Regular Polygon Settings** dialog box, enter a value from 3 to 4000, representing the number of sides the polygon will have.
4. Click **OK** to close the dialog box.
5. Select the **Inscribed Polygon** icon in the mode bar.
6. Drag the mouse in any direction to create the polygon.

   tip! The **Control** key can be used to toggle between inscribed and circumscribed modes while drawing the polygon.

7. When the figure is the correct size, release the mouse to set it into the drawing.

Note: The **L** field in the **Data Display** reports the length of the radius as you draw the polygon. The radius of an **Inscribed Polygon** reflects the distance from the center of the polygon to a vertex. You may also enter the radius value directly into the **Data Display** bar.



The **Circumscribed Polygon** mode lets you draw a regular polygon that is defined by a radius that extends to the midpoint of a side.

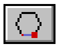
To create a circumscribed polygon:

1. Select the **Regular Polygon** tool in the **2D Tools** palette.
2. Click the **Polygon Preferences** icon in the mode bar.
3. In the **Regular Polygon Settings** dialog box, enter a value from 3 to 4000, representing the number of sides the polygon will have.
4. Click **OK** to close the dialog box.
5. Select the **Circumscribed Polygon** icon in the mode bar.
6. Drag the mouse in any direction to create the polygon.

   tip! The **Control** key can be used to toggle between inscribed and circumscribed modes while drawing the polygon.

7. When the figure is the correct size, release the mouse to set it into the drawing.

Note: The **L** field in the **Data Display** reports the length of the radius as you draw the polygon. The radius if a **Circumscribed Polygon** reflects the distance from the center of the polygon to the midpoint of a side. You may also enter the radius value directly into the **Data Display** bar.



The **Edge Specified** mode lets you create a polygon by drawing one of its sides.

To create an Edge Specified polygon:

1. Select the **Regular Polygon** tool in the **2D Tools** palette.
2. Click the **Polygon Preferences** icon in the mode bar.
3. In the **Regular Polygon Settings** dialog box, enter a value from 3 to 4000, representing the number of sides the polygon will have.
4. Click **OK** to close the dialog box.
5. Select the **Edge Specified** icon in the mode bar.
6. Drag the mouse in any direction to draw a line representing one side of the polygon. You may rotate the polygon as it is being drawn.

   tip! The **Control** key can be used to flip the polygon over the defining line as you are drawing it.

7. When the figure is the correct size, release the mouse to set it into the drawing.

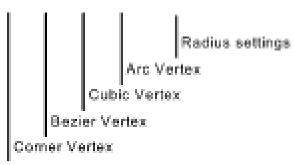
The **Polyline** tool on the **2D Tool** palette is used to create a connected series of arcs, curves and lines that are treated as a single entity. Polylines can be drawn with any line weight, color, or dash pattern. A polyline can also have a fill pattern.

Like a polygon, a polyline can be closed or opened. However, polylines can have any number of control points in addition to, or in place of, polygonal vertices. These control points can be one of three types:

- Bezier Point
- Cubic Spline
- Arc Radius

Occasionally a polyline is converted into a polygon, such as when the **Smoothing** command's **No Smoothing** option (i.e., corner point option) is used.

When the **Polyline** tool is selected, the polyline icons appear in the mode bar. These icons are used to create polylines with different types of control points.



Note: By double-clicking the **Polyline** tool, you can create a polyline by entering the parameters for each vertex directly into the **Create Polyline** dialog box. Select a curve type and enter the parameters for the first vertex, then click **Add** to create a new vertex.

Curves and arcs follow very rigid rules regarding how they are constructed and where their tangent points are placed.

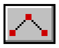
Each curve segment needs three points to be drawn. If a curve segment is described with three points; labeled *Point 1*, the *Control Point*, and *Point 2,* the curve is drawn so that, at *Point 1* and *Point 2,* the curve is tangent to the line extending from the *Control Point*. In the case of an arc, a line segment can be added to satisfy this condition.

If the curve borders a control point, *Point 1* and *Point 2* are set to the previous and next vertex on the polyline. If the curve borders another curve on either side, *Point 1* and/or *Point 2* are set to a point midway between the bordering control point and the *Control Point* itself.

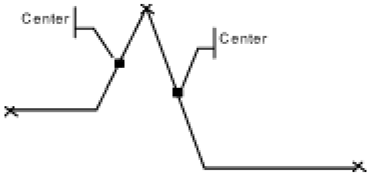
Note: In an open polyline, the end point can be a curve point. When this is the case, the bordering curve segment starts at the midway between its control point and the end point. This occurs because the last curve segment is always a line that extends from the end point to the mid point.

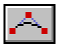
For additional information, see the following topics:

Corner Vertex
Bezier Vertex
Cubic Vertex
Arc Vertex
Editing Polylines



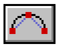
The **Corner Vertex** icon is used to create a polyline with corner point controls.





The **Bezier Vertex** icon is used to create a polyline with Bezier controls.

A **Bezier** control point is the vertex. It does not touch the surface of the polyline. Instead, it pulls the curve toward the point.

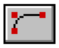


The **Cubic Vertex** icon is used to create a polyline with **Cubic** control points.

A **Cubic** control point produces a curve that passes through the control point. This curve is similar, but not identical to, a cubic polynomial curve fitting.

MiniCad automatically calculates a **Bezier** curve to pass through a particular point on the curve. A phantom control point exists for each **Cubic** curve, which can be seen by selecting the **Cubic Vertex**.

Hint! When using cubic segments, it is often useful to surround them with **Bezier** segments. **Bezier** segments have the useful property of being able to set the tangent of the curve at the beginning and end.



The **Arc Vertex** icon is used to create a polyline with **Arc Radius** control points.

An arc radius control point is identical to a fillet placed at the vertex. The curve follows a circle of a preset radius, which joins the lines approaching from either side of the vertex. If the fillet radius is set to 0 then the largest fillet possible is inserted.

To specify the arc's radius, select the **Radius Settings** icon in the mode bar.

With arcs the control point is the vertex.

A polyline's size can be edited with the mouse. Its size and shape can be modified with the **2D Reshape** tool or the **Object Info Palette**. It can also be edited with the **Smoothing** command.

Editing with the Mouse

- To edit a polyline's size, select it, then place the selection cursor on one of its handles and drag it to a new position.
- The **Data Display** bar will reflect the changes you make to the polyline as you move its handle.

The **Property Bounds** tool on the **2D Tool** palette is used to create a polygon based on bearing and distance values. The boundary line can be made of line segments or curves.

Note: This tool is only available when the **AEC Overlay** is active. If the **Property Bounds** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

To draw a property boundary line:

1. Select the **Property Bounds** tool from the **2D Tool** palette.
2. Click in the drawing at the first coordinate.
3. In the, **Create Property Bounds** dialog box, enter the data for the first segment. Select either the **Line** or **Curve** button, and enter the distance and bearing in the appropriate fields.
4. Click the **Apply** button to add the line segment to the boundary.
5. Repeat the last two steps for each segment in the boundary.

   Note: If you need to remove a segment you've already added, highlight it and click **Remove.**
6. When all coordinates have been entered, click **Done**.

The **Rectangle** tool on the **2D Tool** palette is used to create a rectangle or a square.

A rectangle is created by selecting the **Rectangle** tool, then dragging the mouse diagonally in the drawing. You can constrain the figure to a square by holding the **Shift** key as you draw it, or by setting the **Constrain Symmetrical** option in the **Constraint Palette.**

The area within the figure can have a fill pattern or color, and the figures lines can be of any color (even invisible), thickness or dash pattern.

Note: By double-clicking the **Rectangle** tool, you can create a rectangle or square by entering its parameters directly into the **Create Rectangle** dialog box.

To complete this box, specify the figure's size in the Δx and Δy fields. In the **Position** fields, enter the figure's XY position, or set the **Next Click** button, and use your mouse to specify the figure's position in the drawing.

You can also create a rectangle by selecting the **Rectangle** tool and entering its dimensions directly into the **Data Display** bar.

For related information, see the following topics:

Editing Rectangles
Regular Polygon
Rotated Rectangle
Rounded Rectangle

A rectangle can be edited with the mouse or with the **Object Info** palette.

Editing with the Mouse

- To edit a rectangle, select it, then place the selection cursor on one of its handles and drag it.
- The **Data Display** bar will reflect the changes you make to the line as you move its handle.

Editing with the Object Info palette

- To edit a rectangle with the **Object Info Palette**, select the rectangle and type new values into the appropriate fields of the palette. (If the **Object Info** palette is not currently displayed, open it with the **Object Info** command.)

The **Rotated Rectangle** tool on the **2D Tool** palette is used to create a rectangle at any angle. (The object created by this tool is actually a polygon, not a rectangle.)

Note: This tool lies beneath the **Rectangle** tool on the **2D Tool** palette. To access the **Rotated Rectangle**, press and hold the **Rectangle** icon. When the pop-out icons appear, release the mouse over the **Rotated Rectangle** tool.

To create a rotated rectangle:

1. Select the **Rotated Rectangle** tool from the **2D Tool** palette.
2. Position the mouse at one corner of the rectangle.
3. Drag the mouse the angle and length of one side.
4. Release the mouse.
5. Move the mouse to specify the length and position of the perpendicular side.
6. Click to finish the operation.

For related information, see the following topics:

Editing Rotated Rectangles
Rectangle
Regular Polygon
Rounded Rectangle

Because a rotated rectangle is actually a polygon, not a rectangle, it is edited like a polygon.

A polygon's size can be edited with the mouse. Its size and shape can be modified with the **2D Reshape** tool or the **Object Info Palette**.

Editing with the Mouse

- To edit a polyline's size, select it, then place the selection cursor on one of its handles and drag it to a new position.
- The **Data Display** bar will reflect the changes you make to the polyline as you move its handle.

The **Rounded Rectangle** tool on the **2D Tool** palette is used to create rectangles with rounded corners. The arcs forming the corners of the rectangle can be proportional, symmetrical, or both.

The area within the figure can have a fill pattern or color, and the figures lines can be of any color (even invisible), thickness or dash pattern.

Note: This tool lies beneath the **Rectangle** tool on the **2D Tool** palette. To access the **Rounded Rectangle**, press and hold the **Rectangle** icon. When the pop-out icons appear, release the mouse over the **Rounded Rectangle** tool.

To create a Rounded Rectangle:

1. Select the **Rounded Rectangle** tool from the **2D Tool** palette.
2. Select the **Rounded Rectangle Preferences** icon in the mode bar.

   tip! You can select this icon from the keyboard by pressing the **U** key.
3. In the **Rounded Rectangle Preferences** dialog box, select the type of corner you want to the rectangle to have.
   - If you want the corner to be drawn proportionally to the length of each side (diameter of the arc = 1/3 side length), check **Proportional Corners.**
   - If you want the corner arc to be drawn symmetrically (i.e. with equal length sides), check **Symmetrical Corners.**
   - If you want the same-size horizontal and vertical arc lengths, check **Symmetrical Corners** and **Proportional Corners.**
   - If you want to specify your own arc parameters, leave **Symmetrical Corners** and **Proportional Corners** unchecked, and type your values directly into the **X Diam** and **Y Diam** fields.
4. Click **OK** to close the dialog box.
5. Position the cursor at one corner of the rectangle.
6. Drag the mouse diagonally to form the rectangle.

   You can constrain the figure to a rounded square by holding the **Shift** key as you draw it, or by setting the **Constrain Symmetrical** option in the **Constraint Palette**.
7. When the rectangle is the correct size and shape, release the mouse button.

Note: By double-clicking the **Rounded Rectangle** tool, you can create the figure by entering its parameters directly into the **Create Rounded Rectangle** dialog box.

To complete this box, select a corner type, then specify the figure's size in the Δx and Δy fields. In the **Position** fields, enter the figure's XY position, or set the **Next Click** button, and use your mouse to specify the figure's position in the drawing.

You can also create a rectangle by selecting the **Rectangle** tool and entering its dimensions directly into the **Data Display** bar.

For related information, see the following topics:

Editing Rounded Rectangles

Rectangle

Regular Polygon

Rotated Rectangle

A rounded rectangle can be edited with the mouse or with the **Object Info** palette.

Editing with the Mouse

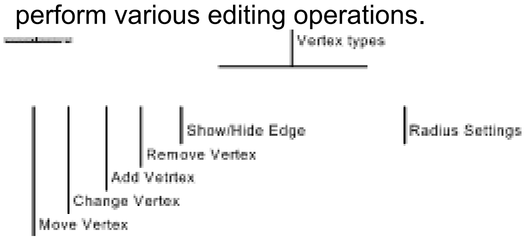
- To edit a rounded rectangle, select it, then place the selection cursor on one of its handles and drag it.
- The **Data Display** bar will reflect the changes you make to the figure as you move its handles.
- In addition to the standard corner handles, a rounded rectangle has an additional handle inside it. This is the reshape handle. The corner diameters of a rounded rectangle can be edited by dragging the reshape handle.
- The **Data Display** bar will reflect the current arc size when you drag the reshape handle.

Editing with the Object Info palette

- To edit a rectangle with the **Object Info Palette**, select the rectangle and type new values into the appropriate fields of the palette. (If the **Object Info** palette is not currently displayed, open it with the **Object Info** command.)

The **2D Reshape** tool on the **2D Tool** palette is used to edit polygons and polylines, including freehand lines. It is also used to "hide" a segment of a polygon or polyline.

When the **2D Reshape** tool is selected, the edit icons appear in the mode bar. These icons are used to perform various editing operations.



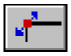
The first group of icons represent editing operations. If the icon representing the editing operation you want to use is not already selected, click it before beginning to edit your figure.

tip! The **U** key can be used to toggle among the first group of icons. The **I** key can be used to toggle among the second group of icons. The **O** key can be used to select the **Set Arc Radius** icon.

When the **2D Reshape** tool is active, selected objects are displayed in "reshape mode." In reshape mode, an object's vertices and mid-points are shown rather than its bounding-box handles. (An object that cannot be reshaped with this tool is displayed with the standard bounding box handles.)

For additional information, see the following topics:

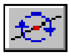
Move Vertex Mode
Change Vertex Mode
Add Vertex Mode
Delete Vertex Mode
Show/ Hide Edge



The **Move Vertex** mode is used to reposition a polyline vertex.

When the **Move Vertex** icon is selected, the cursor changes to the resize cursor when it is over a vertex or mid-point of a selected polygon or polyline. The vertex can be moved by dragging it with the mouse.

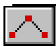
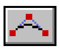
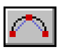
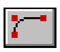
Note: .You can move multiple vertices at once by drawing a marquee around them. Any vertices within this marquee can be moved as the marquee is manipulated.



The **Change Vertex** mode is used to change a vertex to another type (i.e., Corner, Bezier, Cubic or Arc).

To change a vertex type:

1. Select the object you want to edit.
2. Select the **2D Reshape** tool from the **2D Tool** palette.
3. Select the **Change Vertex** icon in the mode bar.
4. Select one of the **Vertex** icons in the mode bar.

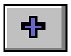
Select...	If you want to...
	Convert the vertex to a Corner Point
	Convert the vertex to a Bezier control point
	Convert the vertex to a Cubic control point
	Convert the vertex to an Arc control point

5. If you selected an **Arc** vertex above, click the **Radius Settings** icon in the mode bar and specify the radius value to be used for the for arc. Then, click **OK** to close the dialog box.
6. In the drawing, click the vertex you want to change.

Note: If you change the vertex of a polygon it will automatically be converted to a polyline (a polygon contains only corner points).

For related information, see the following topics:

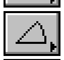
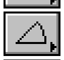
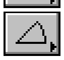
Smoothing



The **Add Vertex** mode is used to add a vertex to a polyline or polygon.

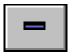
To add a vertex:

1. Select the object you want to edit.
2. Select the **2D Reshape** tool from the **2D Tool** palette.
3. Select the **Add Vertex** icon in the mode bar.
4. Select one of the vertex icons in the mode bar.

Select...	If you want to...
	Add a Corner Point vertex
	Add a Bezier control point
	Add a Cubic control point
	Add an Arc control point

5. If you selected an **Arc** vertex above, click the **Radius Settings** icon in the mode bar and specify the radius value to be used for the for arc. Then, click **OK** to close the dialog box.
6. In the drawing, click on any point between two vertices.

   The new vertex can be dragged into position as it is created. If it is not moved, the vertex is added precisely at the mid-point between the two existing vertices.



The **Delete Vertex** mode is used to remove a vertex from a polyline or polygon.

To delete a vertex:

1. Select the object you want to edit.
2. Select the **2D Reshape** tool from the **2D Tool** palette.
3. Select the **Delete Vertex** icon in the mode bar.
4. In the drawing, click the vertex you want to remove.

The **Show/Hide Edge** icon is used to selectively make segments of a polyline (or polygon) invisible.

To Hide a segment in a Polygon or Polyline:

1. Select the object you want to edit.
2. Select the **2D Reshape** tool from the **2D Tool** palette.
3. Select the **Show/Hide Edge** icon in the mode bar.
4. In the drawing, click the edge (or segment) you want to hide.

   If a segment of a polyline curve is hidden, the entire curve is hidden. If an edge of a polygon is hidden, the object is converted into a polyline.

To Unhide a segment in a Polygon or Polyline:

To unhide an edge (or curve) that has been hidden, follow the procedure above, selecting the hidden segment in step 4.

The **Resize** tool on the **2D Tool** palette is used to scale an object. It resizes all vertices using a fixed point on the drawing as the fulcrum. If the fixed point is at the exact center of the object, the reshape is performed symmetrically. If the fixed point is anywhere but the center, the object is skewed as it is resized.

Note: This tool lies beneath the **2D Reshape** tool on the **2D Tool** palette. To access the **Resize** tool, press and hold the **2D Reshape** icon. When the pop-out icons appear, release the mouse over the **Resize** tool.

To resize an object:

1. Select the object.
2. Select the **Resize** tool from the **2D Tool** palette.
3. Click a point on the drawing. The point can be anywhere on the drawing.
4. Drag the mouse to resize the object.

The **Revision Cloud** tool on the on the **2D Tool** palette is used to create a revision cloud. A revision cloud is typically used to mark changes that have been made to a set of plans.

Note: This tool is only available when the **AEC Overlay** is active. If the **Revision Cloud** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

When the **Revision Cloud** tool is selected, the drawing-mode icons appear in the mode bar. These icons determine which method is used to draw a revision cloud.

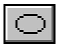
Revision Cloud by Polygon

Revision Cloud by Oval

For additional information, see the following topics:

Revision Cloud by Oval

Revision Cloud By Polygon

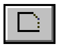


The **Revision Cloud by Oval** mode creates a revision cloud by drawing an oval.

To create an oval revision cloud:

1. Select the **Revision Cloud** tool from the **2D Tool** palette.
2. Select the **Revision Cloud by Oval** icon in the mode bar.
3. Place the cursor in the drawing window where you want the revision cloud to appear.
4. Drag an oval, moving the mouse as needed to form the general shape of the cloud.
5. When the cloud's size and position are correct, release the mouse button.

   The cloud is formed.



The **Revision Cloud by Polygon** mode creates a revision cloud that can be any polygonal shape.

To create an polygonal revision cloud:

1. Select the **Revision Cloud** tool from the **2D Tool** palette.
2. Select the **Revision Cloud by Polygon** icon in the mode bar.
3. Place the cursor in the drawing window where you want the revision cloud to appear.
4. Click the mouse to set the first point.
5. Move the mouse to the second point and click.
6. Continue clicking to set the points that form the cloud.
7. When you've clicked the last point, position the cursor on the start point and click once to finish the cloud.

The **Rotate 2D** tool on the **2D Tool** palette is used to rotate a selected object. Rotation occurs in the screen plane.

To rotate a selected object:

1. Select the object(s) to be rotated.
2. Select the **Rotate 2D** tool from the **2D Tool** palette.
3. Position the cursor at the point around which you want the object to pivot.
4. Press the mouse and draw a line to serve as a fulcrum for the rotation.
5. Release the mouse when the fulcrum line is properly positioned.
6. Move the mouse in the direction you want the object to rotate.
7. When the object is correctly positioned, click the mouse to end the operation.

For related information, see the following topics:

Rotate
Rotate 3D
2D Mirror

The **2D Selection** tool on the **2D Tool** palette is used to select a 2D object in the drawing window. In general, an object must be "selected" before any action can be taken upon it.

To select a single object:

1. Select the **2D Selection** tool.
2. Position the mouse over the object.
3. Click the mouse.

   Handles are displayed at the corners of the object's bounding box to show that it is "selected."

To select a multiple objects using the Shift key method:

1. Select the **2D Selection** tool.
2. Position the mouse over the first object.
3. Click the mouse.
4. Press and hold the **Shift** key.
5. Position the mouse over the next object.
6. Click the mouse.
7. While continuing to hold the **Shift** key down, repeat the last two steps until all objects in the group are selected.

