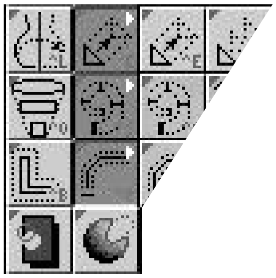

The 3-D operations tools

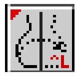
The 3-D Operations Tools

The next section isabout the bottom eight tools in the Tool Palette. These tools take existing geometry and generate new shapes based on the original objects. Most frequently, you will take 2-D sections or contours and generate 3-D shapes from them.

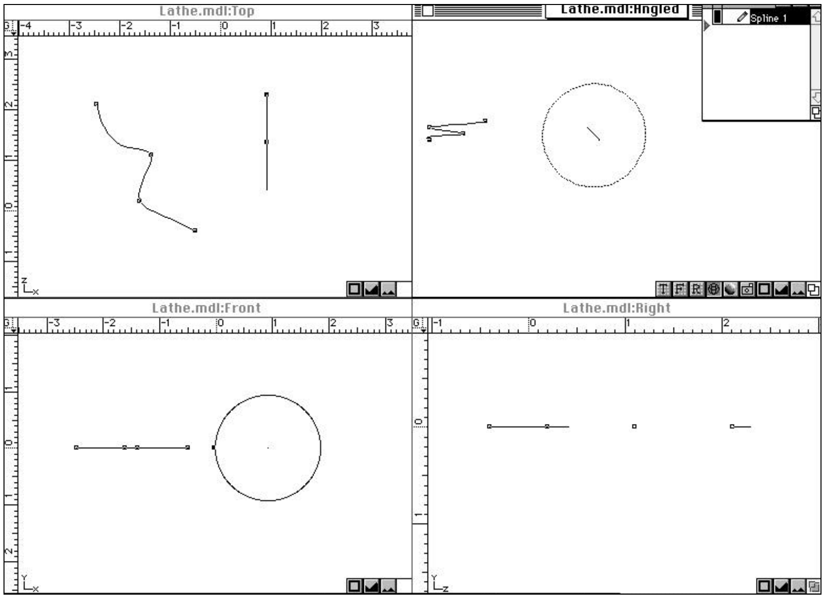
After the surface object is created, the original 2-D objects are normally saved separately and “turned off” (unless you specify to have them deleted, in Edit/Preferences/General).

All of these tools have a number of settings that can be made in the Tool Info palette to control and specify what the finished shape should look like. When you first pick one of these Operation tools, the Tool Info palette comes up with the default settings for that tool. As you make changes to the fields and buttons there, those changes get preserved so that if you leave the tool and then later come back to it, the same settings you were using before are still there. But sometimes this is not what you want. You often want to go back to a “default” group of settings.

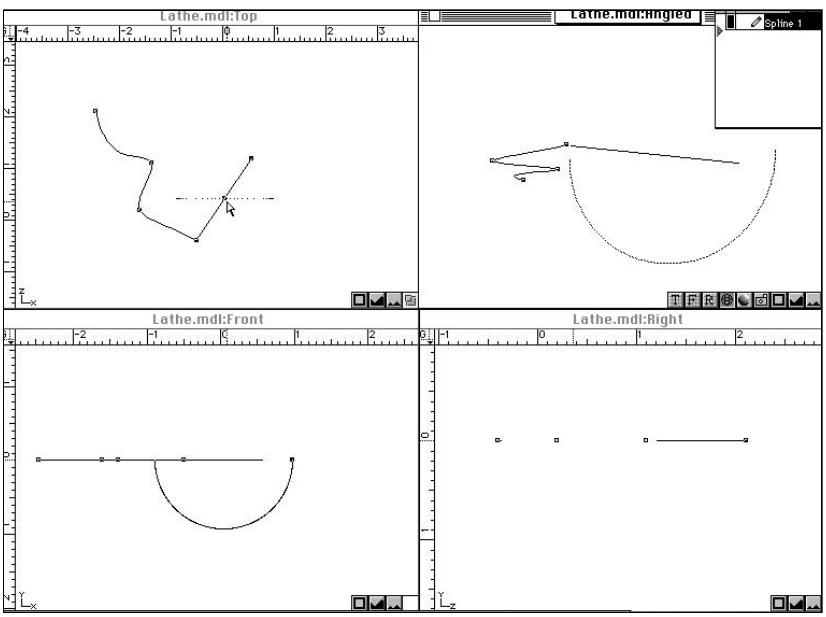
To do this, simply hold the Option key when you select the tool. This will “reset” the fields and buttons in the Tool Info palette for that tool.