To select multiple objects using the Marquee method:

1. Select the **2D Selection** tool.
2. Place the cursor near the group of objects to be selected.
3. Drag the selection arrow diagonally across the object or group of objects.
4. A marquee is drawn around the objects. Objects completely encompassed by the marquee are selected.

   Note: All of an object's handles must be encompassed by the marquee in order for it to be selected. Keep in mind that some objects have handles that may not lie in close proximity to the visible object (polylines for example).

   If you want to select all objects that the marquee touches (rather than encompasses), hold the **Control** key while drawing the marquee. This lets you select partially-captured objects.

   ---

5. If you want to deselect an object, hold the **Shift** key and click the object. If you want to deselect the entire group, click on an empty portion of the drawing.

For additional information, see the following topics:

Smart Cursor

The **Selection** arrow can change as circumstances warrant.

For example:

- The **Selection** arrow changes to the **Move** cursor when it passes over an object. You can move the object by dragging it.
- The **Selection** arrow changes to the **Resize** cursor when it is placed over a handle of a selected object. Dragging the handle with the mouse scales the object horizontally and/or vertically.

  If you have configured MiniCad for **Eight Selection Handles** with the **Preferences** command, objects will have mid-point handles in addition to corner-point handles. When a mid-point is dragged, only that handle's dimension is changed.

The **Shear** tool on the **2D Tool** palette skews all of the vertices of a rectangle, polygon or ellipse using a fixed point on the drawing as a fulcrum.

Note: This tool lies beneath the **2D Reshape** tool on the **2D Tool** palette. To access the **Shear** tool, press and hold the **Reshape** icon. When the pop-out icons appear, release the mouse over the **Shear** tool.

To shear an object:

1. Select the object you want to shear.
2. Select the **Shear** tool from the **2D Tool** palette.
3. Click on a point anywhere in the drawing.
4. Drag the mouse to shear the object.
5. Release the mouse when you are satisfied with the object's shape.

Note: Rectangles and ellipses are converted to polygons when they are sheared.

The **Shutter** tool on the **2D Tool** palette is used to create window shutters in front elevation.

Note: This tool is only available when the **AEC Overlay** is active. If the **Shutters** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

To create a shutter:

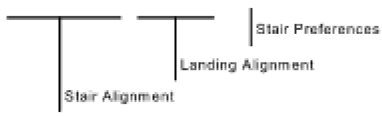
1. Select the **Shutter** tool in the **2D Tool** palette.
2. In the **Shutter** dialog box, enter the frame width and the slat height.
3. Click **OK** to close the dialog box.
4. Position the cursor on the drawing and drag a rectangular shape.

Note: The finished shutter is a 2D object. You may use it in this form, even in a 3D elevation view, however, it won't project in isometric views. In order to have it project properly in a 3D view, it must be extruded or converted to a 3D polygon.

The **Straight Stair** tool on the **2D Tool** palette is used to create stairs. It can be used to make stairs with an open or closed riser. The stairs can be 2D, 3D or both, and they can show the required ceiling clearance.

Note: This tool is only available when the **AEC Overlay** is active. If the **Straight Stair** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

When the **Straight Stair** tool is selected, the stair icons appear in the mode bar. These icons are used to select stair alignment, landing alignment and other specifications.



tips! Keep the following information in mind when creating stairs:

- The **Stair** tool must be used in **Top/Plan** view. If the tool is selected when the drawing is in any other view, you will automatically be switched to **Top/Plan**
- The number of risers in the staircase is determined by both the tread and the floor-to-floor dimension that the stairs will span. Both of these values are useful to have at hand when using the **Straight Stair** tool.
- The dimension of the riser and the number of treads are reported in the **Data Display** bar when the stairs are created. If the riser dimension and tread count needs to be noted on the floorplan, make note of them before releasing the mouse.
- If the stairs have both a 2D and 3D component, it is a good idea to convert the combined staircase into a symbol. As a symbol, the staircase becomes a hybrid object that will appear correctly in any 2D or 3D view.

For additional information, see the following topics:

Stair Preferences

The **Stair Preferences** icon is used to specify parameters for the **Straight Stair** tool. When this icon is selected, the **Straight Stair Preferences** dialog box is opened.

Note: This dialog will automatically open if you attempt to create stairs and **Straight Stair Preferences** have never been set.

This dialog box contains the following options:

2D Checkbox
Creates a 2D stair.

3D Checkbox
Creates a 3D stair.

Ascending and Descending Buttons
Specifies the direction of a set of 3D stairs

Square Landing
Creates a landing using the dimensions entered in the **Stair Width** field.

Draw 2D Stair Break
Draws a stair break. Because a staircase can traverse from one floor to another on a floorplan, a standard drawing technique is to draw a staircase only partially, and show the continuation with a stair break.

Open Risers
Creates stairs with open risers. When selected, the fields for tread and stringer thickness become active.

Denote Head Clearance
Activates the fields for the minimum clearance and ceiling height. Because code requirements vary, be sure the values entered in these fields are correct for the particular stairs being drawn.

Tread
Used to specify the dimension of the stair's tread.

Fit to Height
Calculates the number of risers required to create the staircase. When this checkbox is selected, the **Height** field becomes active. The value in this field is the dimension taken from the top of the floor at the first level, to the top of the floor of the next level.

Two additional fields, for the minimum and maximum dimension of the riser, are displayed when **Fit to Height** is selected. These values are used in conjunction with the values entered in the **Tread** and **Height** fields to calculate the number of treads in the staircase.

Stair Width
Used to specify the width of the stair (i.e., distance from stringer to stringer). This value is also used for the landing, if the **Square Landing** checkbox is selected.

Starting Z
Used to specify the point in 3D space where the stairs originate. Generally, **Starting Z** will be zero.

The **Insert 2D Symbol** tool on the **2D Tool** palette is used to place symbols in the drawing in 2D or Plan projection.

When the **2D Symbol** tool is selected, each click of the mouse places a copy of the active symbol on the drawing. (The active symbol is specified in the **Resource Palette**).

Note: Every symbol has an insertion point; which was specified when the symbol was created. When the symbol is placed in the drawing, this point is aligned on the point that you select with your mouse.

A symbol can be rotated as it is inserted by moving the mouse. After a symbol has been placed, you may rotate it with the **Rotate 2D** tool, or move it with the mouse or the **Move** command.

For related information, see the following topics:

Symbols and Walls

Resource Palette

Symbol Edit

Insert 3D Symbol

If a symbol is placed in a wall, the symbol is automatically inserted into the wall with its insertion point on the wall's control line. The wall is automatically cut at the insertion point to fit the symbol, and the symbol rotates to match the angle of the wall.

A symbol can be removed by selecting it and dragging it away from the wall. If a symbol is dragged out of a wall, the cut in the wall is automatically sealed.

A symbol can be relocated in the wall by selecting it and dragging it to a new location. The old insertion point is sealed, and the new one is created automatically.

A symbol can be rotated to one of four positions by dragging the mouse as the symbol is inserted. This rotation can only be done at insertion however.

If a rotated symbol is relocated to another part of the wall, it retains its rotation angle, unless it's insertion point is on one side and it is inserted into a wall in the opposite direction. Such a symbol can be rotated at insertion to orient it correctly. However, if such a symbol is dragged away from the wall and then dragged back, it reverts to its original orientation. When this occurs, drag the symbol away from the wall, delete it, then reinsert it rotating it to the proper orientation.

Symbols are created with a specific Δz. If you need to change the Δz position of one specific instance of a symbol, use the **3D Reshape** tool. This tool lets you change the selected symbol's Δz value without affecting other instances of the symbol in the drawing.

You can switch to **2D/Plan** projection to select a symbol in a wall. This is often easier than trying to select the symbol in a 3D projection.

For related information, see the following topics:

Insert 2D Symbol
Insert 3D Symbol
Resource Palette
Symbol Edit

The **Symbol Paint** tool on the **2D Tool** palette is used to place multiple copies of the active symbol in the drawing by dragging the mouse. (The active symbol is specified in the **Resource Palette**).

The symbol is inserted with its standard insertion point. Once a symbol has been inserted, it can be moved or rotated with the standard object-manipulation tools.

Note: This tool lies beneath the **Insert 2D Symbol** tool on the **2D Tool** palette. To access the **Symbol Paint** tool**,** press and hold the **Insert 2D Symbol** icon. When the pop-out icons appear, release the mouse over the **Symbol Paint** tool.

The **Snap to Grid** constraint on the **Constraint Palette** must be enabled to use the **Symbol Paint** tool because it is used to control the placement of the symbols. For example, if the grid is set to 1/8", a symbol is placed at 1/8" intervals along the path over which the mouse is dragged.

The **Symbol Paint** tool can be used in one of two modes.

- One mode places symbols and ignores any already-placed symbols on the drawing.
- The other mode replaces any already-placed symbol if the cursor moves near the center of the existing symbol's bounding box.

The **Text** tool on the **2D Tool** palette is used to place text anywhere on a drawing. A text block can have multiple fonts, and those fonts can have multiple sizes, colors, case, and styles.

Text can be rotated to any angle (this works best with PostScript and TrueType fonts--substantial quality loss usually occurs with other fonts). Text can also be scaled. If you want text to be scaled like other objects in the layer, set the **Scale Text** option with the **Layers** command.

To add text to a drawing:

1. In the **Text** menu, select the character font, size and style you want to use (these attributes will become the default).
2. Select the **Text** tool from the **2D Tool** palette.
3. Move the cursor to the drawing.

   The I-beam cursor appears.
4. Position the cursor at the point where text will begin and do one of the following:
   - If you need to type only a few words (less than a line or two), simply click the point at which you want text to begin and start typing. When you need to begin a new line, press **Enter**.
   - If you need to enter several lines of text, drag the cursor diagonally to create a "text block." Release the mouse when the block is approximately the size you need (you can easily change its size later). Type your text in the block. Your words will automatically wrap to fit within the block.
5. To end text mode, click anywhere in the drawing, or select another tool.

   Handles are placed around the text. The text object may now be moved, reshaped or rotated.

For additional information, see the following topics:

Editing Text

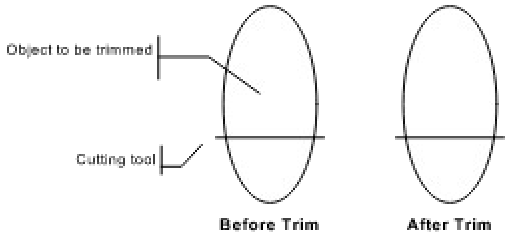
The **Text** tool on the **2D Tool** palette is used to edit text in a text box.

To edit text in a drawing:

1. Select the text block you want to edit.
2. Select the **Text** tool from the **2D Tool** palette.
3. Place the cursor over the text block.
4. Click the mouse.

   The text block is placed in "edit mode," and you can add, delete or modify text within it.
5. To end text mode, click anywhere in the drawing, or select another tool.

The **Trim** tool on the **2D Tool** palette is used to remove a portion of an object using another object as a cutting tool. Lines, arcs, rectangles, polylines, ovals, and polygons can be trimmed, or they can be used as cutting objects.



To trim an object:

1. Select the cutting object(s).

   Important! The cutting object(s) must intersect the object to be trimmed.

2. Select the **Trim** tool from the **2D Tool** palette.
3. Place the cursor over the portion of the object to be removed.
4. Click the mouse.

   The piece between the object's boundary and the cutting tool is removed. If the object is intersected by two cutting objects, the piece between the cutting objects is removed.

For related information, see the following topics:

Clip
Clip Surface
Trim
Intersect Surface

The **Wall** tool on the **2D Tool** palette is used to create walls. Walls can have a fill and can contain a cavity line. Sections of wall can be joined, capped or trimmed.

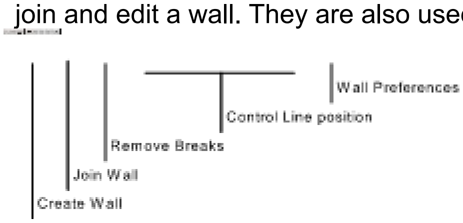
Note: This tool is only available when the **AEC Overlay** is active. If the **Wall** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

The **Wall** tool is a hybrid tool. It creates objects in 2D that are seen as 3D objects when viewed in a 3D orthogonal projection. A wall appears as a 3D object, and can be displayed as wireframe or rendered solidly with a color or fill. Unless the wall is given a custom Δ*z*, the wall has the default Δ*z* for the layer.

A wall can be resized or reshaped, and a peak can be added with the **3D Reshape** tool.

Note: You should consider using the **Wall** tool, even if you generally view your floor plan only in 2D. Walls have special properties that comparable objects (e.g. double line objects) can't provide. For example, when a window or door symbol is inserted into a wall, a hole is automatically cut to receive it. If the symbol is subsequently removed, the hole is automatically sealed.

When the **Wall** tool is selected, the wall icons appear in the mode bar. These icons are used to create, join and edit a wall. They are also used to specify the way in which the wall is drawn.



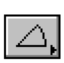
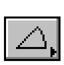
tip! The **U** key can be used to toggle among the first group of icons. The **I** key can be used to toggle among the second group of icons. The **O** key can be used to select the **Wall Preferences** icon.

To create a wall:

1. Place the drawing in **2D/Plan** projection (the **Wall** tool must be used in **2D/Plan** view).
2. Select the **Wall** tool from the **2D Tool** palette.
3. Select the **Wall Preferences** icon in the mode bar, and specify the characteristics of the wall line.
4. In the mode bar, select the control line position. The control line is the invisible line on which the wall is aligned.

   The control line defines the wall's length and position. It is the line that is aligned on the drawing's snap points. It is also the line from which the wall's thickness is measured (e.g. if the wall is 4" thick, and a center control line is specified, the lines are drawn 2" from either side of the control line).

   The control line is also the line on which symbols are aligned when they are placed in the wall.

Select...	If you want...
	The control line to follow the top line of the wall.
	The control line to follow the center of the wall.
	The control line to follow the bottom of the wall.
	The control line to follow a specific offset

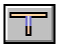
that you have set in **Wall Preferences.**

4. Select the **Create Wall** icon in the mode bar.
5. Position the cursor where you want the wall to begin.
6. Drag the mouse to form the wall.
7. When the wall is the correct length, release the mouse button.
8. The wall is placed in the drawing. A wall is a "special group" of objects, and is treated differently than other objects.
    - Wall are always are restricted to the vertical axis of the working plane.
    - Walls are a "special group" of objects. Do not ungroup them. Once a wall is ungrouped it is no longer a wall, and there is no way to convert it back into a wall.
    - Walls are drawn with an invisible control line, whose position is established when the wall is created. All measurements involving a wall (e.g., area, perimeter, length, and width) are taken from the control line.
    - The control lines of all walls must be considered when grouped walls are reshaped. Unexpected results can occur.
    - Because the control line is used as the insertion point for symbols, the wall's thickness must be considered when the insertion point is set for the symbol. To place symbols properly within the wall, you can specify a user-defined control line, or adjust the symbol's insertion point.
    - Walls have a "direction," which is determined by the direction the mouse was dragged when the wall was created (or by its start point, if the wall was created by a dialog box).

      This characteristic affects the placement of symbols having an insertion point on one side (it has no effect on an insertion point **At Object Center**). A symbol with an insertion point on the left side of the symbol can be inserted into a wall with the insertion point on the right side. This reversal of insertion point is because the wall was drawn in the opposite direction of the insertion point. If the symbol flips when inserted, rotate the symbol at insertion to the proper orientation.

For additional information, see the following topics:

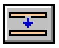
Join Wall
Remove Break
Wall Preferences



The **Join Wall** icon is used to connect wall segments by dragging a line from one wall segment to another. It automatically determines the type of joint to form (i.e., whether to create a "T" "L" or "X" join).

The **Join Wall** icon works the same as the **Join** **command**.

Note: If you have multiple walls converging at odd angles, they may not join properly using the **Wall** or **Butt-Join Wall** tools. In this event, use the **Y-Join Wall** tool to join the walls



The **Remove Break** mode cleans up a wall after editing.

By dragging a marquee around the affected area, the previous joins are cleaned, breaks caused by symbols are removed, and any caps at the ends of walls are removed.

The **Wall Preferences** icon is used to specify parameters for the **Wall** tool. When this icon is selected, the **Wall Preferences** dialog box is opened.

This dialog box contains the following options:

Separation

The **Separation** value specifies the overall thickness of the wall. You must enter the thickness of the whole wall (e.g., for an exterior wall add the siding, sheathing, stud wall and interior wallboard). This insures that your 3D models are accurately proportioned.

Control Off field

The **Control Offset** field determines the offset of the control line. This value is used when the custom control line button is selected on the mode bar (i.e., the last button in the second group).

Caps

The **Caps** option specifies whether the wall's ends are capped. The choices are: **None**, **Start**, **End**, and **Both**.

Type

The **Type** option specifies the type of wall cap to be used, **Flat** or **Round**.

Cavity Lines

Displays the **Cavity Line** dialog box, which is used to create a cavity line. A cavity line is a line that is automatically drawn within the wall. A wall may have more than one cavity line.

To create a cavity line:

1. Click the **New Cavity** button.
2. Specify the position of the line by entering its offset from the center of the wall.
3. Enter the line's weight (thickness) and dash pattern in the **Line** field.
4. If you want a filled line, select the **Filled Cavity** check box, then enter a value in the **Width** field and select a pattern in the **Fill** field.
5. Repeat these steps for each cavity line you want to create.

The **Butt-Join** tool on the **2D Tool** palette is used to join walls with corners butted together instead of mitered.

Butt-joined walls reflect the corner construction of actual walls in a structure. When walls are butt-joined, takeoffs are more accurate, as the true length of the bearing walls are always known. Butt-joined walls insure that beams and other structural members are sized correctly.

Note: This tool is only available when the **AEC Overlay** is active. If the **Butt-Join Wall** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

To use the Butt-Join tool:

1. Select the **Butt-Join** tool from the **2D Tool** palette.
2. Drag the mouse from the non-bearing wall to the bearing wall.
3. If the walls have no caps, select the bearing walls and open the **Object Info Palette**. In the **Object Info** palette, choose **Both** in the **Caps** menu, and **Flat** in the **Type** menu.

   Note: If you have multiple walls converging at odd angles, they may not join properly using the **Wall** or **Butt-Join Wall** tools. In this event, use the **Y-Join Wall** tool to join the walls

The **Y-Joint** tool on the **2D Tool** palette is used to join multiple walls that come together at odd angles, and cannot be joined using the **Wall** or **Butt-Join Wall** tools.

Note: This tool is only available when the **AEC Overlay** is active. If the **Y-Join Wall** icon does not appear on the **2D Tool** palette, select **AEC Overlays** using the **Overlays** command on the **File** menu.

To use the Y-Join tool:

1. Use the **Wall** tool's **Join Wall** button to join two of the walls.
2. Draw the next wall segment, stopping just short of the junction of the first two walls.

   Note: All walls should have a fill.
3. Select the first two walls.
4. Select the **Y-Join** tool on the **2D Tool** palette.
5. Click on the new wall segment.

   It joins the first two walls at their junction.

The **Align Plane** tool on the **3D Tool** palette allows 3D objects to be aligned to the working plane in both rotation and translation. Since the working plane can be set easily to a plane of an existing object, the **Align Plane** tool allows any two objects to be aligned precisely to each other.

After alignment, the working plane remains at the plane between these two objects. This allows the objects to be moved or rotated relative to each other while maintaining this precise alignment.

To use the Align Plane tool:

1. Set the working plane to the plane to which you want to align one or more objects.
2. Select the objects.
3. Select the **Align Plane** tool.

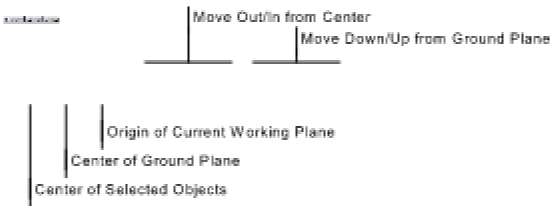
If the drawing is in wireframe, three snap points must be clicked to set the plane which aligns to the working plane. In solid mode, simply click the face of the object.

Note that the **Align Plane** tool is used exactly like the **Set Working Plane** tool.

While the **Align Plane** tool works with precision in solid mode, even more control is available in wireframe. This is because the three points also completely define the axes that align to the working plane axes. When a plane is selected in solid mode, the axes are derived from the points of the polygon that is clicked.

The **Flyover** tool on the **3D Tool** palette is used to move the view of the drawing by dragging the mouse. It changes the angle of view, while constraining the viewer to an upright position. It is the easiest way to manipulate your view of 3D space.

When the **Flyover** tool is selected, the flyover icons are displayed in the mode bar. These icons are used to select an origin, and to perform rotation in discrete steps.



tip! The **U** key can be used to toggle among the first group of icons. The four motion buttons can be selected from the keyboard using the **I**, **O**, **P**, and **[** keys, respectively.

In **Flyover** mode, the following mouse movements rotate the drawing.

- **Vertical mouse movements** change the viewer's latitude with respect to an imaginary sphere about the currently-selected center of rotation.
- **Horizontal mouse movements** change the viewer's longitude with respect to an imaginary sphere about the currently-selected center of rotation.

If the **Control** key is pressed while the mouse is moved, the viewpoint can be modified in ways that are useful in a perspective projection:

- **Control Key + Vertical mouse movements** changes the viewer's height from the ground plane. This movement is only useful in perspective projection.

  You can also use the **Up** and **Down** buttons on the mode bar instead of the mouse. These buttons let you move up and down in step-like fashion. Each click of the button results in a single step in the requested direction.

- **Control Key + Horizontal mouse movements** changes the distance between the viewer and the currently-selected center of rotation. Moving the mouse right increases the distance from center. Moving it left decreases the distance from center. This movement is only useful in perspective projection.

  You can also use the **In** and **Out** buttons on the mode bar instead of the mouse. These buttons let you move in and out in step-like fashion. Each click of the button results in a single step in the requested direction.

If a model is rendered solid, it will turn to wireframe during rotation. It will be re-rendered solid when the mouse is released.

To achieve the best possible performance, MiniCad optimizes the drawing by reducing the level of detail it displays. This is done to retain the same level of control in a complex model as in a very simple one.

You will find that a slow computer will have less range of control than a faster computer. You can use MiniCad's **3D Rotation** preference to optimize the performance of this tool on your system. This option is specified using the **Preferences** command is on the **File** menu.

For related information, see the following topics:

Rotate 3D View

Rotate View

Walkthrough

The **Locus 3D** tool on the **3D Tool** palette is used to place a 3D reference point (represented by a triad) on the drawing.

A 3D Locus has a location (*x*. *y* and *z* coordinates), but no dimension. It can be moved, but not reshaped. A locus can be placed more reliably if the **Snap to Objects** constraint is on. If the 3D Locus isn't snapped to an object, it is placed on the working plane.

If you want to move a **3D Locus**, use the **Object Info Palette**. It has the controls to position the locus in relation to the working plane or the ground plane. The **Move 3D** command can also be used to place the locus.

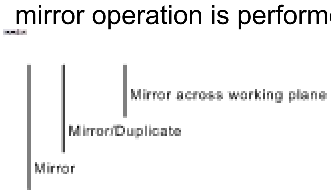
A **3D Locus**, like any other object, can be cut or copied and pasted through the Clipboard

For related information, see the following topics:

2D Locus

The **3D Mirror** tool on the **3D Tool** palette is used to make a mirror image of a selected object in 3D space.

When the **3D Mirror** tool is selected, the mirror icons appear in the mode bar. They determine which mirror operation is performed.



For additional information, see the following topics:

Mirror 3D mode
Mirror/Duplicate 3D mode
Mirror Across Working Plane mode

For related information, see the following topics:

Rotate 3D
Rotate
2D Mirror

The **Mirror 3D** mode flips the selected object on a specified axis. The axis is used to designate the offset, if any, of the mirrored object.

To mirror a 3D object

1. Select the object you want to flip.
2. Select the **3D Mirror** tool from the **2D Tool** palette.
3. Select the **Mirror** icon in the mode bar.
4. Drag a line to designate the axis around which the object will pivot.
5. **Note:** By drawing this line outside the object, you can offset the mirrored result from the object's current position.

---

6. Release the mouse button.

   The object is mirrored.

The **Mirror/Duplicate 3D** mode creates a mirror-image copy of the selected object. An axis line is drawn to designate the pivot point around which the object will rotate.

To mirror/duplicate an object:

1. Select the object you want to duplicate and mirror.
2. Select the **3D Mirror** tool from the **3D Tool** palette.
3. Select the **Mirror/Duplicate** icon in the mode bar.
4. Drag a line to designate the axis around which the copied object will pivot.

   Note: By drawing this line outside the object, you can offset the copy from the original.

5. Release the mouse button.

   The mirrored copy is produced.

The **Mirror Across Working Plane** mode is used to automatically mirror an object and reposition it on the opposite side of the working plane.

To mirror across the working plane:

1. Select the 3D object you want to mirror.
2. Select the **3D Mirror** tool from the **3D Tool** palette.
3. Select the **Mirror Across Working Plane** icon in the mode bar.

The **Extruded Polygon** tool on the **3D Tool** palette is used to extrude objects in 3D without drawing them in 2D first. This command actually creates a mesh object, not an extrude.

Note: This tool lies beneath the **3D Polygon** tool on the **3D Tool** palette. To access the **Extruded Polygon,** press and hold the **3D Polygon** icon. When the pop-out icons appear, release the mouse over the **Extruded Polygon** tool.

To create an Extruded Polygon:

1. Place the drawing in any 3D projection.
2. Select the **Extruded Polygon** tool from the **3D Tool** palette.

   The cursor for the tool will project on the ground plane or a specified working plane.
3. Position the cursor at one of the polygon's vertices.
4. Drag the mouse up to set the height of the polygon.

   The Δz field in the **Data Display** bar reports the height of the figure as the mouse is dragged.
5. When the cursor is at the correct height (i.e., *z* position), release the mouse button.

   Note: If the height vertex is zero, or if the object is created in top view, a dialog box will prompt you for the object's height. If, for the sake of accuracy, you prefer to enter the polygon's height via a dialog, click once at the start point to bring up the dialog box.
6. Move the cursor to the next vertex.
7. Click the mouse.
8. Repeat steps 6 and 7 for each vertex. Double-click the last vertex to end the operation. (If the last point is also the polygon's start point, a single click will end the operation).

   If a point snaps outside the working plane, it projects into the working plane.

For additional information, see the following topics:

Editing Extruded Polygons

Extruded polygons are edited using the **Object Info Palette**. Because the object is a mesh, individual vertices of the object can be edited.

To edit the figure, select it, then enter new values into the appropriate fields of the **Object Info** palette. (If the **Object Info** palette is not currently displayed, open it with the **Object Info** command.) The palette's **Edit** option lets you choose between editing a single vertex or multiple vertices.

For related information, see the following topics:

Extrude
Multiple Extrude
2D Polygon
3D Polygon
Extruded Rectangle

The **3D Polygon** tool on the **3D Tool** palette is used to create 3D polygons without drawing them in 2D first.

A 3D polygon does not have a depth, it does have a *z* value, which gives it a location in 3D space. A 3D polygon can be rotated, placed in perspective, and translated in space like any other 3D object. Unlike a 2D polygon, 3D polygons must be rendered to apply a fill pattern to them.

To draw a 3D polygon:

1. Select the **3D Polygon** tool from the **3D Tool** palette.
2. Place the cursor at the polygon's starting point.
3. Click the mouse.
4. Move the cursor to the next vertex (hold the **Shift** key if you want to constrain the line).
5. Click the mouse.
6. Repeat the last two steps for each vertex. Double-click the last vertex to end the operation. If the last point is also the polygon's starting point, a single click will end the operation.

   Note: The perpendicular constraint allows points to be specified outside the working plane. The constraint vector is determined by the location of the mouse when the constraint is activated. It can be repositioned by deactivating the constraint, repositioning the mouse, and activating the constraint again. Use the keyboard to control the constraint in order to control the location of the constraint vector.

For related information, see the following topics:

Extrude
Multiple Extrude
2D Polygon
Extruded Polygon
Extruded Rectangle

The **Extruded Rectangle** on the **3D Tool** palette creates a 3D rectangle in one operation (i.e., without having to first draw it in 2D and then extrude it).

A mesh object is created when this tool is used. The rectangle is created at the working plane axes.

To create an Extruded Rectangle:

1. Place the drawing in any 3D projection.
2. Select the **Extruded Rectangle** tool from the **3D Tool** palette.

   The cursor for the tool will project on the ground plane or on a specified working plane.
3. Position the cursor at one of the rectangle's corner points.
4. Drag the mouse up to set the height of the rectangle.

   The **Δ*z*** field in the **Data Display** bar reports the height of the rectangle as the mouse is dragged.
5. When the cursor is at the rectangle's height (i.e., *z* position), release the mouse button.
6. **Note:** If the height vertex is zero, or if the object is created in top view, a dialog box will prompt you for the rectangle's height. If, for the sake of accuracy, you prefer to enter the object's height via a dialog, click once at the start point to bring up the dialog box.
7. Move the mouse to form the extruded rectangle.
8. Click to finish the operation.

   If a point is snapped outside the working plane, it is projected into the working plane so that the extruded rectangle is planer. Click once on the start point or double-click anywhere to end the operation.

For related information, see the following topics:

Editing Extruded Rectangles
Extrude
Multiple Extrude
Extruded Polygon

Extruded rectangles are edited using the **Object Info Palette**. Because the object is a mesh, individual vertices of the object can be edited.

To edit the figure, select it, and enter new values into the appropriate fields of the **Object Info** palette. (If the **Object Info** palette is not currently displayed, open it with the **Object Info** command.) The palette's **Edit** option lets you choose between editing a single vertex or multiple vertices.

The **3D Reshape** tool on the **3D Tool** palette is used to edit extrudes, walls, roofs and 3D polygons.

The **3D Reshape** tool can be used to reshape a single 3D object. But, because of conflicting constraints imposed on different 3D objects, this tool cannot be used on a group of objects.

In reshape mode, an object shows all reshape handles and, if necessary, the working plane changes. If points can be added or deleted from an object, then the mode bar will shows those options.

3D operations that allow movement perpendicular to the working plane do not work if the working plane is being viewed straight on. This means that operations that allow movement only in the working plane will work if the working plane is viewed on its side. For example, it is impossible to edit the height of an object if viewed from above. Because of this, you may want to use an isometric view of the working plane when using the **3D Reshape** tool.

Note: The **3D Reshape** tool cannot be used on 3D polygons or extruded rectangles. Because these figures are mesh objects, not true extruded objects, they must be reshaped as mesh objects.

For additional information, see the following topics:

Reshaping Extrudes
Reshaping Walls
Reshaping Roofs
Reshaping 3D Polygons

The **3D Reshape** tool can be used to change the height of an extruded object. At the center of each end face is a control handle, which can be dragged along the vector perpendicular to the face of the extrude.

If a 3D point is snapped during this operation, that point is projected onto the constraint vector, aligning the extruded face to that point.

In order to edit an extrude, the **Edit Group** command must be used.

The **3D Reshape** tool can be used to change the height of the top and bottom surface, change the height and location of any point peak and change the location of the end points. All reshape operations affect the wall's front and back faces.

Because a wall must remain vertical with respect to the ground plane, reshape operations on a wall constrain its freedom of motion. Moving the top or bottom surface at its end point constrains movement perpendicular to the ground plane and prevents creation of a wall of a negative height.

The "+" and "-" icons on the mode bar can be used to add or delete peak points.

- To add a peak point, click on an existing peak or end point and drag a new peak point from it.
- To delete a peak point, click on the point to remove it.

The **3D Reshape** tool can be used to reshape the rise/run (angle) of a roof.

Control handles are placed on the defining plane of the roof (on which the roof axis lies) at the corners of the area enclosing the entire roof. A roof reshape is constrained to 0° - 85° inclination. If another 3D point is snapped during reshape, the roof is aligned such that the point specified lies in the roof plane.

Note that the roof plane is the top surface of a horizontally-mitered roof, the bottom surface of a vertically-mitered roof, and the surface intersecting the lowest corner of a double-mitered roof.

The **3D Reshape** tool can be used to reshape a 3D polygon. If the 3D polygon is a planar polygon, the working plane is set to that plane. The polygon is edited in the working plane in the same way as a 2D polygon is edited.

If a point is snapped to a point outside of the plane, the polygon becomes non-planar. If the perpendicular constraint is active, a point is constrained to the vector that passes through it perpendicular to the working plane.

If the polygon is non-planar when it is selected, the current working plane remains set and editing operations take place in that plane.

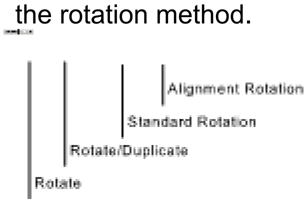
For related information, see the following topics:

2D Reshape

The **Rotate 3D** tool on the **3D Tool** palette is used to rotate a selected object in 3D space. An object can be optionally copied when it is rotated.

To use the **Rotate 3D** tool, the drawing must be in 3D view and a 3D object must be selected.

When the **Rotate 3D** tool is selected, the 3D rotation icons appear on the mode bar. The first set of icons determine whether the selected object or a duplicate of it is rotated. The second set of icons specifies the rotation method.

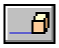


tip! The **U** key can be used to toggle among the first group of icons. The **I** key can be used to toggle among the second group of icons.

For additional information, see the following topics:

Standard Rotation

Alignment Rotation

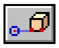


The **Standard Rotation** mode rotates selected objects about the working plane normal.

To perform a standard rotation of a 3D object:

1. Select the 3D object you want to rotate.
2. Switch to an orthogonal projection.
3. Select the **Rotate 3D**tool from the **3D Tool** palette.
4. If you want to rotate a copy of the selected object, select the **Rotate Duplicate Object** icon in the mode bar. Otherwise, select the **Rotate Selected Object** icon.
5. Select the **Standard Rotation** icon on the mode bar.
6. Drag a rotation axis and release the mouse.
7. Move the mouse to rotate the object.

   The object rotates with the "lever" you created with the axis line. If the **Snap To Points** constraint is active, the axis allows precise rotation.
8. Click to end the rotation.



The **Alignment Rotation** mode rotates the selected object so that the rotation lever aligns with an alignment vector.

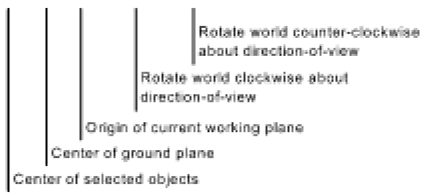
To perform an alignment rotation of a 3D object:

1. Switch to an orthogonal projection.
2. Select the **Rotate 3D**tool from the **3D Tool** palette.
3. If you want to rotate a copy of the selected object, select the **Rotate Duplicate Object** icon in the mode bar. Otherwise, select the **Rotate Selected Object** icon.
4. Select the **Alignment Rotation** icon on the mode bar.
5. Select the object and snap to a center of rotation.
6. Drag to a second point defining the rotation lever and snap to this point.
7. Release the mouse and snap to a third point. This aligns the rotation lever with the alignment lever defined by the first and third points.
8. Click again to end the rotation.

The **Rotate View** tool on the on the **3D Tool** palette is used to change a model's angle of view. It allows full control over the three rotation axes, allowing any view orientation.

Note: This tool lies beneath the **Flyover** tool on the **3D Tool** palette. To access **Rotate View,** press and hold the **Flyover** icon. When the pop-out icons appear, release the mouse over the **Rotate View** tool.

When the **Rotate View** tool is selected, the rotate view icons appear on the mode bar. These icons are used to set the center of rotation and the direction of rotation.



tip! The **U** key can be used to toggle among the first group of icons. The **I** key can be used to select the **Rotate Clockwise** icon; the **O** key can be used to select the **Rotate Counter-clockwise** icon.

In **Rotate View** mode, the active area of the screen is divided into nine sectors, made up of three vertical and three horizontal stripes. Each stripe responds to mouse motion, such that the entire screen becomes a virtual trackball.

1	2	3
4	5	6
7	8	9

Vertical Mouse Movement

Mouse movement in sectors 1-4-7 and 3-6-9 rotates the world about the direction of view, "tilting" the viewer orientation.

Mouse movement in sector 2-5-8 rotates the world about the axis parallel to the *x* axis of the screen, passing through the center of rotation selection in the mode bar.

Horizontal Mouse Movement

Mouse movement in sectors 1-2-3 and 7-8-9 rotates the world about the direction of view, "tilting" the viewer orientation.

Mouse movement in sector 4-5-6 rotates the world about the axis parallel to the *y* axis of the screen and passing through the center of rotation selected in the mode bar.

Note: The **Rotate View** tool provides so much movement that it can become awkward to use. If you want to constrain the view attitude, use the **Flyover** tool.

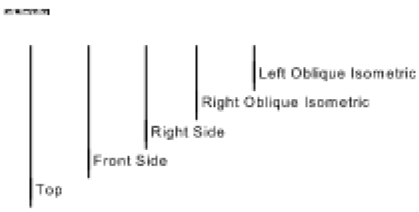
For related information, see the following topics:

Rotate 3D View
Translate View
Walkthrough
Flyover

Rotate

The **3D Selection** tool on the **3D Tool** palette is used to select objects in 3D orthogonal views. This tool works with the 3D SmartCursor to allow 3D snaps and constraints.

When the **3D Selection** tool is selected, the standard views icons appear in the mode bar. These views are identical to the views available with the **Standard Views** command.



The **Set Working Plane** tool on the **3D Tool** palette is used to set the working plane based on either a currently extant object's plane or three points in space specified by the user.

The "working plane" is a movable, user-defined plane that allows precise 3D editing. When 3D tools are active, the mouse's position in 3D space is determined by the working plane. It is a constant reference that is used with all 3D tools.

The working plane can change temporarily to facilitate editing operations when the 3D SmartCursor snaps to points that are not in the current working plane.

The working plane is set by clicking on any face of a rendered object or by snapping to any three points in wireframe mode with the **Set Working Plane** tool.

To select a working plane by selecting three points:

1. Select the **Set Working Plane** tool from the **3D Tool** palette.
2. Snap to first the point, press the mouse button and hold down (this becomes the working plane origin).
3. Move the mouse to next point and release. (This vector is the *x* axis of the working plane.)
4. Snap to the last point and click.

   The working plane is set to the plane that intersects the three points you've specified. Remember that the working plane can be set only by snapping to points on an object. If the mouse is clicked when the selection pointer is not located over a snap point, the working plane is set to its default state and aligns with the ground plane.

For related information, see the following topics:

Move Working Plane
Working Plane Palette

The **Insert 3D Symbol** tool on the **3D Tool** palette is used to insert symbols in 3D orthogonal projections.

Symbols are selected from the **Resource Palette**. You can open the **Resource Palette** by selecting the **Resources** command on the **Window** menu, or by double-clicking the **Insert 3D Symbol** tool.

Note: The selected symbol becomes the default symbol,. It will remain so until a new symbol is selected.

To place a symbol in a 3D orthogonal view:

1. Select the **Insert 3D Symbol** tool.
2. Position the mouse pointer at the symbol's insertion point.
3. Click the mouse.

   The selected symbol is inserted.

The symbol appears with its origin at the insertion point and is aligned such that the coordinate axes of the symbol align with the current working-plane axes. If the mouse is dragged after inserting the symbol, it is rotated about the vertex through its origin, which is normal to the current working plane.

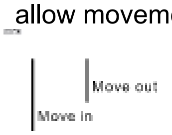
3D symbols that do not have a 2D component, are visible in the 2D/Plan view. The symbols appear the same as in 3D Top view.

The **Translate View** tool on the **3D Tool** palette moves the 3D projection around the page. (Translate is a mathematical term for changing a coordinate system's origin, or zero point. In effect, this moves objects through 3D space without rotating them or changing their shape.)

Note: This tool lies beneath the **Walkthrough** tool on the **3D Tool** palette. To access **Translate View**, press and hold the **Walkthrough** icon. When the pop-out icons appear, release the mouse over the **Translate View** tool.

The **Translate View** tool can move the viewpoint up, down, left, right, in or out. It uses the current-view orientation as a reference for translation axes. This means the viewpoint changes relative to the viewer's current position in space.

When the **Translate View** tool is selected, the translate icons appear in the mode bar. These buttons allow movement in steps along the axis perpendicular to the screen.



The **Translate View** tool also allows perspective to be altered if it is used with the **Shift** key pressed. Drag the mouse to the right to decrease perspective. Drag to the left to increase it.

In **Translate View** mode, the mouse is used to move the view point.

- **Vertical mouse movements** move the viewpoint up or down with respect to the current direction of view. This, in effect, moves the 3D projection down and up on the page.
- **Horizontal mouse movements** move the viewpoint left or right with respect to the current direction of view. This, in effect, moves the 3D projection right and left on the page.

If the **Control** key is pressed while the mouse is moved, the viewpoint can be modified in ways that are useful in a perspective projection:

- **Control Key + Vertical mouse movements** move the viewpoint in and out along the direction of view. This movement is only useful in perspective projection.

  You can also use the **In** and **Out** buttons on the mode bar instead of the mouse. These buttons let you move in and out in step-like fashion. Each click of the button results in a single step in the requested direction.
- **Control Key + Horizontal mouse movements** change the amount of perspective, or the "focal length" of the perspective. Moving the mouse right increases the focal length (telephoto). Moving it left decreases the focal length (fisheye). This movement is only useful in perspective projection.

For related information, see the following topics:

Rotate 3D View
Rotate View
Walkthrough
Flyover
Rotate

The **Walkthrough** tool on the **3D Tool** palette is used to visualize an architectural model. Movement and rotation is centered around the viewer, giving one the sense of walking or driving in the 3D world.

The **Walkthrough** tool is used in the perspective projection. If this projection is not active when the tool is selected, you will be prompted to switch to this mode.