The Lathe Tool



Select Lathe



Position Lathe Axis Select Arc and Banking Angle.

The Lathe Tool creates a new surfaced object by taking the currently selected 2-D objects and rotating them about a user-defined axis, resulting in a surface that is defined by the “sweep” of the selected objects. Only planar objects can be Lathed.

If the original selected objects are “filled” (the small ellipse in the Draw-Options portion of the Tool palette was filled in with grey when the objects were created), then the resulting lathed surface object will have its ends “capped.” If the original selected objects are “open” (the small ellipse was white), then the resulting lathed surface object will have ends that are open, or un-capped. For example, if you draw a polygon that is “filled” (it is a solid object and will render as a surface), and if you then lathe that polygon through a 90° arc, the beginning and ending faces of the new surface object will be solid, resulting in a lathed shape that is capped. But if you draw that same polygon as “open,” then the beginning and ending faces of the new surface object will also be open, resulting in a lathed shape that is like an open curved tunnel.

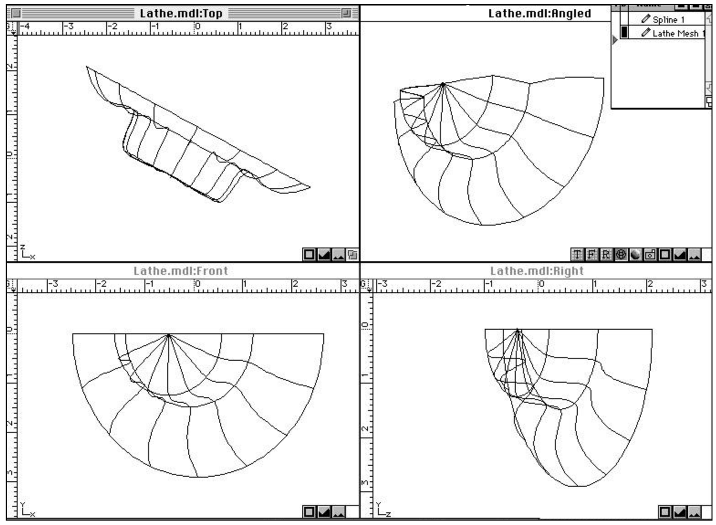
Using the Lathe tool

1) Select the Object(s) that you want to lathe with the Selector tool.
2) Pick the Lathe tool from the Tool palette (or press Shift-L).

You may reverse step 1 and 2.

Click on the Lathe Tool; a Lathe Axis and a Lathe Circle appear. The Lathe Axis will always appear exactly in the plane of the 2-D object, and the Lathe Circle will appear perpendicular to that Lathe Axis.

3) Make modifications to the data fields shown in the Tool Info palette by either typing in new values, or by dragging control points of the Lathe Axis and Lathe Circle.
4) When everything is as you want, hit the “Lathe” button in the Tool Info palette, or double-click in any View window.



Perform Lathe

The 2-D shape will be “rotated” around the Lathe axis and a new surfaced object will be created along that sweep. The original 2-D objects are saved separately and “turned-off” in the Group palette. They can be restored at anytime.

Controlling the Lathe tool with the Tool Info Palette

Center (X,Y,Z)

Displays the location of the intersection of the Lathe Axis and the Lathe Circle.

Enter values from the keyboard for X, Y, and Z, or drag the control point at the intersection of the Lathe Axis and Lathe Circle to set the center point graphically. Only two of these values can be modified at any one time. Modifying the third value would move the Lathe Axis out of the plane of the selected 2-D object, and that is not allowed.