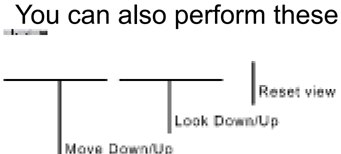
In **Walkthrough** mode, movements forward, backward, left and right are performed using the mouse. (Movement is always constrained to the plane parallel to the ground plane.)

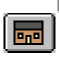
- When the mouse is in the center of the screen, no movement occurs.
- When the mouse is above the horizontal center of the screen, you move forward.
- When the mouse is below the horizontal center of the screen, you move backward.
- When the mouse is on the left of the vertical center of the screen, you turn left.
- When the mouse in on the right of the vertical center of the screen, you turn right.

The **Shift** key can be used in conjunction with the mouse to move up and down, or look up and down.

- When the **Shift** key is pressed while the mouse is above the horizontal center of the screen, you move up (perpendicular to the ground plane).
- When the **Shift** key is pressed while the mouse is below the horizontal center of the screen, you move down (perpendicular to the ground plane).
- When the **Shift** key is pressed while the mouse is on the left of the vertical center of the screen, you look up.
- When the **Shift** key is pressed while the mouse is on the right of the vertical center of the screen, you look down.

You can also perform these actions in incremental movements using the buttons on the mode bar.





The **Reset** button on the mode bar resets the orientation so that you are level and facing the front of the model. This is useful when a walkthrough begins. You can also set the starting point for a walkthrough with the **Set 3D View** command.

For related information, see the following topics:

Flyover
Rotate 3D View
Translate View
Rotate View
Rotate

The **Zoom In** tool on the **2D Tool** and **3D Tool** palettes magnifies the view of the drawing. Each double-click of the **Zoom In** tool doubles the magnification. (Selected objects are centered in the magnified view.)

The magnification level is displayed in the mode bar. A value of 100% represents the drawing's normal scale (the scale at which the drawing is printed), a value of 200% represents twice the normal scale, and so forth.

Note: The **Zoom In** tool does not affect the physical size (scale) of the drawing. It merely magnifies your view of it. To change the drawing's scale, you must use the **Scale** command.

The **Zoom In** tool can also be used to magnify a specific section of a drawing.

To magnify a section of the drawing:

1. Select the **Zoom Tool**.
2. Drag a marquee around the area to be examined.
3. Release the mouse.

   The area within the marquee is magnified to fill the screen

tip! There are many built-in shortcut keys to zoom the display to different levels. See the **Zoom Keyboard commands** for a complete list.

For related information, see the following topics:

Fit to Window
Fit to Objects
Normal Scale

The **Zoom Out** tool on the on the **2D Tool** and **3D Tool** palettes reduces the view of the drawing. Each double-click on the **Zoom Out** tool reduces the view by half.

The magnification level is displayed in the mode bar. A value of 100% represents the drawing's normal scale (the scale at which the drawing is printed), a value of 50% represents half the normal scale, and so forth.

There are many built-in shortcut keys to zoom the display to different levels. See the **Zoom Keyboard commands** for a complete list.

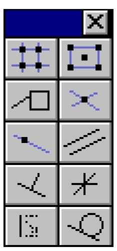
Note: The **Zoom Out** tool does not affect the physical size (scale) of the drawing. It merely reduces your view of it. To change the drawing's scale, you must use the **Scale** command.

For related information, see the following topics:

Fit to Window
Fit to Objects
Normal Scale

Additional zoom operations are available through the keyboard and the command menu.

Key Combination	Description
Cntrl-1	Enlarges the drawing by a factor of 2
Cntrl-Shift	Enlarges the drawing by a factor of 4
Cntrl-2	Reduces the drawing by half
Cntrl-Shift-2	Reduces the size of the drawing to 25% of its normal size
Cntrl-4	Fits the page into the screen (same as the **Fit to Window** command)
Cntrl-3	Displays the drawing at normal scale (same as the **Normal Scale** command)

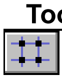
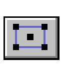
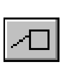


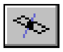
The **Constraint** palette is used to position the cursor at a specific point when drawing, moving or reshaping objects (i.e., *snapping* an object to a position). It is also used to limit a tool to a specific angle, shape or position (e.g., *constraining* the **Rectangle** tool to produce only squares).

The **Constraint** palette is opened using the **Constraints** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette.

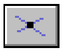
The **Constraint** palette automatically shows 2D constraints when a 2D drawing tool is selected and 3D constraints when a 3D tool is selected.

Icons on the palette can be selected by clicking them with the mouse or by typing their key equivalent. A constraint can be selected before you draw an object or while you are drawing the object.

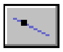
Tool	Description
	The **Snap to Grid (2D)** tool causes the cursor to align on the snap grid. The **Snap to Grid (3D)** tool constrains objects on the working plane.
	The **Snap to Object (2D)** tool causes objects to align on an object's "snap points." The **Snap to object (3D)** tool makes the snap points at 3D vertices active.
	The **Snap To Surface** tool causes the cursor to snap to the surface (side) of an object.



The **Constrain to the Working Plane** tool projects any point outside the working plane to the working plane.

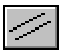


The **Snap to Intersection** tool causes the cursor to snap to the intersection of objects.



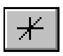
The **Snap to Distance (2D)** tool snaps the cursor at a specified distance from a vector.

The **Snap to Distance (3D)** tool lets you to snap to a specified distance from either end of any 3D vector.



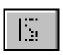
The **Constrain Parallel** tool lets you produce parallel or concentric objects.

The **Constrain Perpendicular** tool allows a vector to be drawn perpendicular to or from an object.



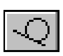
The **Constrain Angle (2D)** tool constrains vectors to 30°, 45° and 90°.

The **Constrain Angle (3D)** tool constrains angles to 30°, 45° and 90° in current working plane space

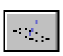


The **Constrain Symmetrical (2D)** tool restricts objects to a height:width ratio of one (1:1)

The **Constrain Symmetrical (3D)** tool is used to constrain all 3D drawing to the working plane.



The **Constrain Tangent** tool constrains lines to be tangent to or from arcs, ovals, and circles.



The **Constrain Perpendicular (3D)** tool is used to snap objects perpendicular to the working plane.

This constraint causes the cursor to align on the snap grid when objects are drawn, moved or reshaped. The snap grid is defined using the **Set Grid** command.

This constraint causes objects to align on an object's "snap points." Every object has snap points. On a line they are its end points. In a rectangle, they are its center and its corners. In a polygon, they are the vertices.

Note: Grouped objects retain their constituent object's snap points.

If the **Preferences** command has been used to turn on screen hints, a label will appear at the snap points, and a dotted line will connect the point to your cursor arrow.

This constraint causes the cursor to snap to the surface (side) of an object.

When this constraint is selected, a "snap dot" will appear next to the cursor when it is within three pixels of an object's surface. This dot signals that clicking or releasing the mouse will snap the object to that point.

You can designate a specific object to be the only one affected by this constraint, by double-clicking the **Snap to Surface** icon, or by typing the **S** key twice in rapid succession. Then, when the **Finger Tip** cursor is on the desired object, click the mouse. The selected object will be the only object subject to the **Snap to Surface** constraint.

To deactivate the constraint on this object, deselect the **Snap to Surface** icon.

This constraint causes the cursor to snap to the intersection of objects.

If the **Preferences** command has been used to turn on screen hints, a circle labeled "Intersect" will appear at the intersection, and a dotted line will connect the circle to your cursor.

This constraint allows you to snap the cursor at a specified distance from either end of a vector in the drawing.

To set the snapping distance, select the object and double-click the **Snap to Distance** icon. In the **Snap Distance** dialog box, set the distance in terms of a fraction, percent or distance.

When you move the cursor along a vector, it snaps it to the nearest **Snap To Distance** point. If the **Preferences** command has been used to turn on screen hints, the description "Along Line" appears when the cursor is in snap position.

This constraint restricts a drawing tool to producing parallel or concentric objects. If the constrain parallel icon is dim, it means that a parallel object cannot be created with the selected drawing tool.

To use this constraint:

1. Select the **Constrain Parallel** icon.
2. Click and draw toward the line the new line is to be parallel with. When the cursor is within three pixels, the new line positions itself parallel to the existing line.
3. Click on the line to be parallel to the new line. While drawing the new parallel object, the cursor can be moved to either side of the line.

You can designate a specific object to be the only one affected by this constraint, by double-clicking the **Constrain Parallel** icon, or by typing the **E** key twice in rapid succession. Then, when the **Finger Tip** cursor is on the desired object, click the mouse. The selected object will be the only object subject to the parallel constraint.

When the line tool is selected, the **Tab** key can be used to enter the **Offset** field in the **Data Display** bar.

When objects are created parallel to lines or other objects, they retain the height-to-width ratio of the objects they are drawn parallel to. This allows concentric objects can be drawn with the parallel constraints.

The **Constrain Perpendicular** tool allows a vector to be drawn perpendicular to or from an object. This constraint can be used with the following drawing tools:

Tool	Constrain Perpendicular Effect
Single Line or **Double Line**	Can be used to constrain the line to any Line, Rectangle, Wall or Polygon
Full Arc or **Quarter Arc**	Can be used to constrain the radius to any Line, Rectangle, Wall or Polygon
Polyline	Can be used to constrain a polyline segment to any Line, Rectangle, Wall or Polygon
Wall	Can be used to constrain the wall to any Line, Rectangle, Wall or Polygon
Polygon	Can be used to constrain a polygon segment to any Line, Rectangle, Wall or Polygon

You can designate a specific object to be the only one affected by this constraint, by double-clicking the **Constrain Perpendicular** icon or typing the **F** key twice in rapid succession. Then, when the **Finger Tip** cursor is on the desired object, click the mouse. The selected object will be the only object subject to the perpendicular constraint.

The **Constrain Angle** tool will constrain vectors to 30°, 45°, 90° and their complements.

An additional constraint angle can be set with the **Preferences** command on the **File** menu.

The **Constrain Symmetrical** constraint restricts objects to a height:width ratio of one (1:1).

This constraint can be used with the following drawing tools:

Tool	Constrain Symmetrical Effect
Rectangle tool	Creates squares
Rounded Rectangle tool	Creates rounded squares
Ovals tool	Creates circles
Quarter Arc tool	Creates quarter circles

The **Constrain Tangent** tool constrains lines to be tangent to or from arcs, ovals, and circles. It also allows a tangent-to-tangent constraint between two circles.

- When drawing to an arc, oval, or circle, the line snaps tangent to it.
- When drawing from an arc, oval, or circle, the line floats around the surface until the end point of the line is set. Pressing the **Control** key flips the tangent line to the opposite side of the oval or circle.

You can designate a specific object to be the only one affected by this constraint, by double-clicking the **Constrain Tangent** icon, or by typing the **T** key twice in rapid succession. When the **Finger Tip** cursor is on the desired object, click the mouse. The selected object will be the only object subject to the tangent constraint.

The **Snap to Grid** constraint on the 3D **Constraint** palette constrains objects on the working plane. When the working plane also is the ground plane, the **Snap to Grid** constraint works as it does in 2D. However, if the working plane is set to some other plane, then the snapping is based on the working plane origin and orientation.

The snap grid is defined using the **Set Grid** command.

The **Snap to Object** constraint on the 3D **Constraint** palette makes all snap points at vertices of 3D objects active in 3D space for both snapping and alignment with the 3D SmartCursor.

It is possible to snap to objects between layers when three conditions are met:

- The **Layer Options** command is set to **Show Snap Others** or **Show/Snap/Modify Others**.
- The layers have been set to normal (visible) with the **Layers** command.
- The layers are in the same scale, projection, and view.

The 3D cursor location is normally at the intersection of the mouse vector and the working plane. All points specified in 3D are on the working plane unless the cursor is over a 3D snap point, and the **Snap to Object** constraint is selected.

If screen hints are on, the type of snap point is shown and visual clues are given as to where the point is in relation to the current working plane. A dashed line is displayed perpendicular to the working plane which passes through the snap point and intersects the crosshairs on the working plane.

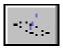
This constraint projects any point outside the working plane to that point on the working plane directly beneath the original.

When using this constraint, if a point is snapped outside the current working plane, the point specified is that point on the working plane that is the shadow of the original.

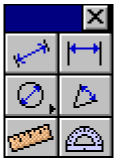
The **Constrain Symmetrical** tool on the 3D **Constraint** palette is used to constrain all 3D drawing to the working plane.

The **Snap to Distance** tool in the 3D **Constraint** palette allows you to snap to a specified distance from either end of any 3D vector in the drawing. To set the snapping distance, double-click the **Snap to Distance** tool and specify the distance in the **Snap Distance** dialog box.

The **Constrain Angle** tool on the 3D **Constraint** palette works in current working plane space. Angles are constrained to 90°, 60°, 45°, or 30°. An additional constraint can be specified with the **Preferences** command.

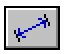
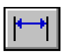
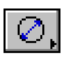
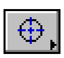
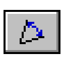


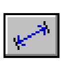
The **Constrain Perpendicular** tool on the 3D **Constraint** palette is used to snap objects perpendicular to the working plane.



The tools on the **Standard Dimension** palette are used to measure linear and angular dimensions. Some tools produce measurement lines and place them in the drawing. Others (e.g., the **Tape Measure** and the **Protractor**) simply report a dimension, but do not create an object.

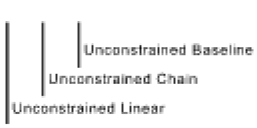
The **Standard Dimensions** palette is opened using the **Tools** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette. It contains the following tools:

Tool	Description
	The **Diagonal Dimension** tool is used to measure and mark linear dimensions of any angle.
	The **Constrained Linear Dimension** tool is used to measure and mark vertical and horizontal linear dimensions that are constrained to the horizontal and vertical axes.
	The **Circular Dimension** tool is used to measure and mark the diameter or radius of an arc or circle.
	The **Center Mark** tool is used to place a center mark in a circle, ellipse, rectangle or rounded rectangle.
	The **Angular Dimension** tool is used to measure and mark an angle.
	The **Tape Measure** tool is used to measure the distance between points on the drawing.
	The **Protractor** tool is used to measure the angle between points on the drawing.



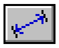
The **Diagonal Dimension** tool on the **Standard Dimension** palette is used to measure and mark linear dimensions of any angle.

When the **Diagonal Dimension** tool is selected, the diagonal icons are displayed in the mode bar. These icons are used to select the type of dimensioning operation you want to perform.

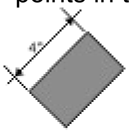


For additional information, see the following topics:

Unconstrained Linear Dimension
Unconstrained Chain Dimension
Unconstrained Baseline Dimension



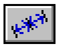
The **Unconstrained Linear Dimension** mode measures and marks the distance between any two points in the drawing.



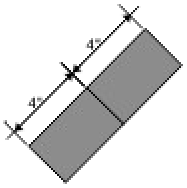
To create an unconstrained linear dimension:

1. Select the **Diagonal Dimension** tool from the **Standard Dimension** palette.
2. Select the **Unconstrained Linear** icon in the mode bar.
3. Position the cursor at the beginning of the line.
4. Press and hold the mouse button.
5. Drag the cursor to the end point of the line.
6. Release the mouse.

   The dimension outline appears.
7. Move the cursor to where you want the dimension line placed.
8. Click the mouse to set the line in the drawing.



The **Unconstrained Chain Dimension** mode measures and marks the distance between multiple points on the same line.



To create an unconstrained chain dimension:

1. Select the **Diagonal Dimension** tool from the **Standard Dimension** palette.
2. Select the **Unconstrained Chain** icon in the mode bar.
3. Position the cursor at the first point on the line.
4. Press and hold the mouse button.
5. Drag the cursor to the end of the first segment in the chain.

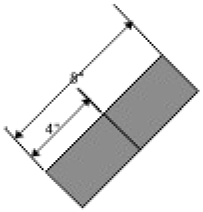
   Note: the position of this segment establishes the angle for the entire chain.

6. Release the mouse button.
7. Move the cursor to where you want the dimension line drawn.
8. Click the mouse.

   The cursor changes to a bull's-eye.

9. Move the cursor to the endpoint of the next segment.
10. Click the mouse.
11. Repeat steps 9 and 10 for each segment in the chain. When you reach the last point, double-click the mouse or press **Esc** to end the operation.

The **Unconstrained Baseline** mode measures the distance between a point (or set of points) and a fixed baseline drawn at any angle.

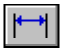


To create an Unconstrained Baseline dimension:

1. Select the **Diagonal Dimension** tool from the **Standard Dimension** palette.
2. Select the **Unconstrained Baseline** icon in the mode bar.
3. Position the cursor at the start point of the first length you want to measure.
4. Press and hold the mouse button.
5. Drag the cursor to the end of the first length you want to measure.

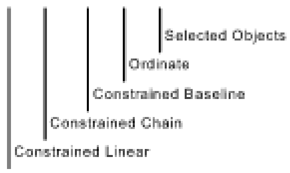
   Note: the position of this line establishes the angle for the rest of the measurement.

6. Release the mouse button.
7. Move the cursor to the offset position at which you want this first dimension line to appear.
8. Click the mouse. The cursor changes to a bull's-eye.
9. Move the cursor to the endpoint of the next length to be measured.
10. Click the mouse.
11. Repeat steps 9 and 10 for each length to be measured. When you reach the last point, double-click the mouse or press **Esc** to end the operation.



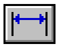
The **Constrained Linear Dimension** tool on the **Standard Dimension** palette is used to measure and mark vertical and horizontal linear dimensions. As its name implies, measurements are always taken on (i.e., constrained to) the horizontal or vertical axes.

When the **Constrained Linear Dimension** tool is selected, the linear icons are displayed in the mode bar. These icons are used to select the type of dimensioning operation you want to perform.

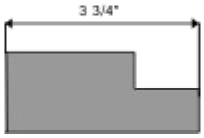


For additional information, see the following topics:

Constrained Linear Dimension
Constrained Chain Dimension
Constrained Baseline Dimension
Ordinate Dimension
Auto dimension selected objects mode



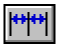
The **Constrained Linear** mode measures the distance between any two points on the drawing.



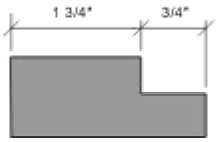
To create a Linear dimension:

1. Select the **Constrained Linear Dimension** tool from the **Standard Dimension** palette.
2. Select the **Constrained Linear** icon in the mode bar.
3. Position the cursor at the start point of the length you want to measure.
4. Press and hold the mouse button.
5. Drag the cursor to the end point of the dimension.
6. Release the mouse button.
7. Move the cursor to the offset position at which you want the dimension line to appear.
8. Click the mouse.

   The dimension line is drawn and labeled.

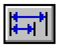


The **Constrained Chain** mode measures the distance between multiple points on a line.

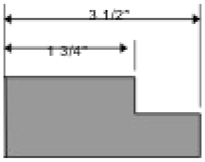


To create a Chained dimension:

1. Select the **Constrained Linear Dimension** tool from the **Standard Dimension** palette.
2. Select the **Constrained Chain** icon in the mode bar.
3. Position the cursor at the start point of the line.
4. Press and hold the mouse button.
5. Drag the cursor to the end of the first segment in the chain.
6. Release the mouse button.
7. Move the cursor to the offset position at which you want the dimension line to appear.
8. Click the mouse.
9. Move the cursor to the endpoint of the next segment.
10. Click the mouse.
11. Repeat steps 9 and 10 for each segment in the chain. When you reach the last point, double-click the mouse or press **Esc** to end the operation.



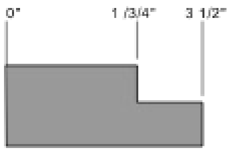
The **Constrained Baseline** mode measures the distance between a point (or set of points) and a fixed vertical or horizontal baseline.



To create a Baseline dimension:

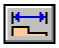
1. Select the **Constrained Linear Dimension** tool from the **Standard Dimension** palette.
2. Select the **Constrained Baseline** icon in the mode bar.
3. Position the cursor at the start point of the first length you want to measure. This point establishes the baseline from which all subsequent points are measured.
4. Press and hold the mouse button.
5. Drag the cursor to the end of the first length you want to measure.
6. Release the mouse button.
7. Move the cursor to the offset position at which you want the first dimension line to appear.
8. Click the mouse. The cursor changes to a bull's-eye.
9. Move the cursor to the endpoint of the next length to be measured.
10. Click the mouse.
11. Repeat steps 9 and 10 for each length to be measured. When you reach the last point, double-click the mouse or press **Esc** to end the operation.

The **Ordinate Dimension** mode is similar to **Constrained Baseline** mode in that it measures the distance between a point (or set of points) and a fixed baseline. However, **Ordinate Dimension** mode, rather than labeling each interval with an individual dimension line, labels the intervals along a fixed, offset line.

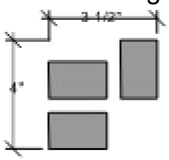


To create a Ordinate dimension:

1. Select the **Constrained Linear Dimension** tool from the **Standard Dimension** palette.
2. Select the **Ordinate Dimension** icon in the mode bar.
3. Position your cursor on the baseline, placing it where you want the measurement labels to appear. This point establishes the baseline from which all subsequent points are measured.
4. Move the cursor to the second point.
5. Click the mouse.
6. Repeat the two steps above for each point to be measured. When you reach the last point, double-click the mouse or press **Esc** to end the operation.



The **Auto Dimension** mode measures and marks the horizontal and vertical dimensions of a selected object (or objects). If multiple objects are selected, the total measurement from end to end of the multiple selection is given.



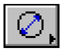
To Auto Dimension an object or group of objects:

1. Select the object, or group of objects, you want to measure.
2. Select the **Constrained Linear Dimension** tool from the **Standard Dimension** palette.
3. Select the **Auto Dimension** icon in the mode bar.
4. Press and hold the mouse button.
5. Move the cursor to where you want the dimension line drawn.
6. Release the mouse.

   The dimension line is placed in the drawing.

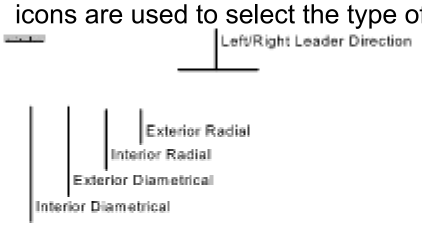
Note: When clicking inside the object to be dimensioned, the tool returns horizontal dimensions by default. Press the **Control** key to place a vertical dimension inside a selected object.

When clicking outside the object to be dimensioned, the type of dimension is determined by where the mouse is clicked.



The **Circular Dimension** tool on the **Standard Dimension** palette is used to measure and mark the diameter or radius of an arc or circle.

When the **Circular Dimension** tool is selected, the circular icons are displayed in the mode bar. These icons are used to select the type of dimensioning operation you want to perform.



The **Leader** icons are used by all four circular dimensioning modes. They specify the position of the leader line relative to the measurement label (i.e., whether the line is placed to the left or right of the text).

For additional information, see the following topics:

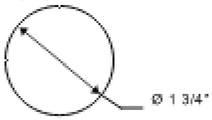
Interior Diametrical Dimension

Exterior Diametrical Dimension

Interior Radial Dimension

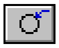
Exterior Radial Dimension

The **Interior Diametrical** mode is used to draw an interior line representing the diameter of a circle or arc.

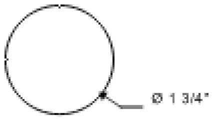


To create an Interior Diametrical dimension:

1. Select the **Circular Dimension** tool from the **Standard Dimension** palette.
2. Click the **Left Leader** or **Right Leader** icon, depending upon which side of the label you want the leader line to stem from.
3. Select the **Interior Diametrical** icon in the mode bar.
4. Position the cursor inside the arc or circle.
5. Press the mouse button and while holding it down, drag the mouse toward the opposite side of the arc or circle.
   - If you want the measurement to appear inside the circle or arc, release the mouse button on the opposite side of the circle.
   - If you want the measurement to appear outside the circle or arc, drag the mouse outside of the arc or circle before releasing the mouse button.



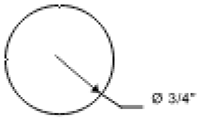
The **Exterior Diametrical** mode is used to mark a circle or arc's diameter on the outside of the object.



To create an Exterior Diametrical dimension:

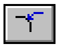
1. Select the **Circular Dimension** tool from the **Standard Dimension** palette.
2. Click the **Left Leader** or **Right Leader** icon, depending upon which side of the label you want the leader line to stem from.
3. Select the **Exterior Diametrical** icon in the mode bar.
4. Position the cursor inside the arc or circle.
5. Press the mouse button and drag the cursor to the point where you want the diameter to appear.
6. Release the mouse button.

The **Interior Radial** mode is used to draw an interior line representing the radius of a circle or arc.

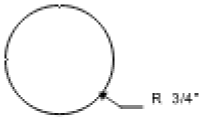


To create an Interior Radial dimension:

1. Select the **Circular Dimension** tool from the **Standard Dimension** palette.
2. Click the **Left Leader** or **Right Leader** icon, depending upon which side of the label you want the leader line to stem from.
3. Select the **Interior Radial** icon in the mode bar.
4. Position the cursor inside the arc or circle.
5. Press the mouse button and while holding it down, drag the mouse toward the opposite side of the arc or circle.
    - If you want the measurement to appear inside the circle or arc, release the mouse button on the opposite side of the object.
    - If you want the measurement to appear outside the circle or arc, drag the mouse outside of the arc or circle before releasing the mouse button.

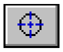


The **Exterior Radial** mode is used to mark a circle or arc's radius on the outside of the object.



To create an Exterior Radial dimension:

1. Select the **Circular Dimension** tool from the **Standard Dimension** palette.
2. Click the **Left Leader** or **Right Leader** icon, depending upon which side of the label you want the leader line to stem from.
3. Select the **Exterior Radial** icon in the mode bar.
4. Position the cursor inside the arc or circle.
5. Press the mouse button and drag the cursor to the point where you want the radius to appear.
6. Release the mouse button.



The **Center Mark** tool on the **Standard Dimension** palette is used to place a center mark in a circle, ellipse, rectangle or rounded rectangle.

Note: This tool lies beneath the **Circular Dimension** tool on the **Standard Dimension** palette. To access the **Center Mark** tool, press and hold the **Circular Dimension** icon. When the pop-out icons appear, release the mouse over the **Center Mark** tool.

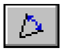
To create a center mark in a circle, ellipse, rectangle or rounded rectangle:

1. Select the **Center Mark** tool from the **Standard Dimension** palette.
2. Click the object you want to mark.

   The center mark appears.

To create a center mark in the corner of a rounded rectangle:

1. Select the **Center Mark** tool from the **Standard Dimension** palette.
2. Press the **Control** key and click the corner of the rectangle you want to mark.



The **Angular Dimension** tool on the **Standard Dimension** palette is used to measure and mark an angle.

When the **Angular Dimension** tool is selected, the angular icons are displayed in the mode bar. These icons are used to select the method you want to use to specify the angle.

For additional information, see the following topics:

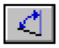
Angular Dimension
Reference Dimension

The **Angular Dimension** mode is used to specify an angle from surface to surface.

To mark an angle in Angular mode

1. Select the **Angular Dimension** tool from the **Standard Dimension** palette.
2. Click the **Angular** icon in the mode bar.
3. Position the cursor over the first line (or surface) to be dimensioned.
4. Press and hold the mouse button down.
5. Drag the mouse to the other line (or surface).
6. When the cursor is positioned correctly, release the mouse.

   An outline of the dimension appears.
7. Move the mouse to position the dimension inside or outside the angle as needed.
8. Click the mouse to set the dimension.



The **Reference Dimension** mode is used to specify an angle between a reference line and a surface.

To mark an angle in Reference Dimension mode

1. Select the **Angular Dimension** tool from the **Standard Dimension** palette.
2. Click the **Reference Dimension** icon in the mode bar.
3. Draw a reference line. This line may be drawn at any angle.
4. Place the fingertip cursor on the surface forming the opposite side of the angle.

   An outline of the dimension appears.
5. Move the mouse to position the dimension inside or outside the angle as needed.
6. Click the mouse to set the dimension.

The **Tape Measure** tool on the **Standard Dimension** palette is used to measure the distance between points on the drawing. This tool reports a linear measurement but does not place it in the drawing.

Multiple, consecutive, line segments can also be measured with this tool. When multiple segments are drawn, the length of the active segment and the length of the total line are reported in the **Data Display** bar.

To measure between two points:

1. Select the **Tape Measure** tool from the **Standard Dimension** palette.
2. Position the cursor at the first point on the line.
3. Drag the cursor to the second point and release the mouse.

   The length of the line segment is shown in the **L** field of the **Data Display** bar. The length of the total line is shown in the **TL** field.
4. If you want to add another segment to the line, move the mouse and click at the segment's end point.
5. When you are finished, double-click the mouse.

   The tape measure disappears.

The **Protractor** tool on the **Standard Dimension** palette is used to measure the angle between points on the drawing. This tool reports the angle measurement, but does not place it in the drawing.

When the **Protractor** tool is selected, the protractor icons appear in the mode bar. They let you measure the angle in one of two ways.

For additional information, see the following topics:

Two Lines Mode

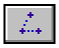
Three Points Mode

The **Two Lines** mode is used to measure the angle between two objects.

To use Two Line mode:

1. Select the **Protractor** tool from the **2D Tool** palette.
2. Select the **Two Lines Mode** icon in the mode bar.
3. Place the cursor on one of the objects to be measured.
4. Drag the cursor to the second object.

   The angle is reported in the **Data Display** bar. It will be displayed until you release the mouse button.

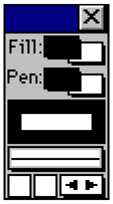


The **Two Lines** mode is used to measure the angle between two lines that you draw.

To use Three Points mode:

1. Select the **Protractor** tool from the **2D Tool** palette.
2. Select the **Three Points Mode** icon in the mode bar.
3. Position the mouse at the start point and click.
4. Move the mouse to the end of the first line and click.
5. Move the mouse to form the second line.

   The angle is reported in the **Data Display** bar.
6. Click to end the operation.



The **Attribute** palette is used to control the line and color attributes of a graphic object.

The **Attribute** palette is opened using the **Attributes** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette.

When an object is selected, the **Attribute** palette displays the attributes of the object. To change the object's attributes, simply change the attribute in the palette. Multiple objects can be selected and changed in this way too (when multiple objects are selected, the palette displays the attributes of the first object selected).

To change default attributes, change the **Attribute** palette while nothing is selected Default attributes are the ones automatically assigned to all new objects you create.

For additional information, see the following topics:

Fill Foreground & Background Color
Pen Foreground & Background Color
Patterns
Line Style & Weight
Marker Style

The two **Fill** swatches on the **Attribute Palette** are used to specify the colors used to fill an object. The top swatch specifies the fill pattern's foreground color and the rear swatch specifies the fill pattern's background color.

Generally, when a solid fill is used, the background swatch is the one that specifies the color.

To change a fill color, click the swatch you want to change, then, in the 256-color pop-out palette, select a new color.

Note: The content of the color palette can be changed using the **Edit Attributes** command.

The two **Pen** swatches on the **Attribute Palette** are used to specify the colors assigned to the pen's pattern. The pen produces the outline of an object.

The top swatch specifies the pen's foreground color and the rear swatch specifies the pen's background color.

Generally, when a solid pen pattern is used, the foreground swatch specifies the color of the pen.

To change a pen color, click the swatch you want to change, then, in the 256-color pop-out palette, select a new color.

Note: The content of the color palette can be changed using the **Edit Attributes** command.

The **Pattern** segment of the **Attribute Palette** is used to select the pattern that will be used for the pen and for the fill.

Note: Both the pen and fill patterns can be set to "no pattern," making them invisible.

The outside box in this segment represents the pen's pattern. To change the pen's pattern, click the ring, and in the pop-up menu, select a new pattern.

The center box in this segment represents the fill pattern. To change this pattern, click the box, and select a new pattern from the pop-up menu that is opened.

Note: The content of the pattern pop-up menu can be changed using the **Edit Attributes** command.

The **Line Style** segment of the **Attribute** palette is used to specify a line's thickness and its dash pattern.

To change a line's attributes, click the **Line Style** segment in the **Attribute Palette**.

In the top section of the menu, specify the line's thickness. If you want to use a thickness that is not listed on the menu, select the **Set Thickness** option at the top of the menu, and specify a custom thickness.

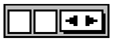
If you want the line's dimensions to be displayed with the line, select the **-- xx --** option in the middle of the menu.

In the bottom section of the menu, select the line's dash pattern.

Note: The content of the line thickness and dash pattern portions of the pop-up menu can be changed using the **Edit Attributes** command.

For related information, see the following topics:

Marker Style



The objects at the bottom of the **Attribute** palette are used to assign line markers to a line (often arrowheads).

The large button in the lower-right corner is used to select which style of marker is used. To change the marker style, click this button. In the marker style menu, select a new style. If the style you want to use isn't listed, select the **Edit** option to create a new one (or use the **Edit Attributes** command).

The two small buttons in the lower left corner are used to assign line markers to a line. When enabled, the left button assigns a marker to the beginning of the line, and the right button, the end (if a line is to be drawn without line markers, these two buttons should be disabled).

For related information, see the following topics:

Line Style & Weight

The **Working Plane** palette is used to select and save a working plane. (See **An Overview of Working Planes** for additional information on working palettes).

The **Working Plane** palette is opened using the **Working Planes** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette. It can be moved to any position on the screen by dragging its title bar.

A working plane is activated by selecting it from the **Saved Working Planes** list, or by activating one of the current, unnamed planes using the **Current Working Plane** buttons.

The **Working Plane** palette contains the following controls

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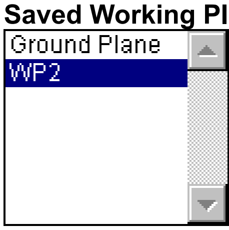
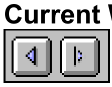
Saved Working Planes



The **Saved Working Planes** list displays the names of all saved working planes. To activate one of these planes, double-click its name in the list.

Current Working Plane



The **Current Working Plane** buttons are used to toggle among the current working planes. The ten most recently created working planes are maintained in an unnamed list. To activate one of these working planes, click the left or right arrow to cycle through the list of current working planes.

Add

The **Add** button is used to permanently save the active working plane. When you click **Add**, the **Assign Name** dialog box is opened. Enter a unique name for the working plane and click **OK**. The name will be added to the **Saved Working Planes** list at the top of the **Working Planes** palette, and can be activated at any time by double-clicking it.

Delete

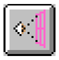
The **Delete** button is used to remove a saved working plane. To delete a working plane, highlight its name in the **Saved Working Planes** list at the top of the palette and click **Delete.**

Note: The **Ground Plane** cannot be deleted or renamed.

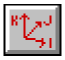
Rename

The **Rename** button is used to assign another name to a saved working plane. To rename a working plane, highlight its name in the **Saved Working Planes** list at the top of the palette and click **Rename**. Enter a new, unique name in the dialog box and click **OK**.

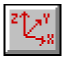
Note: The **Ground Plane** cannot be deleted or renamed.



The **Top View** button is used to place the drawing in top view.



The **i, j, k** button is used to place the display in i, j, k mode.



The **x, y, z** button is used to place the display in x, y, z mode.

In MiniCad, all objects are created on the working plane which, by default, is also the ground plane. However, the working plane can be changed so that you can create 3D objects in place, rather than having to create them on the ground plane, and rotate them into position afterwards.

On a color monitor, MiniCad displays the ground plane in light blue and the working plane in pink. The *x* and *y* axes of the working planes are defined by small dots or handles, at the edge of the plane. The center of the working plane also is represented with a handle. If the **Show 3D Axis Labels** preference is active, the two axes are shown with their respective labels.

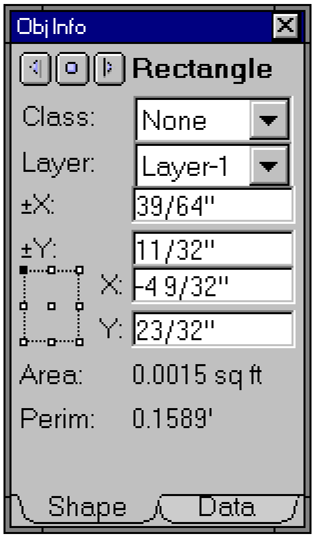
The working plane's position can be changed with the **Set Working Plane** tool or by moving one of its handles. A working plane's angle can be changed by dragging one of its handles or by entering a value in the plane's **Angle** field on the **Data Display** bar.

When the cursor is placed directly over one of the working plane handles, the **SmartCursor** shows you the axis that is affected by the movement of that handle. Additionally, a small marker indicates what movement is possible with that handle.

When you drag the plane to a 30°, 45°, 60°, or 90° angle, the cursor's dotted line becomes a solid line and, if the **Use Sound** preferences is set, you will hear a click.

MiniCad automatically maintains a list of the last ten working planes (more than enough for most drawings). These working planes are recorded in the order in which they were created. They can be accessed using the **Current Working Plane** buttons on the **Working Plane Palette**. When the eleventh working plane is created, the first plane on the list is purged.

A plane can be permanently saved using the **Add** button on the **Working Plane** palette. Saved planes are listed at the top of the **Working Plane** palette, and can be activated by double-clicking its name in the list.



The **Object Info** palette is used to inspect or change an object's size, shape or position. It is also used to inspect or change information relating to an object's database record.

This palette has two panes:

The **Shape** pane contains an object's size shape and position information. It will also report the area and perimeter of a closed shape.

The **Data** pane contains an object's database information.

You can switch between panes by clicking the tabs at the bottom of the palette.

The **Object Info** palette is opened using the **Object Info** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette. It can be moved to any position on the screen by dragging its title bar

For additional information, see the following topics:

The Shape Pane

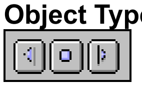
The Data Pane

The **Shape** pane on the **Object Info Palette** is used to inspect or change the size, shape or position of a selected object(s). It can also be used to change the layer on which an object is placed, and the class to which it is assigned.

The specific contents of the **Shape** pane varies depending upon the type of object you have selected (e.g., when an Arc is selected, this palette will contain a **Sweep** field, when a polyline is selected it will contain a **Vertex** list, and so forth). However, it always has the following basic controls.

Note: When a group is selected, only its bounding box and position can be edited. To select individual objects in the group you must use the **Edit Group** command or use the **Ungroup** command to break the group into its component parts.

Object Type



An object's type is always displayed at the top of the palette next to the **Object Selection** buttons. The arrow buttons can be used to step through a multiple selection one object at a time. The center button is used to identify the currently selected object.

If nothing is selected, this field will read "No Selection."

Class

The **Class** field displays the selected object's class. To change an object's class assignment, click the **Class** field and select a new class from the pop-up menu.

If no objects are selected, selecting a class from this menu will change the active class.

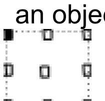
For more information about classes, see the **Classes** command.

Layer

The **Layer** field displays the layer on which the selected object exists. The object can be moved to another layer by clicking the **Layer** field and selecting another layer from the pop-up menu (if it is moved to an invisible layer, or one that is not active, the object will disappear).

X, Y, Z Fields

The **X**, **Y** and **Z** fields on the **Shape** pane represent the object's position in the drawing. You can move an object by changing the values in these fields.



The X, Y, Z fields are accompanied by a diagram of the object's bounding box, which specifies the handle aligned on the point specified by **X**, **Y**, **Z**.

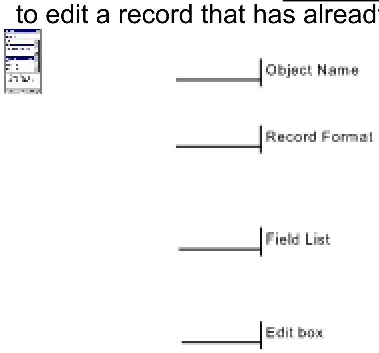
Note: When a 3D object is selected, the **Shape** pane contains buttons allowing you to specify the object's position relative to the ground plane (**X**, **Y**, **Z**) or the working plane (**I**, **J**, **K**).

ΔX, ΔY ΔZ Fields

The **ΔX**, **ΔY** and **ΔZ** fields on the **Shape** pane represent the object's size. You can change the object's size by entering new values in these fields.

Note: When a line object is selected the **Shape** pane contains buttons allowing you to specify the line's length using **ΔX**, **ΔY** coordinates or using polar coordinates (i.e. length and angle).

The **Data** pane on the **Object Info Palette** is used to attach a database record to the selected object, or to edit a record that has already been attached. It is also used to assign a unique name to an object.



Object Name

This field is used to assign a unique name to an object in the drawing. To give a name to an object, click in the **Object Name** field and type a new name. This name must be different from the name of any other object, layer, class, symbol or record format in the drawing.

Record List

This segment of the **Data Pane** specifies the format of the record assigned to the object. It contains a list of all record formats in the drawing. See the **Resource Palette** for more about record formats.

To attach a record to the selected object, click the empty checkbox next to the record format's name in the list. If the selected object is a symbol, this record will also be attached to the symbol in the library. This means that, from now on, it will automatically be assigned whenever the symbol is inserted.

Note: Changing the record format assignment of a symbol does not affect instances of the symbol already placed in the drawing. If you wish to attach a record to existing instances of this symbol, you must use the **Data Pane** to attach it to each instance individually.