° of Arc

Controls the amount of rotation through which the selected object(s) is taken. Enter a value between 0° and 360°. Or, drag the control handle on the Lathe Circle to graphically set the angle.

Segments

Defines how many copies of the original 2-D shape get distributed through the final object as “ribs” to support the surface.

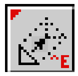
The higher the value you enter here, the more complex your final object will become. Simpler objects will save file size, but will also perhaps give up some modeling flexibility.

Bank

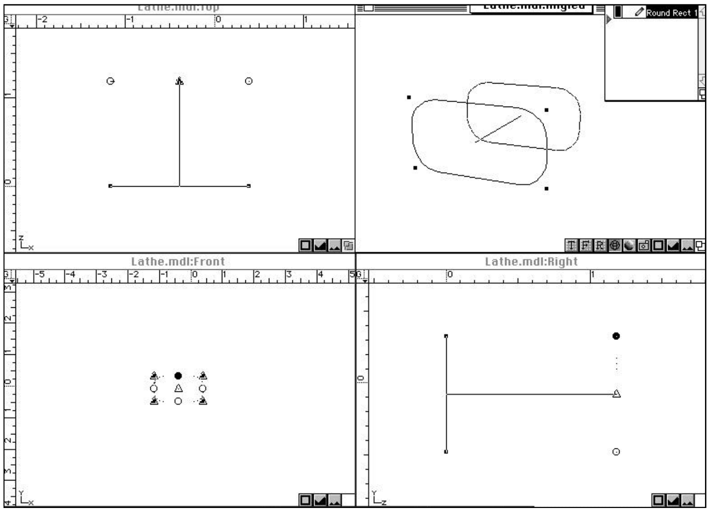
Controls the angle of the Lathe Axis as it is oriented in the plane of the selected 2-D object(s).

Enter a value between 0° and 360°. Or, drag the control point at the end of the Lathe Axis to set the Bank angle graphically. Notice as you go beyond 45°, the Lathe Circle switches from one view to another.

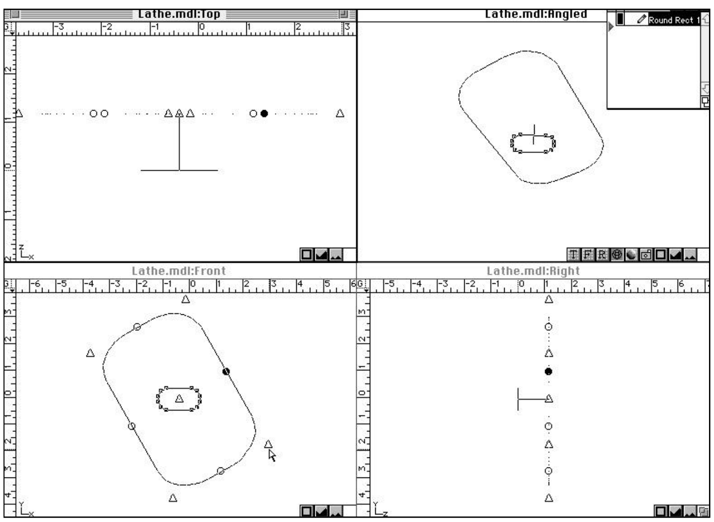
Lathe Tool Info palette



The Extrude Tool



Select Extrude



Position angle of extrusion and distance, rotate object.

The Extrude Tool creates a new surfaced object by taking the currently selected objects and projecting them away from the plane they lie in, usually in a perpendicular direction, resulting in a surface that is defined by the “extrusion lines” of the selected objects. Only objects made up of lines, arcs, ellipses and splines can be Extruded.

The objects can also be resized and rotated at the end of the extrusion, resulting in a shape that is transformed along the length of the extrusion.

If the original selected objects are “filled” (the small ellipse in the Draw-Options portion of the Tool palette was filled in with grey when the objects were created), then the resulting extruded surface object will have its ends “capped.” If the original selected objects are “open” (the small ellipse was white), then the resulting extruded surface object will have ends that are open, or un-capped.