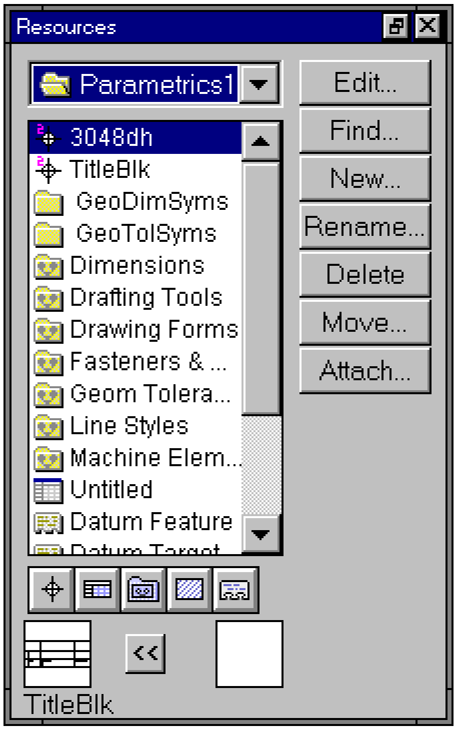
To detach the object's record, click the checkbox to uncheck it.

If the checkbox contains a grayed checkmark, some of the selected objects are attached to the record. If you click the checkbox, the record will be attached to all selected objects.

Field List

This segment of the **Data** pane displays the individual fields of the selected object's database record. You can change the contents of this record by selecting the field's name in the list and editing its contents at the bottom of the palette.

If you change the contents this field when multiple objects are selected, your change will be applied to the records of all selected objects.



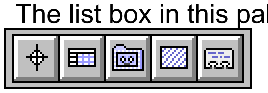
The **Resource** palette is used to access and manage the following components (i.e., resources):

- **Symbols**
- **Worksheets**
- **Macros**
- **Hatch Patterns**
- **Record Formats**

Except for macros, which can be stored as external text files, all of these resources are maintained within your drawing. They are created, accessed and edited using the **Resource** palette. When necessary, the **Resource** palette can also be used to import these resources from other drawings.

The **Resource** palette is opened using the **Resources** command on the **Window** menu. It is closed by clicking the **Close** box in the corner of the palette. It can be moved to any position on the screen by dragging its title bar.

The list box in this palette displays the resources in the selected file.



By default, the resource list shows *all* resources in the selected file. To simplify the display, you can use the **Show/Hide** buttons at the bottom of the palette to limit the list to a particular resource type or type(s).

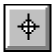
When a **Show/Hide** button is pressed, resources of that type are "hidden." That is, they are not displayed in the **Resource Palette**'s list box.

When a resource type is hidden, its button is displayed with a large "X' on its face. For example, if you want only symbol and worksheet resources to appear in the palette, you would click the **Macros**, **Hatch Pattern** and **Record Format** buttons to exclude these resources from the list. These 3 buttons would be displayed with a large, red "X" across each of them.

The **Show/Hide** buttons are toggles. To unblock a suppressed resource, simply click the button again.

When the **Resource Palette**'s list box contains external file names, not drawing-file resources, these buttons are dimmed. They are automatically re-activated when a drawing file or library is entered.

The **Resource Palette** is used to select, edit and import symbols. A symbol is an object that has been saved into the drawing's symbol library using the **Create Symbol** command.



Note: Check the status of the **Symbol** button at the bottom on the palette. If the button has a large, red "X" across it, click it to allow symbols to be listed in the palette.

Before a symbol can be placed in a drawing, it must "selected" in the **Resource** palette. This makes it the "active" symbol, which means it is available to, and can be placed with, the **Insert 2D Symbol** or **Insert 3D Symbol** tool.

tip! You can also select the active symbol my double-clicking the **Insert 2D Symbol** or **Insert 3D Symbol** icon. This action will open the **Resource Palette**.

For additional information, see the following topics:

Selecting Symbols
Importing Symbols
Editing Symbols
Creating a New Symbol Folder
Renaming a Symbol
Deleting Symbols
Moving Symbols to other Folders
Attaching Symbols to a Record Format

Symbols are selected from the scrolling list in the **Resource Palette**. When a symbol is selected, its thumbnail is displayed in the palette's preview window in the lower-right corner.

If you want to make this the "active" symbol, click the **<<** button at the bottom of the palette. This action copies the symbol to the window in the lower-left corner, which makes it the "active" symbol.

Symbols can be selected from the drawing's internal library, or from an external MiniCad file (e.g. another drawing or library). To select a symbol from an external file, select the file using the pop-up menu at the top of the palette, then select a symbol from within it.

Note: The **Find** button can be used to locate a symbol anywhere on your system. When clicked, the **Find** button opens the **Find Resource** dialog box. Enter the name of the symbol you want to locate, use the **Set Where** button to set the search path and click **OK**.

When you make an external symbol, the "active" symbol, it will automatically be imported into your drawing the first time it is placed. If your drawing already contains a symbol with the imported symbol's name, MiniCad compares the two versions. (MiniCad time-stamps symbols, so it knows which version is the most current.)

If the versions are the same, the version already in the drawing's library is used. If the versions are different, you will be asked whether the older version of the symbol should be replaced by the newer one.

For related information, see the following topics:

Symbols
Importing Symbols
Editing Symbols

The **Resource** palette can be used to import symbols from other drawings. This process will happen automatically if you select and place a symbol from an external file.

You can also import an entire folder of symbols. When an external folder is selected in the **Resource Palette**, it can be imported by clicking the **Import** button on the palette.

If your drawing already contains a symbol with an imported symbol's name, you will be asked whether the existing symbol should be replaced by the imported one.

If an symbol has components in different classes, this class information is retained when the symbol is imported. If it contains class information that differs from the active file, that class information is automatically added to the active file.

If you import a folder(s) of symbols, but don't end up using all of the symbols in it, the unused symbols can easily be removed with the **Purge Unused Objects** command.

For related information, see the following topics:

Symbols

Selecting Symbols

Editing Symbols

Renaming a Symbol

Deleting Symbols

Creating a New Symbol Folder

Moving Symbols to other Folders

A symbol can be edited by selecting its name in the **Resource Palette** and clicking the **Edit** button. You can also edit a symbol by double-clicking its name in the list box.

When the **Edit Symbol** dialog box is displayed, choose whether you want to edit the symbol's 2D or 3D component.

The symbol will be presented in the **Symbol Edit** window. Make the changes you need.

Important! Any edits you make will be applied to all instances of this symbol in the drawing.

The symbol's insertion point is represented by the *x, y* axes of the window. On a 3D symbol, the symbol's *z* dimension can be changed by editing the *z* value.

When you are finished editing the symbol, close the window. The symbol will be updated. If you had added 2D components to a 3D symbol (or vice versa), MiniCad will automatically apply them to the appropriate version when it updates the symbol

You can also edit a symbol directly in the drawing using the **Edit Group** command.

For related information, see the following topics:

Symbols
Selecting Symbols
Importing Symbols
Renaming a Symbol
Deleting Symbols
Creating a New Symbol Folder
Moving Symbols to other Folders

The **New** button on the **Resource Palette** is used to create folders to organize your symbol library. When the **Symbol Folder** button is clicked, a dialog box opens for the new folder's name. Like all objects in MiniCad, symbol folders must be given a unique name.

For related information, see the following topics:

Symbols
Moving Symbols to other Folders
Importing Symbols
Renaming a Symbol
Deleting Symbols

Symbols in the active document's library can be renamed. To rename a symbol, select its name in the **Resource Palette** and click the **Rename** button. A dialog box opens to enter the new name for the symbol.

Every symbol in a drawing must have a unique name. If a symbol is giving a name that already exists, you will be asked whether you want to replace the existing resource with the newer one.

For related information, see the following topics:

Symbols
Selecting Symbols
Creating a New Symbol Folder
Moving Symbols to other Folders
Deleting Symbols

Symbols can be deleted from the active document's library by selecting the symbol's name in the **Resource Palette** and clicking the **Delete** button.

In the drawing, any instances of the deleted symbol are removed and replaced with loci.

For related information, see the following topics:

Symbols
Selecting Symbols
Editing Symbols
Renaming a Symbol
Moving Symbols to other Folders

Symbols can be consolidated into folders within the symbol library. To move a symbol to another folder, select the symbol's name in the **Resource Palette** and click the **Move** button. Multiple symbols can be selected for this operation.

A list of symbol folders in the library is opened. The pop-up menu at the top of the dialog box is used to navigate through the folders in the library.

Select the desired folder and click the **Open** button. Once the folder is open, click **OK**.

For related information, see the following topics:

Symbols
Creating a New Symbol Folder
Selecting Symbols
Renaming a Symbol
Deleting Symbols

To assign a record format to a symbol, select the symbol's name in the **Resource Palette** and click the **Attach** button. In the **Attach Record** dialog box, select the record format to be attached to the symbol. Click **OK**.

From this point forward, the specified database record will automatically be assigned to the symbol whenever it is place.

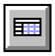
Note: This action does not affect any instances of this symbol already in the drawing. If you wish to assign a database record to an existing instance of the symbol, you must attach it using the **Data** pane in the **Object Info Palette**.

A record format can be "detached" from a library symbol by selecting the symbol in the **Resource** palette, clicking the **Attach** button, and deselecting the record's checkbox in the **Attach Record** dialog box.

For related information, see the following topics:

Symbols
Selecting Symbols
Editing Symbols
Record Formats

The **Resource Palette** is used to create, open, and import worksheets.



Note: Check the status of the **Worksheet** button at the bottom of the palette. If the button has a large, red "X" across it, click it to allow worksheets to be listed in the palette.

MiniCad worksheets provide a fully functional spreadsheet facility with over thirty built-in functions. Additionally, MiniCad's worksheets can return virtually any 2D information contained in a drawing. They can be used to perform counts, calculations and generate schedules.

A worksheet can be linked to other worksheets within the same drawing. You can also export information to other spreadsheets and databases, and import information from other MiniCad drawings or from other spreadsheet applications.

For additional information, see the following topics:

Creating a Worksheet

Opening a Worksheet

Importing a Worksheet

Renaming a Worksheet

Deleting a Worksheet

Basic Worksheet Editing

The Worksheet Window & Commands

The **New** button on the **Resource** palette is used to create a new worksheet in a drawing. When you click the **New** button, the **Create Resource** dialog box opens. Select the **Worksheet** button, then click **Create**.

You will be asked to name the worksheet. This name must be different from any other named resource or object in the drawing.

You will also be asked to specify the number of rows and columns your worksheet requires. The default is ten rows and five columns. Be aware that every cell within a worksheet has an overhead of 4 bytes per cell, so don't make the worksheet unnecessarily large. Specify a size that will initially accommodate your needs and click **OK**. The worksheet can be easily resized later if you need to make it larger.

A worksheet editing window is opened in which you can create your worksheet. Close the window when you are finished working with it.

For additional information, see the following topics:

Importing a Worksheet

Basic Worksheet Editing

The Worksheet Window & Commands

A worksheet in the drawing can be opened by selecting its name from the list box in the **Resource Palette** and clicking the **Open** button. You can also double-click the worksheet's name to open it.

When a worksheet is opened, it is presented in its own a worksheet editing window.

For related information, see the following topics:

Importing a Worksheet
Basic Worksheet Editing
The Worksheet Window & Commands

The **Resource** palette can be used to import worksheets from other drawings.

A worksheet can be imported by selecting its name from the list box in the **Resource Palette** and clicking the **Import** button. You can also double-click the worksheet's name to import it.

Note: The **Find** button is can be used to locate a worksheet anywhere on your system. When clicked, the **Find** button opens the **Find Resource** dialog box. Enter the name of the worksheet you want to locate, use the **Set Where** button to set the search path and click **OK**.

When a worksheet is imported, the original worksheet is copied into the active drawing. The original worksheet is not affected by any editing you do to it in the active drawing.

If the imported worksheet's name already exists in your drawing, you will be asked whether you want it to replace the one in your drawing.

Note: The **Copy** and **Paste** commands in the drawing window can also be used to move portions of a worksheet to another worksheet. Single cells, or a range of cells can be copied.

Spreadsheets from applications such as Excel or Lotus can be imported using the **Import** command on the **File** menu.

For related information, see the following topics:

Basic Worksheet Editing

The Worksheet Window & Commands

Worksheets can be renamed by selecting the worksheet's name in the **Resource Palette** list box and clicking the **Rename** button. You will be prompted for a new name. Remember that every worksheet in a drawing must have a unique name.

Worksheets can be deleted from the active document by selecting the worksheet's name in the **Resource Palette** list box and clicking the **Delete** button.

Deleting a worksheet permanently removes it and its contents from a drawing. When you delete a worksheet you will be asked to confirm your request.

Anyone familiar with spreadsheets will be right at home using MiniCad's worksheet facility because it is similar to many common spreadsheet programs.

Keep the following in mind when using MiniCad's worksheets:

- Select a range first, then execute a command.
- A row is either a spreadsheet row or a database row.
- Enter a formula using an equal sign (=) as its first character.
- Use two periods (..) to denote a range of cells, instead of the colon (:) used by Microsoft Excel.
- A worksheet uses its own menus for worksheet functions and the main drawing window menus for text (font, font style) and editing (cut, copy and paste).
- A worksheet can be placed in the drawing.
- Worksheets are printed independent of the drawing.

Worksheets can be linked to share information. In addition, the formulas and values on a worksheet can be copied and pasted from one worksheet to another.

A worksheet row can be used as either a spreadsheet or a database. Spreadsheets have cells which contain formulas or values.

A database row is linked to a specific criteria, as selected by the **Search Criteria** dialog box. A database row becomes a header for the records that match its criteria. The matching records are listed as subrow beneath the database row.

Each time the database row is recalculated, it expands or contracts to hold the number of objects matching its search criteria. Each subrow in the database returns a value based on the formula in the header.

For related information, see the following topics:

The Worksheet Window & Commands

Every worksheet opens in its own window with its own scroll bars, cursors, and separate menus.

There are two menus in a MiniCad worksheet.

- The **Main Menu** is accessed by clicking the menu icon in the upper left corner of the worksheet. This menu is used to perform global tasks such as recalculating and printing.
- The **Row Menu** is accessed by clicking a row heading. This menu is used to choose between spreadsheet and database functions. It is also used to determine functions and search criteria used for the database row.

To the right of the menu icon are three icons that are used with database rows. These icons are active only when a **Database Row** is selected.

- The first two are sort icons, used to sort database rows into ascending and descending order. Drag either the **Ascending** or the **Descending** icon to the column heading of the column to be sorted. Up to three sort icons can be placed on a **Database Row**.
- The last icon is a summation icon. The **Summation** icon can be dragged to a column heading for a **Database Row**, where it sums all subrows that meet the search criteria. Up to three summation fields can be placed in a database row

For additional information, see the following topics:

Main Menus (worksheets)
Row Menus (worksheets)
Entering Worksheet Data
Selecting Cells & Moving Around the Worksheet
Using a Worksheet as a Database
Placing a Worksheet as a Graphic
Resizing worksheets

The **Main** menu is accessed by clicking the menu icon in the upper left corner of the worksheet. The menu is used to perform global tasks such as recalculating and printing.

This menu contains the following worksheet commands:

Recalculate
Paste Criteria
Paste Function

Database Headers
Number
Alignment
Border
Column Width
Preferences

Insert
Delete

Page Setup
Print

The **Recalculate** command on a worksheet's **Main** menu updates the worksheet to reflect a change in the drawing, a change to the values on the worksheet, or to update external references.

MiniCad calculates on the basis of the stored value in a cell, not the displayed values.

The **Paste Criteria** command on a worksheet's **Main** menu displays the **Search Criteria** dialog box, which allows the construction of complicated conditional formulas to be used in your worksheet.

Search criteria are objects or attributes that can be used as a standard on which a judgment can be made. The following are search criteria codes used in a MiniCad formula. They are placed in a formula, with proper parentheses and syntax by the **Search Criteria** dialog box. They can also be typed directly in the formula edit bar.

Object	Symbol	Object	Symbol
Object Name	N	Class Name	C
Layer Name	L	Fill Pattern	FP
Line Weight	LW	Line Style	LS
Object Type	T	Symbol Name	S
Selected Status	Sel	Object Record	R
Pen Pattern	PP	Fill Foreground	FF
Fill Background	FB	Pen Foreground	PF
Pen Background	PB	Arrowhead	AR
Visibility	V	Every Object	All

The **Paste Function** command on a worksheet's **Main** menu presents a list of available functions, and places the selected function in the **Formula Edit Bar**.

Worksheet functions can be used to calculate and return values based on information in the drawing or that you supply directly.

Worksheet functions take a value, perform some operation on them, and return a result. Functions are calculated as soon as you enter them. However, if auto recalculation is off, any subsequent changes do not calculate automatically.

See ***MiniCad Functions*** for a complete description of the available functions.

A **Database Row** becomes a header for subrows, and any totals generated by the subrows are placed in the **Database Row**.

The **Database Headers** command determines whether a **Database Row** is displayed. When the command is enabled, the **Database Row** is visible Deselecting the command hides the **Database Row.**

Database Rows that are not displayed can still be referenced.

The **Number** command on a worksheet's **Main** menu determines how a selected cell is formatted.

Each cell in a MiniCad worksheet can be formatted to display numbers in a particular way. The stored value in the cell is stored as an extended-precision floating point number, which is accurate to about nineteen places.

Note: The format in a database header controls how subrow cells are displayed.

When the **Number** command is selected, the **Number** dialog box is opened. It contains the following format types:

General

This format displays the whole numbers as written (internally it generates six decimal places, strips off any trailing zeros, and displays the result). Values that MiniCad interprets as text are aligned to the left side of the cell. Values that are numbers or integers are aligned to the right.

Decimal

When formatting a cell in decimals, you can choose to display commas with the number (by selecting the check box) and the number of decimal places that are displayed (by entering the value in the **Decimal Places** field). To display a value as currency, enter the appropriate symbol in the Leader field ($ or £ for example).

Scientific

This displays the number in scientific notation. Enter the desired display accuracy in the decimal place field.

Fractional

The **Fractional** option controls the denominator for rounding. For instance, enter a 4 if you want the numbers rounded to 1/4 unit.

Dimension

Formatting a cell for dimensions displays the contents with the current unit setting as set in the **Units** command on the **Page** menu.

Angle

This option formats the cell in degrees, degrees and minutes, or degrees, minutes and seconds. The selection is made from a drop menu.

Date

MiniCad can display dates and time in thirteen combinations. The choices and results are shown below.

Format	Example	Format	Example
mdy	5/17/97	mdy hmm	5/17/97 1:20
dmy	17/5/97	dmy hmm	17/5/97 1:20
ymd	97/5/17	ymd hmm	97/5/17 1:20
d-mmm-y	17 May 97	d-mmm	17 May
mmm-y	May 97	h mm	1:20
h mm s	1:20:30	h mm (AM/PM)	1:20 PM
h mm s (AM/PM)	1:20:30 PM		

Boolean

A **Boolean** formula returns a true or false value when evaluated. A **Boolean** cell can display a specific value depending on the evaluation of the formula. When **Boolean** is selected in the **Number** dialog box, fields labeled **True** and **False** are displayed.

To format a **Boolean** field, enter the values you want displayed when the formula is evaluated as true in the **True** field. Then, in the **False** field, enter the value you want displayed when the formula is evaluated as false. The fields can contain up to seven characters.

Leader

The characters in this field are prefixed to the value in the cell. The leader can be up to seven characters.

Trailer

The characters in this field are appended to the value in the cell. The trailer can be up to seven characters.

The **Alignment** command on a worksheet's **Main** menu controls the alignment of the value within a cell. The default is **General**. The most recent setting overrides the previous setting.

The **Border** command on a worksheet's **Main** menu specifies the type of border that is to be placed around the selected cell(s).

Individual cells, or ranges of cells, can have a border. The border can be on any or all of the four sides of the cell or range. The border color is always black, regardless of the pen color setting in the **Attribute** palette.

To put a border around a range, select the range and select the **Outline** checkbox in the **Border** dialog box. To put a border around every cell in a range, select **Top**, **Bottom**, **Left** and **Right**.

The **Column Width** command on a worksheet's **Main** menu lets you control the column's width in points (there are 72 points to an inch).

The default column width is 96 points.

Setting a column width to zero hides the column. (A hidden column can still be referenced.)

To redisplay a column of zero width, select the column to the left and right of the hidden column, then set the column width to a value greater than zero. The columns are displayed at the specified width, and you can resize them individually as required.

Note: Row height is not directly controllable in MiniCad. However, you can indirectly increase a row's height by increasing the font size within a cell. Rows cannot be hidden.

You can also increase the column width directly on the worksheet. When the cursor is dragged over a divider in the column header, it changes into the **Column Resize** cursor. Click on the divider with the **Resize Cursor** and drag the column to a new size.

The **Preferences** command on a worksheet's **Main** menu is used to specify the following preferences for the active worksheet.

Note: These preferences are applied to the open worksheet only. They do not affect any other open worksheet, nor are they carried over to new worksheets created in the drawing file.

Header
The header is text that appears at the top of every printed page of the worksheet. Only one font or style can appear in a header, and it must be text.

Footer
The footer is text that appears at the bottom of every printed page of the worksheet. Only one font or style can appear in a footer, and it must be text.

Margins
The page margins are set here. Be careful to stay within the margin required by your printer.

Display
The check boxes control how these items display.

Show Grid
When checked, the reference grid is displayed. When displayed, the grid is printed.

Show Tabs
Shows tabs when the worksheet is in the drawing.

Auto-Recalculation
When checked, the worksheet recalculates internal references each time a formula is accepted.

The **Insert** command on a worksheet's **Main** menu allows you to add a new column or row to an existing worksheet.

Data in the worksheet is not altered by an insertion, but a careless insertion can have negative effects because a referenced cell can be displaced. Columns and rows that are inserted into ranges are included in the range. Only whole rows and columns can be inserted in a worksheet. Any number of columns can be inserted into a worksheet.

To insert a column, select the column just to the right of where you want the new column inserted, and choose the **Insert** command. Multiple columns can be inserted.

Columns and rows can be inserted into ranges of cells but the range itself does not expand. The new rows are substituted for the existing rows displaced when the **Insert** command was chosen. This may have undesirable results. For example, if a new row is inserted into an existing range of cells that are referenced by a formula, the new row(s) will displace the cells in the range. This will cause the results of the formula to change. Always inspect a worksheet carefully before inserting rows to be sure you do not make inadvertent changes to formulas.

The **Delete** command on a worksheet's **Main** menu deletes a selected column or row from a worksheet.

Be careful when deleting a column because you may not be aware of all referenced cells in the column to be deleted. Ranges that have had a column removed are reduced accordingly. Any formulas or values on the deleted rows and columns are removed.

To delete rows or columns from the worksheet, select the column(s) or row(s) to be deleted and choose the **Delete** command.

The **Page Setup** command on a worksheet's **Main** menu displays the currently selected printer's **Page Setup** dialog box. Selections remain active until changed and affect all items printed.

The **Print** command on a worksheet's **Main** menu prints the active worksheet using the currently selected printer.

A **Row** menu is opened when a row number is clicked. Commands selected from a **Row** menu affect only the selected row.

Spreadsheet
Database

Set Criteria
Edit Criteria

Select Data Items

The **Spreadsheet** command on a worksheet's **Row** menu designates the row as a **Spreadsheet Row**. Spreadsheets have cells that contain formulas or values.

The **Database** command on a worksheet's **Row** menu designates the row as a **Database Row**. A **Database Row** is linked to a specific criteria, as selected by the **Search Criteria** dialog box.

A database row becomes a header, and each instance of a match on the drawing with that row's search criteria is listed in a subrow below it.

Each subrow in the database returns a value based on the formula in the header.

The **Set Criteria** command on a worksheet's **Row** menu opens the **Set Criteria** dialog box.

Any changes made with this command replace any previous settings for the row.

The **Edit Criteria** command on a worksheet's **Row** menu places the active **Database Row** in the **Formula Edit Bar** for editing.

The **Set Criteria** command on a worksheet's **Row** menu selects all objects on the drawing that meet the search criteria for the selected row.

The following types of data can be entered in a worksheet:

- Constant values
- Formulas

Constant values are typed directly in the **Formula Edit Bar**, which places the value in the cell. This value can be text, numbers, or dates. Constant values do not change unless you change them.

A formula is a sequence of functions, cell references and operators that produce a new value from existing values. A formula must always begin with an equal sign (=). Values produced as the result of a formula can be changed when the formula is evaluated.

For additional information, see the following topics:

Formula Edit Bar

Operators

Functions

Cell References

All entries are made to a worksheet cell by typing them in the **Formula Edit Bar**.

The formula edit bar becomes active when a cell is active. It will display the contents, if any, of the active cell.

The **Formula Edit Bar** has an **Enter** box and a **Cancel** box. After entering your formula or constant value, click the **Enter** box or press **Enter** or **Tab** to place the entry in the active cell.

Entering Numbers

To enter a number as a constant, select a cell and type in the number. To enter a negative number, precede it with a minus sign.

Do not format the number with commas or dollar signs as you enter it. (You can format the cell to display commas or dollar signs). If you enter a value containing commas or dollar signs, it will be treated as text.

Dates And Times

Dates and times are stored as a unique sequential numbers. In Windows, time is measured from January 1, 1900.

If you enter the date (e.g. 6/19/93) directly into an unformatted cell, MiniCad automatically formats it as a date cell and displays it correctly.

If you enter the date into a cell that is formatted with a different number format, it is displayed as a number. For example, if you enter the date January 1, 2000, into a cell already formatted as currency, the number is displayed as 35064.00, which is the number of days from January 1, 1904 to January 1, 2000.

Entering Text

In a MiniCad worksheet, text is any set of characters entered into a cell that is not a number, formula, date, or time.

To enter text as a constant, select the cell and type the text. As with any constant value, text entered into a cell remains until changed. Text can be entered into a cell as the result of a formula being executed. A cell can contain up to 255 characters of text.

Text can be any character, letter, number, or any other character the selected font can produce.

Text can be numbers, but to keep MiniCad from treating the cell as a number, enclose it in single quotation marks.

To enter a text string in a formula, enclose the text string in single quotation marks. You do not need to do this if the text string is a single word, but it is required if the text string has any spaces between the words. For example, notice the quotation marks in the following formula.

=IF(H9>100,yes,IF(H9>90,maybe,IF(H9>80,'I don't know' ,'no contract')))

If you omit the quotation marks, MiniCad displays a message saying "comma expected" because it cannot evaluate a formula with a space in it.

Entering Formulas

Formulas are used to produce calculated values within the worksheets.

Formulas can contain cell references, operators, values, or functions. Many formulas can be constructed from dialog boxes with just a few mouse clicks. Formulas also can be typed directly in the **Formula Edit Bar**.

A formula must begin with an equal sign (=). This tells MiniCad to evaluate this cell as a formula.

A formula will not evaluate properly if its syntax is not exactly correct. Parentheses must be in pairs, and commas are sometimes required even if an argument is not present. Refer to the **MiniCad Functions** reference for a list of the supported functions.

A formula can include references to another cell – even cells on other worksheets – in addition to constant values.

Formulas are not case-sensitive. You may enter function names and cell addresses in upper or lowercase.

For additional information, see the following topics:

Cell References

Operators

Functions

A cell reference identifies a cell or range of cells on a worksheet. They can be used to tell a formula where to find values needed to calculate the formula.

A references can refer to cells in another worksheet, however the worksheet must be in the active drawing. If it exists in another drawing, you must import it.

Although a referenced worksheet must be in the active drawing, it does not have to be open.

It is a good idea to put referenced cells and assumptions in a prominent place at the top of the worksheet. It is also a good idea to return the solutions to the worksheet formulas in the same area. This way you can easily change an assumption or value and immediately see the results (a "what if" analysis).

Cell references are based on the row and column headings on the worksheet.

References in MiniCad can be relative or absolute. An absolute reference refers to a specific cell referring to that cell's "address."

A relative reference tells MiniCad where to find a cell, "relative" to the cell that contains the formula. Relative references maintain the reference even if the cell is copied to another location on the worksheet.

An absolute reference is a reference to a specific location on the worksheet. An absolute reference breaks if the referenced cell is copied to a new location.

Internal references can be entered by clicking on the referenced cell while constructing the formula. You can also type the cell address directly into the formula.

External references must be typed in using `[worksheet name:cell/range address]` format.

External references are updated when the **Recalculate** command is chosen. The worksheet containing the external reference does not need to be open for the reference to be updated.

For additional information, see the following topics:

Formula Edit Bar

Operators

Functions

MiniCad supports use of the following operators in a formula. If the operator can be created with a special key combination, it is shown.

Arithmetic Operators

The arithmetic operators perform basic mathematical operations. They combine numeric values and produce numeric results.

Operation	Symbol
Addition	+
Subtraction	-
Multiplication	*
Division	/
Exponentiation	_ or **

Comparison Operators

The comparison operators compares two values and produces the logical value TRUE or FALSE.

Comparison	Symbol
Equal	=
Not Equal	<> or (**Control** =)*
Less than	<
Less than or equal to	<= or (**Control** <)*
Greater than	>
Greater than or equal to	>= or _ (**Control** >)*

Reference Operators

The reference operator combines two cell references into a single joint reference.

Reference	Symbol
Range	..

Reserved Words and Symbols

The following words and characters are reserved for use in formulas.

Symbol	Description
(	Left parenthesis
)	Right parenthesis
,	Comma
;	Semicolon
:	Colon
(Control v)	Square Root
Sum or _ (Control w)*	Summation

(Control p)	Pi
TRUE	True
FALSE	False

For additional information, see the following topics:

Formula Edit Bar

Functions

The following are the worksheet functions available in MiniCad.

Capitalized functions use search criteria. Lower case functions take a number value or a cell range. Results and arguments for trigonomic functions are expressed in radians.

Function	Description
acos	Returns the arc cosine of a number
Angle	Returns the angle of lines and walls
Area	Returns the total area of objects
asin	Returns the arc sine
atan	Returns the arc tangent
average	Returns the average (mean) of a series of arguments
BotBound	Returns the bottom boundary of an object
cos	Returns the cosine
Count	Returns the number of objects
deg2rad	Converts degrees to radians
exp	Exponential
Height	Returns the height of objects
if	Returns one value if logical test evaluates true, and another value if it evaluates false
int	Rounds a number down to nearest integer
LeftBound	Returns the left boundary of an object
Length	Returns the length of lines and walls
ln	Natural logarithm
log	Returns the base ten logarithm
max	Largest of a series of values
min	Smallest of a series of values
Perim	Returns the perimeter of objects
rad2deg	Converts radians to degrees
RightBound	Returns the right boundary
round	Rounds values
sin	Returns the sine
sqrt	Returns the square root
sum	Returns the total of a series of arguments
tan	Returns the tangent
TopBound	Returns the top boundary
Width	Returns the width of objects

Xcenter	Returns the center of objects in *x* direction
Ycenter	Returns the center of objects in *y* direction

acos(number)

Returns the arccosine of a number. The arccosine is the angle whose cosine is *number*. The returned angle is given in radians in the range 0 to π.

Number is the cosine of the angle, and must be from -1 to 1

acos(cos(x))=x

See also:

cos
List of MiniCad Functions

Angle(criteria)

Returns the angle of lines and walls, the span angle of arcs (in degrees), and the slope angle of slabs (in degrees).

See also:

Width
List of MiniCad Functions

Area(criteria)

Returns the total area of 2D objects.

See also:

Perim

List of MiniCad Functions

asin(number)

Returns the arcsine of a number. The arcsine is the angle whose sine is *number*. The returned angle is given in radians in the range $-\pi/2$ to $\pi/2$.

Number is the sine of the angle you want and must be from -1 to 1.

To express the arctangent in degrees use the rad2deg function (or multiply the result by 180/pi).

Related functions:

See also:

sin
List of MiniCad Functions

atan(number)

Returns the arctangent of a number. The arctangent is the angle whose tangent is *number*. The returned angle is given in radians in the range -π//2 to π//2.

Number is the tangent of the angle you want.

To express the arctangent in degrees, multiply the result by 180/pi().

See also:

tan
List of MiniCad Functions

average(number1,number2...)

Returns the average (mean) of the arguments.

See also:

max

min

sum

List of MiniCad Functions

BotBound(criteria)

Returns the minimum *y* coordinate of an object.

See also:

LeftBound
RightBound
TopBound
List of MiniCad Functions

cos(number)

Returns the cosine of a given angle.

Number is the angle in radians for which you want the cosine

See also:

List of MiniCad Functions

Count(criteria)

Returns the number of objects.

See also:

Width

List of MiniCad Functions

deg2rad(number)

Converts *number* from degrees to radians.

Number is the value in degrees you want converted to radians.

See also:

Width

List of MiniCad Functions

exp(number)

Returns e raised to the power of *number*. The constant e equals 2.71828182845904, the base of the natural logarithm.

Number is the exponent applied to the base e.

See also:

ln
List of MiniCad Functions

Height(criteria)

Returns the Δy (height) of objects.

See also:

Width

List of MiniCad Functions

if(logical_test, value_if_true,value_if_false)

Returns one value if *logical_test* is true, and another value if it is false.

Use this function to conduct conditional tests on values and formulas and to branch based on the results of that test. The outcome of the test determines the value returned by the if function.

The *logical_test* can be any value or expression that can be evaluated to true or false.

Up to seven if statements can be nested as *value_if_true,value_if_false* arguments to construct elaborate formulas. Boolean statements within an if statement must be in parentheses. Text within an if statement should be enclosed within quotation marks.

See also:

Width
List of MiniCad Functions

int(number)

Removes any fractional part of a number.

Number is the real number you want to round down to an integer.

See also:

round

List of MiniCad Functions

LeftBound(criteria)

Returns the left side minimum *x* (left boundary) of the object

See also:

BotBound
RightBound
TopBound

Length(criteria)

Returns the length of lines or walls.

See also:

Width

List of MiniCad Functions

ln (number)

Returns the natural logarithm (base e).

Number is the positive real number for which you want the logarithm

See also:

exp
List of MiniCad Functions

log(number)

Returns the base 10 logarithm of a number.

Number is the positive real number for which you want the logarithm.

See also:

ln
List of MiniCad Functions

max(number1,number2,...)

Returns the largest number in the list of arguments

Number1,number2,... are 1 to 14 numbers for which you want to find the maximum value.

See also:

min
List of MiniCad Functions

min(number1,number2,...)

Returns the smallest number in the list of arguments

Number1,number2,... are 1 to 14 numbers for which you want to find the maximum value.

See also:

max

List of MiniCad Functions

Perim(criteria)

Returns the perimeter of objects.

See also:

Width

List of MiniCad Functions

rad2deg(number)

Converts *number* from radians to degrees.

Number is the value in radians you want converted to degrees.

See also:

Width

List of MiniCad Functions

RightBound(criteria)

Returns the right side minimum *x* (right boundary) of the object.

See also:

BotBound
LeftBound
TopBound
List of MiniCad Functions

round(number,num_digits)

Rounds a number to the specified number of digits.

Number is the number you want to round

Num_digits specifies the number of digits to which you want to round *number*.

See also:

int
List of MiniCad Functions

Returns the sine of a given angle.

Number is the angle in radians for which you want the sine

See also:

List of MiniCad Functions

sqrt(number)

Returns a positive square root.

Number is the number for which you want the square root

See also:

Width

List of MiniCad Functions

sum(number1, number2,...)

Returns the sum of all the numbers in the list of arguments.

Number1,number2,... are 1 to 14 arguments for which you want the sum.

See also:

average
Count
max
min
List of MiniCad Functions

tan(number)

Returns the tangent of the given angle.

Number is the angle in radians for which you want the tangent

See also:

List of MiniCad Functions

Topbound(criteria)

Returns the maximum *y* (top boundary) of the object.

See also:

BotBound
LeftBound
RightBound
List of MiniCad Functions

Width(criteria)

Returns the Δx (width) of objects.

See also:

Height

List of MiniCad Functions

Xcenter(criteria)

Returns the center of objects in *x* direction.

See also:

Xcenter

List of MiniCad Functions

Ycenter(criteria)

Returns the center of objects in *y* direction.

See also:

Ycenter

List of MiniCad Functions

Before working on a worksheet, you must select the part of the worksheet with which you want to work. This may be a single cell, a range of cells, or characters in the edit bar.

To select...	**Do this...**
Single cells	Click on the cell
A range of cells	Drag through the range. Or click one corner of the range, then click the opposite corner while pressing the **Shift** key.
A row or rows	Click on the row heading (number). Multiple rows can be selected by dragging across the headings.
A column or columns	Click on the column heading (letter). Multiple columns can be selected by dragging across the headings.
The entire worksheet	Click on the square in the upper left corner of the worksheet

A selected cell is displayed with a thick border. A selected range of cells is darkened as well as surrounded by a thick border.

The **Reference Box** in the upper left corner of the worksheet displays the column and row location of the active cell.

The **Enter** key moves the active cell down. **Shift-Enter** moves the active cell up.

The **Tab** key moves the active cell to the right. **Shift-Tab** moves the active cell to the left.

You can move from cell to cell with the arrow keys. The active cell is highlighted. If a range of cells is selected, cell-to-cell movement is constrained to the selection.

For additional information, see the following topics:

Main Menus (worksheets)
Row Menus (worksheet window)
Entering Worksheet Data

Worksheets can be formatted in various ways. The text in worksheets is controlled using the commands on the **Text** menu, just like any text in MiniCad. However, because all pen color is black, color cannot be controlled.

The following commands on the worksheet's **Main** menu, let you perform additional formatting:

The **Number** command lets you format numbers in numerous ways, including decimals, dimensions, dates and Boolean calculations.

The **Alignment** command lets you control the alignment of a value in a cell.

The **Border** command lets you place a border around a cell.

The **Column Width** command. lets you specify the width of the column, in points.

For additional information, see the following topics:

Entering Worksheet Data
Selecting Cells & Moving Around the Worksheet
Main Menus (worksheets)
Row Menus (worksheets)

A MiniCad drawing can have a worksheet that contains both a spreadsheet and a database. In fact, the database in MiniCad is the drawing itself, and the data it contains is gathered using **Database Rows** in a worksheet.

When a **Database Row** is configured, it creates a subrow for each instance of the item referenced by its search criteria. Each subrow is part of a "parent" row, which expands and contracts automatically as the referenced data changes. A small diamond next to the row number shows that the row is a **Database Row**.

The information returned to a the subrows is contained in database records. Database records are created with the **Resource Palette**. Assignment of these records is done with the **Data** pane of the **Object Info Palette**, where these records can be referenced and edited.

The best way to show the usefulness of the database is by example. Please refer to the tutorials on database manipulation for more information.

For additional information, see the following topics:

Entering Worksheet Data
Selecting Cells & Moving Around the Worksheet
Main Menus (worksheets)
Row Menus (worksheets)

Worksheets can be placed as a graphic on the drawing. The worksheet does not have to be open for it to be placed as a graphic.

A worksheets placed as graphic initially appears as a box with an "X" through it. It will display normally when the worksheet is closed.

To place a worksheet as a graphic, click the **On Drawing** checkbox in the **Resource** palette before you open the worksheet.

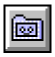
As a graphic object, the on-drawing worksheet can be selected and repositioned as needed. However, it cannot be resized.

There are two ways to resize a worksheet.

- You can use the **Insert** and **Delete** commands on the **Main** worksheet menu to add or delete columns and rows as needed.
- You can place the cursor on the lower right corner of the worksheet window and drag the corner left, right, up or down to add or remove rows and columns as you need.

Be aware that resizing a worksheet can affect values returned to referenced cells.

The **Resource Palette** is used to create and edit command palettes and macro commands.



Note: Check the status of the **Macro** button at the bottom of the palette. If the button has a large, red "X" across it, click it to allow macro commands to be listed in the palette.

There are two types of macros used in MiniCad:

- Internal MiniPascal macro commands, which run from customizable palettes.
- External macros, stored in external text files, which are run directly from the **Resource** palette.

The internal macros, also called commands, are created and edited from the **Resource** palette. Internal macro commands are contained in command palettes. Command palettes are created and edited from the **Resource** palette. Internal macro commands also can be made with the **Custom Selection**, **Custom Tools/Attributes**, **Custom Visibility** and **Save View** commands.

For additional information, see the following topics:

Creating a Command Palette
Importing a Command Palette
Opening a Command Palette
Creating a Macro Command
Editing a Macro Command
Command Editor
Running an External Macro

Command palettes are created by clicking the **New** button on the **Resource** palette. In the **Create Resource** dialog, select the **Command Palette** option.

When the **Assign Name** dialog opens, enter a palette name. Palette names must be unique. Click on the **OK** button to create the palette.

Like all palettes, **Command** palettes are floating windows that can be placed anywhere on the screen. A **Command** palette can be resized. A MiniCad file can have many command palettes, and palettes can hold many commands. This allows "sets" of command palettes to be created, and filled with different commands for different types of jobs.

Command Palettes can be imported from other drawing files. This is done by selecting the palette's name in the **Resource Palette's** list box and clicking the **Import** button. Use the pop-up menu at the top of the **Resource** palette to locate the file that contains the palette you want to use.

The palette will be copied into the active drawing. When a command palette is imported, all of the commands it contains are imported with it.

A Command Palette can be opened by selecting the palette's name in the **Resource Palette's** list box and clicking the **Open** button.

Internal macro commands can be created by clicking the **New** button on the **Resource** palette.

When the **Create Resource** palette opens, select the **MiniPascal Macro** button and click **Create**.

A dialog box with all of the file's command palettes appears. Select the palette on which you want the new command placed, then click the **Add** button.

A dialog box opens for the command's name. Type a unique name for the macro. This name must be different from any other object or resource name in the drawing. Click **OK**.

The command is listed in the **Resource** palette.

Select the command in the **Resource Palette's** list box and click the **Edit** button. The **Command Editor** is opened for you to begin creating the macro.