Using the Extrude tool

1) Select the Object(s) that you want to extrude with the Selector tool. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Extrude tool from the Tool palette. (Or press Shift-E on the keyboard.)

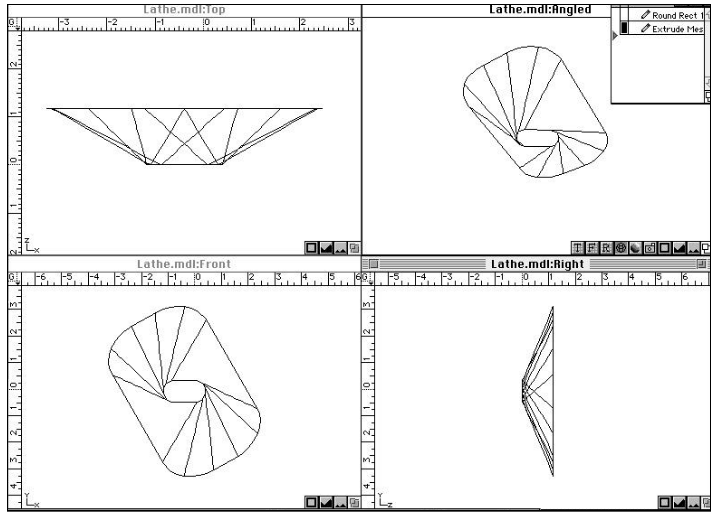
You may reverse step 1 and 2.

As soon as you click on the Extrude tool, a default extrusion line appears, as well as copy of the selected object at the end of the extrusion line which is enclosed by a bounding rectangle.

The **corner points** of the bounding rectangle are marked by small triangles. These triangles are used for resizing the bounding rectangle, resulting in a reshaped and resized object at the end of the extrusion.

The **midpoints** of the sides of the bounding rectangle are marked by small circles. These circles are used to rotate the bounding rectangle, resulting in a rotated copy of the object at the end of the extrusion. (Note: The copy of the object at the end of the extrusion line cannot be rotated out of the plane it lies in. In other words, the objects at both ends of the extrusion line will always lie in parallel planes.)

The **center point** of the bounding rectangle is marked by a small triangle. It is used to relocate the ending point of the extrusion line.



Perform extrusion

3) Make modifications to the data fields shown in the Tool Info palette by either typing in new values or by dragging control points of the bounding rectangle at the end of the extrusion line.
4) When everything is as you want, hit the “Extrude” button in the Tool Info palette, or double-click in any View window.

The selected shape(s) will be “extruded” along the extrusion line and transformed in a linear fashion according to the ending bounding rectangle. A new surfaced object will be created along that extrusion. The original objects are saved separately and “turned-off” in the Group palette. They can be restored at anytime.

Controlling the Extrude tool with the Tool Info Palette

ΔX, Δ Y, ΔZ

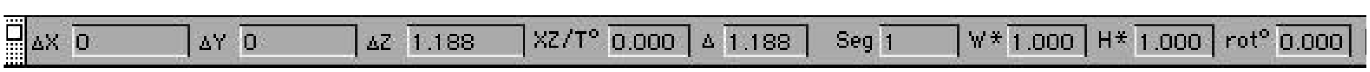
Controls the displacement of the endpoint of the extrusion line relative to its beginning point.

Enter values from the keyboard for ΔX, ΔY, and ΔZ, or drag the small triangular control point at the center of the bounding rectangle to set the center point graphically.

Angle

Controls the angle of the extrusion line.

Enter a value between 0° and 360°. Or, drag the small triangular control point at the center of the bounding rectangle to graphically set the angle.



Extrude Tool Info palette

Δ

Controls the length of the extrusion line.

Enter a value for the length of extrusion you want. Or, drag the small triangular control point at the center of the bounding rectangle to graphically set the length.

Segments

Defines how many copies of the original 2-D shape get distributed through the final object as “ribs” to support the surface.