For additional information, see the following topics:

Command Editor

To edit an internal macro command, select the command palette's name in the **Resource Palette** list box and click **Open** button.

Once the palette has been opened, select the name of the macro command to be edited and click the **Edit** button.

The selected macro is presented in the **Command Editor** window.

tip: You can also edit a macro from a command palette. This is done by holding the **Control** key while you double-click its name in the palette. The macro will be opened in the **Command Editor**.

For additional information, see the following topics:

Command Editor

The **Command Editor** is a powerful tool for editing MiniCad macros. It lets you create macro statements by clicking in dialog boxes. For most users, the **Command Editor** is used to edit custom selections, custom tool/attributes, and custom visibility commands created by their respective commands on the **Organize** menu.

The **Command Editor** is automatically opened when a command macro is created or edited from the **Resource Palette**. See ***Creating a Macro Command*** and ***Editing a Macro Command*** for these procedures.

The **Command Editor** window contains the controls listed below.

Procedures

Criteria

Tools / Attr

Functions

Get Text

View Errors

The **Procedures** button in the **Command Editor** window opens the **Procedure** dialog box.

This box has two lists, which are linked. When a heading is selected in the scrolling list labeled **Chapter**, the available procedures appear in the adjacent **Procedure** scrolling list.

Procedures are selected from the list. When a procedure is selected, a brief description of the procedure is shown.

Note: A complete description of the available procedures is given in your *MiniPascal* manual.

The selected procedure is entered into the **Command Editor** when the **OK** button is clicked. All formatting and syntax is included, but variables and arguments have to be entered. While the **Command Editor** greatly assists you with this process, programming experience is helpful when creating macros.

The **Criteria** button is used to open the **Search Criteria** dialog box. By using this feature, a command can be created that searches across the drawing for every instance of any of the criteria you specify.

The **Search Criteria** dialog box has the following options:

Preset/Edit

Clicking the **Preset** button allows creation of the search criteria by clicking any of the individual criteria buttons on the dialog box. Clicking the **Edit** button allows the search criteria to be edited.

Visible

When selected, the search is limited to visible objects.

Selected

When selected, the search is limited to selected objects.

All

When selected all objects in the drawing are subject to the search.

Layer

A search can be limited to a particular layer or layers. To set the layer criteria, click the **Layer** button.

When clicked, a list of all layers in the drawing is displayed. Select any layers to be included in the search. Multiple layers can be selected by pressing the **Shift** key while selecting.

Click the **Add** button to add selected layers to the search criteria. The **Remove** button deletes the selected layer from the search criteria. The layer itself is not deleted. When finished, click the **Done** button.

Cancel returns to the **Command Editor** without any changes.

Class

A search can be limited to include instances of a particular class or classes. This is done by clicking the **Class** button.

When clicked, a list of all classes in the drawing is displayed. Select any classes to be included in the search. Multiple classes can be selected by pressing the **Shift** key while selecting. Click the **Add** button to add the class or classes to the search criteria. The **Remove** button clears the selected classes from the search criteria. The class itself is not deleted. When finished, click the **Done** button.

Cancel returns to the **Command Editor** without any changes.

Type

A search can be made for any of the object types used in MiniCad. To search for an object type, click on the **Type** button on the **Search Criteria** dialog box.

When clicked, a list of object types is displayed. Select any of the object types to be included in the search. Multiple types can be selected by pressing the **Shift** key while selecting. Click the **Add** button to add the object types to the search criteria. The **Remove** button clears the selected object types from the search criteria. When finished, click the **Done** button.

Cancel returns to the **Command Editor** without any changes.

Name

A drawing can be searched for named objects. To include a named object in a search criteria, click **Name**.

When clicked, a list of all named objects in the drawing is displayed. Select any of the object names to be included in the search. Multiple object names can be selected by pressing the **Shift** key while selecting. Click the **Add** button to add the object names to the search criteria. The **Remove** button clears the selected object names from the search criteria. When finished, click the **Done** button.

Cancel returns to the **Command Editor** without any changes.

Symbol

MiniCad can search for instances of symbols placed on the drawing. To include a symbol in the search criteria, click on the **Symbol** button. When clicked, a list of symbols is displayed. Select any symbol to be included in the search. Symbols can be located in folders nested within other folders. When finished, click the **Done** button to add the symbols to the search criteria.

Cancel returns to the **Command Editor** without any changes.

Record/Field

The search criteria can search the drawing for instances of an object or symbol attached to a specific record or fields within the record. In addition, certain operations can be performed on the record to further refine the search. To search for a particular record or field, click the **Rec/Fld** button. Select the records and/or fields to be searched for by the program, and double-click.

You may specify a limit by selecting one of the following constraints in the **Select an Operator** list.

- When the **No operators** option is specified, no limit is placed on the search criteria.
- When the **Less than (<)** option is specified, the search is successful only if the search criteria is less than the value in the **Enter a Limit Value** field.
- When the **Greater than (>)** option is specified, the search is successful only if the search criteria is greater than the value in the **Enter a Limit Value** field.
- When the **Equal (=)** option is specified, the search is successful only if the search criteria is equal to the value in the **Enter a Limit Value** field.
- When the **Less than or equal (<=)** option is specified, the search is successful only if the search criteria is less than or equal to the value in the **Enter a Limit Value** field.
- When the **Greater than or equal (=>)** option is specified, the search is successful only if the search criteria is greater than or equal to the value in the **Enter a Limit Value** field.
- When the **Not equal (<>)** option is specified, the search is successful only if the search criteria is not equal to the value in the **Enter a Limit Value** field.

If you want the search to be limited to whole record, without individual fields, check the **Record Search Only** checkbox.

After making your selections, click the **Done** button. Clicking the **Delete** button removes selected record or fields within the record from the selected list. It is not going to remove the specific record or fields within the record from the drawing.

Fill Pattern

The search can be limited to instances of a specific fill pattern. To search by fill pattern, click the **FilPat** button. Select the fill patterns to be searched for by the program, and double-click. After making all selections, click the **Done** button. **Delete** removes a fill pattern from the selected list.

Edited fill patterns retain the ID number of the original fill pattern, so the search criteria remains the same no matter how many times the pattern is edited.

Line Weight

MiniCad can limit the search to instances of a specific line thickness. To search by line weight, click the **LineWgt** button. Select the line weights to be searched for by the program and double-click. After making all selections, click the **Done** button. **Delete** removes a line weight from the selected list.

It is possible to have more line weights stored in the drawing than the five selections available on the attribute palette. If more than five different line weights have been created in the drawing, click on **Set Thickness** in the selection box. The line weight is entered directly in the dialog box.

Line Style

MiniCad can limit the search to instances of a specific dash style. To do this, click the **LineStyl** button. The selection list displays all available dash styles. Select the dash styles for which you want to conduct a search, and double-click. After making all selections, click the **Done** button. Delete removes a dash style from the search criteria list. Clicking **Cancel** returns to the **Command Editor**.

Arrow Shape

The search can be limited to instances of specific arrowheads and line markers. To search by line marker, click the **Arw Shape** button. Select the arrowheads and line markers for which you want to conduct a search, and double-click. When all selections are made, click the **Done** button. **Delete** removes an arrowhead or line marker from the search criteria list.

Arrow Size

The search can include instances of specific arrowheads sizes. Click the **Arw Size** button. A scrolling list of the markers on the drawing is displayed. Select the particular **arrow size** for which a search is to be conducted and double-click. After making the selections, click the **Done** button. Click on the Delete button to remove a selected arrowhead or line marker from the list.

Fill / Pen Foreground / Background

The program can limit the search to instances of a specific color. MiniCad isolates the foreground color from the background color and also separates the pen (lines and text) and fill (surfaces). To search by color, click one of the color buttons (**FillFore**, **FillBack**, **PenFore**, **PenBack**). The color palette appears.

Select the colors for which you want to conduct a search, and double-click. After making all selections, click the **Done** button. **Delete** removes a color from the selected list.

The **Tools / Attr** button opens the **Custom Attributes** dialog box, which allows a combination of object attributes to be applied as part of a command. Before selecting **Tools / Attr**, select or create an object with the attributes to be included in the command. Specify color and font and any other attribute to be added to the command.

The attributes can be toggled on and off as a group by clicking the appropriate buttons. The individual attributes are:

Pen
The pen color controls how a line's color and pattern are drawn. To select all, click **Pen**. There is individual control over foreground color, background color, and pattern

Fill
The fill color controls how the surface's color and pattern are drawn. To select all, click **Fill**. Foreground color, background color, and pattern can be used as individual attributes.

Line
The line attributes can be toggled on and off as a group by clicking Line. Line's weight, style (dash pattern), markers (arrowheads), and marker size can be used as individual attributes.

Text
The text attributes can be toggled on and off as a group by clicking **Text**. Text font, size, style, spacing, and justification can be used as individual attributes.

Class
The object's **Class** can be used as an attribute.

Layer
The object's **Layer** can be used as an attribute.

Symbols
If the object is a symbol, its name can be used as an attribute.

Tool
The selected tool can be used to make a custom tool.

Constraint
The current snap constraint can be selected as an attribute.

The **Functions** button is used to add MiniCad functions to a macro command.

For a list of the supported functions, see ***MiniCad Functions***.

The **Get Text** button is used to import a text file into the **Command Editor**. This can be used if you have created your macro code using an external ASCII editor rather than the **Command Editor**.

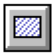
To enter text files, click the **Get Text** button, select the text file from the directory dialog box and click **Open**. The contents of the file are loaded into the **Command Editor**.

The **View Errors** button is used to inspect the errors are generated when a MiniPascal command is executed.

Internal macros are executed from the command palette to which they were assigned.

External macros, which are stored in standard text files, are executed from the **Resource** palette. When a text file is selected in the **Resource** palette, the top button on the palette becomes the **Run** button. To run the macro, simply select its name in the **Resource Palette**'s list box and click the **Run** button.

Be aware that any text file displays in the palette, even if it is not a macro. If the **Run** button is clicked on a non-macro file, an error message is displayed.

The **Resource Palette** is used to create, edit and import hatch patterns.


Note: Check the status of the **Hatch** button at the bottom of the palette. If the button has a large, red "X" across it, click it to allow hatch patterns to be listed in the palette.

Hatch patterns are patterns of lines that are used to fill a closed object. They are typically used to represent various construction materials. They can also be used as "section lines," which identify an object's profile when it is cut.

Hatch patterns differ from fill patterns in that:

- They can be scaled. Fill patterns cannot.
- They can be exported with DXF. Fill patterns cannot.

Hatches are made of line segments. Each line segment has a starting point, a variable offset (i.e., the distance between the line segments), a repeat factor (i.e., how often the pattern is duplicated), and a dash factor (or how much space there is between line segments).

Segments are in layers, with the first segment at the bottom and each new one atop the last. There can be up to thirty segments or layers in a hatch pattern. This allows elaborate hatch patterns to be created.

Hatch patterns are either in page scale, where the measurements shown are used no matter what the scale of the active layer, or in world scale, where hatch measurements are scaled to match the active layer. In page scale, a quarter inch spacing in a hatch pattern is a quarter-inch spacing on the screen when in normal scale. This scaling ability is another way hatch patterns differ from patterns.

A hatch pattern is placed in the drawing using the **Hatch** command.

For additional information, see the following topics:

Importing a Hatch Pattern
Editing a Hatch Pattern

If a hatch pattern exists in another drawing or in a MiniCad library, it can be imported into the active drawing.

To import a hatch pattern, select its name in the **Resource Palette**'s list box, then click the **Import** button. The pattern will be imported into the active drawing.

Note: The **Find** button is can be used to locate a pattern anywhere on your system. When clicked, the **Find** button opens the **Find Resource** dialog box. Enter the name of the pattern you want to locate, use the **Set Where** button to set the search path and click **OK**.

If the imported hatch pattern's name already exists in your drawing, you will be asked whether you want it to replace the one in your drawing.

To edit a hatch pattern, select its name in the **Resource Palette**'s list box, then click **Edit** button.

The hatch pattern will be presented in the **Edit Hatch** dialog box. This dialog contains the tools you need to make changes to the hatch pattern. After you have made your changes, click **OK** to close the dialog box.

Like all edits to a file, changes in the active hatch do not become be permanent until the active document is saved (i.e., if you create a pattern, but don't save the drawing, the pattern will be lost).

For more information about the **Edit Hatch** window, see ***The Edit Hatch dialog box***.

The **Edit Hatch** dialog box is used to edit a hatch pattern. This window is opened by double-clicking the hatch pattern's name in the **Resource Palette**.

Along the left side of the **Edit Hatch** dialog are a set of editing tools. Along the right side of the dialog box are the hatch values. You can define a hatch pattern by entering new values in these fields, or by drawing it with the mouse in the edit window.

The **Edit Hatch** dialog contains the following fields and controls:

Start Point

The **Start Point** group box contains the coordinates of the starting point handle, which represents the pattern's origin. The origin is used when the hatch is applied on the drawing. It represents the point in the pattern that will be aligned on the cursor's position when the pattern is placed in the drawing.

The buttons on the right side of **Start Point** group box can be used to toggle the values between Cartesian and polar format.

In the edit window, the **Start Point** is represented by a solid round dot. You can change this dot's position by entering new values in the **Start Point** field, or simply by dragging the dot to a new position.

Note: If you hold the **Control** key and drag the **Start Point** handle, the pattern will be copied to a new layer.

Repeat

The **Repeat** group box specifies the angle and length of each hatch layer before the segment begins again. There may be a portion of the layer that is open space, determined by the dash factor, or the segment can be drawn continuously.

A value of 0 in either coordinate field results in a hatch layer that is either horizontal or vertical. Equal absolute values result in the hatch layer lying on a 45° line.

The buttons on the right side of **Repeat** group box can be used to toggle the values between Cartesian and polar format.

You can also change the **Repeat** value by dragging the small, unfilled, square handle in the edit window. Dragging this handle will change the value in the **Repeat** fields.

The line defined by the **Repeat** and **Start Point** values can never be parallel to the line defined by the **Offset** point and the **Start Point** values.

Offset

The **Offset** group box defines one edge or corner of the repeating tile that the hatch layer is in. Each layer can have a different repeat tile. Equal absolute values entered into the coordinate fields result in the tile edge or corner lying on a 45° line from the start point.

The buttons on the right side of **Offset** group box can be used to toggle the values between Cartesian and polar format.

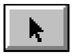
You can also change the **Offset** value by dragging the small, unfilled, round handle in the edit window. Dragging this handle will change the value in the **Offset** fields.

The line defined by **Offset** and **Start Point** can never be parallel to the line defined by the **Repeat** and **Start Point**.

Dash Factor

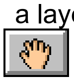
The **Dash Factor** determines the line-to-space ratio. To produce a solid line, enter a value of 1 in this field. A value less than one produces a dashed line..

You can also change the **Dash Factor** by dragging the small, solid, square handle in the edit window. Dragging this handle will change the value in the **Dash Factor** field.



The **Select** tool is used to select a layer in the edit window. When editing hatches with more than one layer, be sure to select the proper layer before entering values.

Layers are selected by clicking on one of the elements on the desired layers. The solid cursor shows as a hollow cursor when it is over a selectable object . Handles display on the objects in the selected layer. You may want to use this dialog's **Zoom In** tool to magnify the hatch pattern and make it easier to select a layer.



The **Pan Tool** is used to pan the edit window. Panning repositions the hatch origin.

The **Zoom In** tool is used to magnify objects in the edit window. Click the tool to magnify the contents of the window by a factor of 2.

The **Zoom Out** tool is used to reduce the displayed size of objects in the edit window. Click the tool to reduce the displayed size by a factor of two.

Add Layer

The **Add Layer** button is used to add a layer to the edit window. A hatch pattern can have up to thirty layers. The new layer is created with the default hatch values.

You can alternatively, press the **Control** key and drag the current layer's start point handle to add a new layer to the pattern.

Delete Layer

The **Delete Layer** button is used to remove the active layer in the hatch. This action cannot be undone.

Page

The **Page** button sets the pattern's scale to absolute dimensions. When this option is set, the hatch maintains its scale, regardless of the scale of the layer in which it is placed.

Page scale is useful when you want an absolute distance between hatch lines on printed output.

World

World allows the pattern to be scaled to that of the layer in which it is placed. This setting is useful for hatches that represent materials such as brick elevations or floor tiles that are used on objects drawn to scale.

Name

This field specifies the hatch pattern's name. Each hatch must have a unique name. The maximum length for a hatch name is twenty characters.

Revert

The **Revert** button returns the active hatch to its original condition, without any changes.

Cancel

The **Cancel** button closes the **Edit Hatch** dialog box without making any changes.

OK

Accepts the changes and closes the dialog box. Like all edits to a file, changes in the active hatch do not become be permanent until the active document is saved (i.e., if you save the pattern, but don't save the drawing, the pattern will be lost).

The **Resource Palette** is used to create, edit and import record formats.

Note: Check the status of the **Record Format** button at the bottom of the palette. If the button has a large, red "X" across it, click it to allow record formats to be listed in the palette.

Record formats are record descriptions that are used to attach database records to objects in a drawing. When a record format is attached to an object, it creates a database record (you can view or change this record by selecting the object and displaying the **Data** pane of the **Object Info Palette**).

Database records can be collected into a worksheet, where they can be viewed, sorted, analyzed and printed. You can even extract records that meet specific criteria (e.g., a certain manufacturer or a > 90 day lead time). See the **Worksheets** topic for more information about worksheets.

One powerful use of a record format is to attach it to a symbol. For instance, you might create a record format called "Appliances" that contains individual fields for manufacturer, type, model number, availability, price, etc. This record format can then be attached to different appliance symbols in the library, modifying it appropriately for each symbol.

Each time an appliance symbol is placed in a drawing, an "Appliance" database record is automatically created. You can easily pull these "Appliance" records into a worksheet to see how many appliances (and what type) are needed.

For additional information, see the following topics:

Creating a Record Format

Editing a Record Format

Importing a Record Format

Before a record can be assigned to an object or symbol, a record format must be created for it. The record format defines the fields that make up the record, and defines the default value of each.

Record formats are created by clicking the **New** button in the **Resource** palette.

When the **Create Resource** dialog box appears, select the **Record Format** button and click **Create**.

In the **Create Record** dialog box, enter a name for the record format. This name must be unique, and must not exceed 20 characters.

In the **Record Format** dialog box, click **New** to create the first field in the record.

In the **Edit Field** dialog box, enter the field's name, data type and default value, then click **OK**.

The **Edit Record** dialog closes and you are returned to the **Record Format** dialog. The field you just created will appear in this dialog's scrolling list.

Define each of the record's fields in the same way. A record format can contain many fields (with a limit of 32k).

When all fields have been defined, click **OK**.

For related information, see the following topics:

Editing a Record Format
Importing a Record Format

The **Edit Field** dialog box is used to create or edit a field in a record format. It contains the following options:

Name

The **Name** field specifies the name of the field. This name does not need to be unique.

Type

The **Type** field specifies the kind of information the field will hold. Select one of the four types:

- **Integer.** Integer fields contain whole numbers. Because an integer is a whole number, it can be very efficiently stored by the computer, and uses less memory than other numeric types. If the field will only hold whole numbers, select the **Integer** type instead of **Number**.
- **Number.** Number fields can contain any number, including fractional values.
- **Boolean.** Boolean fields can return a descriptive text string of up to eight characters. In the record format, a Boolean field returns either true or false.
- **Text** Text fields can contain up to 32,000 characters of information, but only 255 characters of that text can be displayed in the spreadsheet cell. A database subrow will display sixty-three characters of the text in a **Text** field.

Default Values

The **Default Value** specifies the data that will automatically be placed in this field when the record is attached to an object in the drawing. Once the record is attached to an object, you can use the **Data** pane of the **Object Info Palette** to change the information in that record.

Record formats that have already been created can be modified by selecting the name of the record format in the **Resource Palette**'s list box and clicking the **Edit** button (you can also double-click the record format's name). The **Record Format** dialog box is opened.

- If you want to change a field's name, type or default value, select the field and click the **Edit** button.
- If you want to add a new field, click the **New** button and enter the field's name, type and default value in the **Edit Field** dialog box. For more information about this dialog box, see **The Edit Field dialog box**.
- If you want to move a field's position in the record, select it and click the **Up** or **Down** button to move it one line up or down.
- If you want to delete a field, select it and click the **Remove** button.

For related information, see the following topics:

Creating a Record Format
Importing a Record Format

If a record format exists in another drawing or in a MiniCad library, it can be imported into the active drawing.

To do this, select the name of the record format in the **Resource Palette**'s list box. Then click the **Import** button. The record format will be imported into the active drawing.

Note: The **Find** button is can be used to locate a record format anywhere on your system. When clicked, the **Find** button opens the **Find Resource** dialog box. Enter the name of the record format you want to locate, use the **Set Where** button to set the search path and click **OK**.

If the imported record format's name already exists in your drawing, you will be asked whether you want it to replace the one in your drawing.

For related information, see the following topics:

Creating a Record Format

Editing a Record Format