The higher the value you enter here, the more complex your final object will become. Simpler objects will save file size, but will also perhaps give up some modeling flexibility.

W*, H*

Defines the percentage in change of width and height of the object(s) from the beginning to the end of the extrusion, thereby resizing and reproportioning the object(s) along the way.

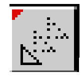
Enter a value to indicate the ratio of the ending size compared to the beginning size. Values less than 1 shrink the object. Values larger than 1 expand the object. Or, drag the small triangular control points at the corners of the bounding rectangle to graphically change the width or height.

Rot °

Defines the rotation of the copy of the object at the end of the extrusion line compared to the orientation of the object at the beginning.

Enter a value between 0° and 360°. Or, drag the small circular control points on the sides of the bounding rectangle to graphically set the rotation angle.

Linear Duplicate Tool



Note: The Linear Duplicate tool is on the Extrude tool pop-up.

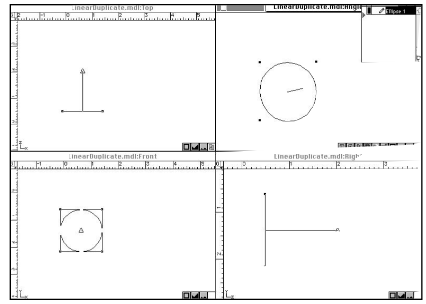
The Linear Duplicate tool takes the currently selected 2-D or 3-D objects and distributes a user-defined number of copies of those objects along a user-defined vector. The copies will maintain the same rotational orientation as the original.

The Linear Duplicate tool has an interesting optional behavior: It allows you to do simple planar polar duplication.

Using the Linear Duplicate tool

1) Select the Object(s) that you want to duplicate with the Selector tool. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Linear Duplicate tool from the Tool palette.

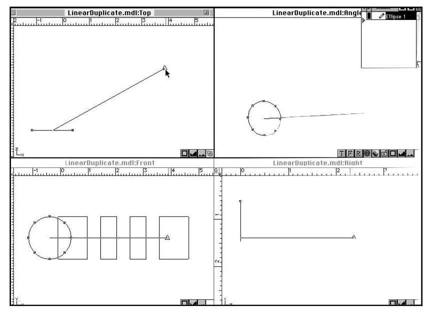
You can actually reverse steps 1 and 2. You don't have to have something selected before you pick the Linear Duplicate tool.



Select Linear Duplicate

As soon as you click on the Linear Duplicate tool, a default duplication vector appears, as well as copy of the selected object at the end of the vector.

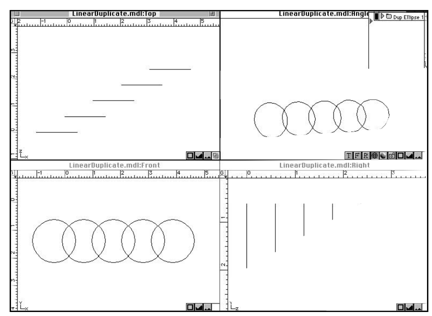
3) Set the number of copies you want by typing a value into the "Total # of copies" data field. That number of copies will appear in the preview.
4) Drag the small triangular control point at the end of the duplication vector to set it graphically.
5) When the vector is as you want, hit the "Duplicate" button in the Tool Info palette.



Enter 5 copies, change angle, and distance of duplicate.

Another way to activate the Duplicate operation is to double-click in any View window.

The selected objects will be copied and distributed along the duplication vector.



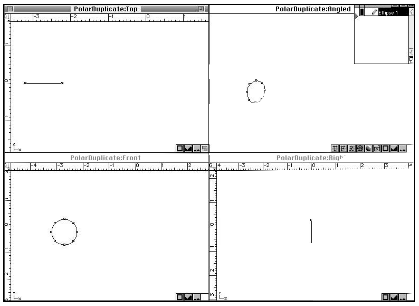
Perform linear duplication

Using the Linear Duplicate tool to do 2-D Polar Duplication

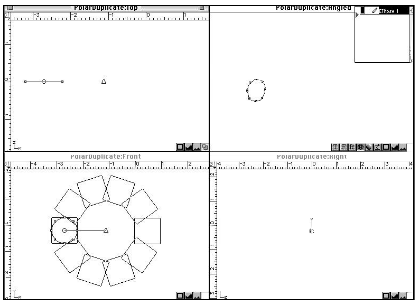
1) Select the Object(s) that you want to duplicate with the Selector tool. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Linear Duplicate tool from the Tool palette.

You can actually reverse steps 1 and 2. You don't have to have something selected before you pick the Linear Duplicate tool.

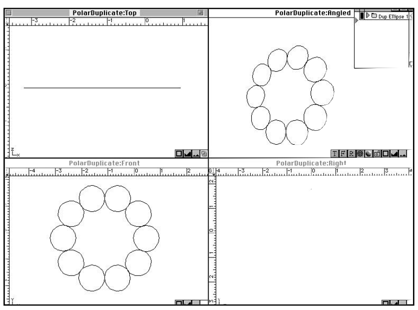
Linear Duplicate Tool Info palette



Select Linear Duplicate



Put X in Polar box, enter 10 copies, and move center handle to right in Front View



Perform polar duplication.

3) Click on the “Polar” check box to put an “X” in it. This switches you from the Linear Duplicate mode to the Polar Duplicate mode.
4) Set the number of copies you want by typing a value into the “Total # of copies” data field. That number of copies will appear in the preview.
5) Click on one of the radio buttons for X, Y, or Z to indicate which axis you want the rotation to occur.

As soon as you click on the “Polar” check box, two small triangular handle points appear, one each on the two ends of a rotation vector. One of the triangles is the “center” handle. It marks the center point about which the polar duplication will occur. By default, this handle appears at the Depth Origin. The other is the “angle” handle. It defines the amount of angular displacement that your copies go through. It is attached to the center of the selected object(s).

6) Go to the orthogonal view window that is perpendicular to your chosen axis of rotation and drag either of the two small triangular control points at the ends of the rotation vector to set them graphically. You will see a preview of the results of your actions in that window.

Note: You can only create a rotation in one of the view windows, depending on what you have chosen for you axis of rotation. For example, if you pick the “X” axis radio button in the Tool Info palette, then you can only work in the Right view window, since that window has the “X” axis coming straight out of it. Any changes to the small triangular handles in the Front or Top view windows will have no effect.

7) When the vector is as you want, hit the “Duplicate” button in the Tool Info palette.

Another way to activate the Duplicate operation, is to double-click in any View window.

The selected objects will be copied and distributed around the rotation point.

The Loft Tool

The Loft tool allows you to take a series of separate 2-D shapes, that are set out in 3-D space, and use them as “ribs” to support a “skin” that you stretch across from one 2-D shape to the next. This is much like the ribs of an airplane wing that get wrapped by the skin of the wing itself.

The curves that are used as the “ribs” must be planar. They do not all need to be parallel to each other in 3-D space. They can be rotated in any manner relative to each other.

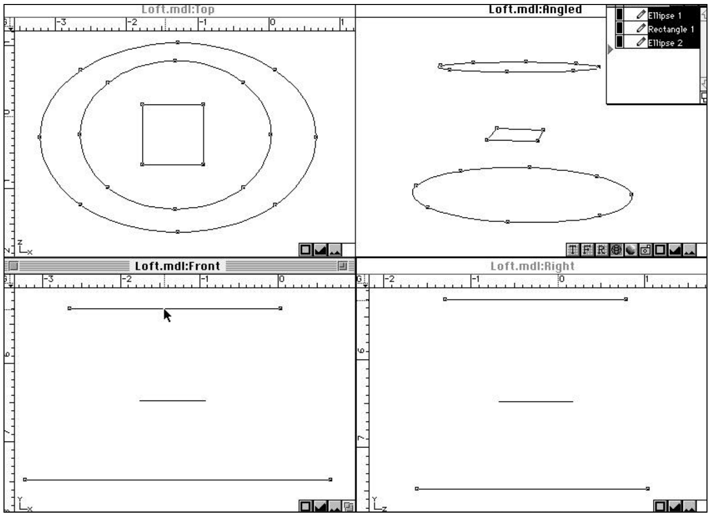
Though it is possible to loft between open curves and closed curves (Lines and Splines that don’t close their beginning point to their endpoint are open curves, Ellipses and Polygons are closed curves), the results generally are less than useful. The best way to work is to use either all open curves or all closed curves.

Since it is very possible that the first curve is very different looking than the second, and has a different number of vertices, ModelPro attempts to make a simple correlation between the vertices of each curve and tries to connect meaningful points between the two curves.

For example, if the two curves are both circles, then ModelPro can match a loft line straight from a vertex on the first circle, to a matching vertex on the second. All of the vertices have a matching partner on the other curve. But if the first curve is a circle and the second curve is a rectangle, then there is a problem. The circle has eight vertices but the rectangle only has four. So ModelPro has to infer extra points along the rectangle so it has something to connect loft lines to from the circle. In general, you can think of lofting as a process of connecting-the-dots from one curve to the next. If the two curves have an unequal number of vertices, then ModelPro just “pretends” that the lesser curve has as many points as the greater curve.

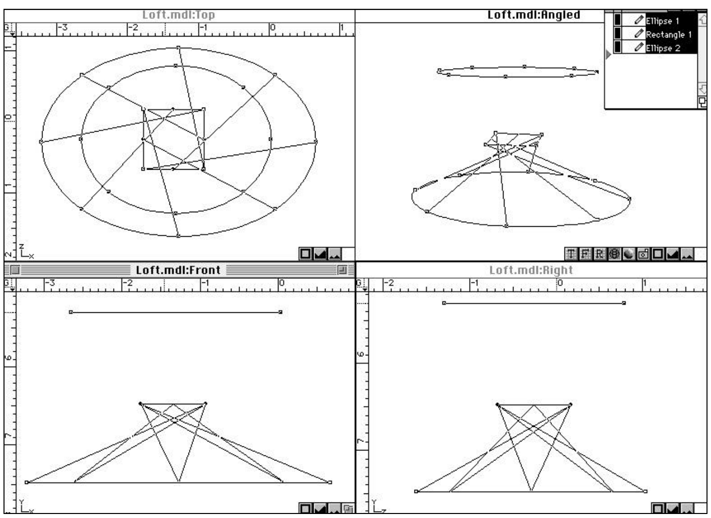
Since there is sometimes some guessing going on within ModelPro about which points to connect, that is why the buttons you see in the Tool Info palette are there.

Let's go back to the example of the two curves that are both circles, one above the other. Instead of correctly connecting the points that are right above each other to give straight, perpendicular loft lines which make a cylindrical surface, it is equally possible to connect from one point on the first circle to a point that is one vertex over (or more), resulting in a loft line that is at an angle. This results in a surface shape that is like a cylinder that has a narrowing in its middle. (This is the shape that is used for the big stacks at nuclear power plants like Three-Mile Island.)



Select curves to be lofted.

So, ModelPro can easily connect the two curves with loft lines that are offset or rotated one, two, or more vertices either direction from one curve to the next. Typically you want to wind up with a loft surface whose loft lines go as straight as possible from one curve to the next. This is where the "Shift Left" and "Shift Right" buttons come into play, as well as the "Reverse" and "Evenly" buttons. They allow you to shift the loft lines around the curves to get the loft surface you desire.



Select Loft.

If the original selected objects are closed curves and they are "filled" (the small ellipse in the Draw-Options portion of the Tool palette was filled in with grey when the objects were created), then the resulting lofted surface object will have its ends "capped." If the original selected objects are closed curves but they are "open" or "not-filled" (the small ellipse was white), then the resulting lofted surface object will have ends that are open, or un-capped.

Using the Loft tool

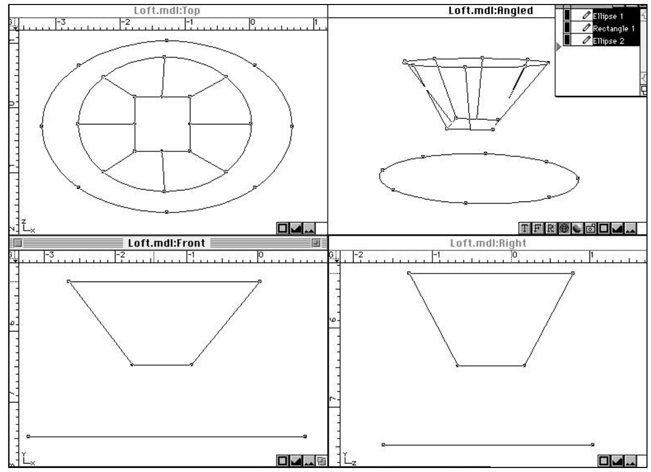
1) Using the Selector tool, select the curves that you want to loft in the order that you want them to be lofted by Shift-clicking on them one after another.

ModelPro will follow your click order to create the skin from one section to the next.

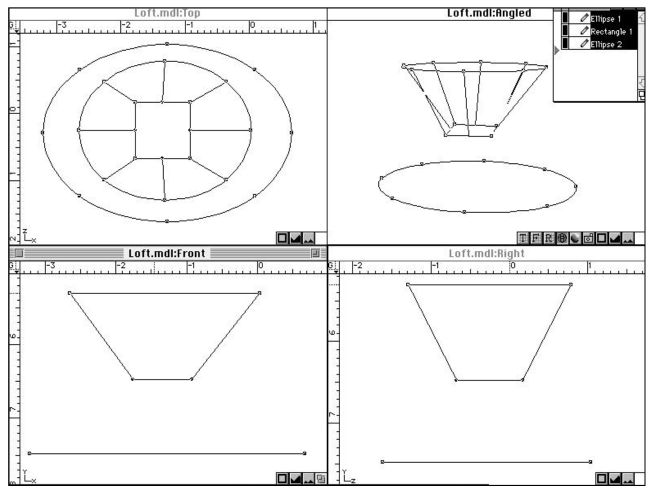
2) Choose the Loft tool from the Tool palette.

ModelPro will process for a moment and then a series of buttons will become visible in the Tool Info palette. The first pair of your curves will become connected with loft lines.

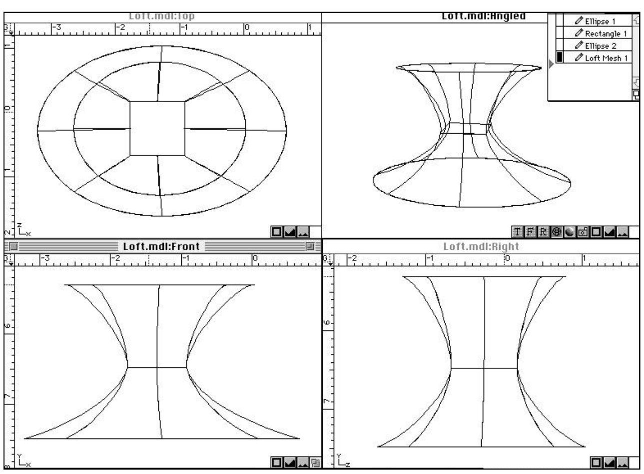
3) Use the “Shift Left” and “Shift Right” buttons, as well as the “Reverse” and “Evenly” buttons to adjust the loft lines to look the way you want between that first pair of curves. Be sure to check the results in all the views.
4) Hit the “Next” button to have ModelPro generate loft lines between the second pair of curves.
5) Repeat steps 3 and 4 for all of the pairs of curve you have selected.
6) Click on the “Circular” check-box if you want the lofted surface to loop back to the first curve from the last curve to make a closed loft surface.
7) Click on the “Smooth” check-box if you want the lofted surface to go through a smooth transition from curve to curve, resulting in curved loft splines. Leave it unchecked if you want the loft lines to be straight-line segments which will result in sharp-cornered transitions from curve to curve.
8) When all the loft sections have been set up to your satisfaction, hit the “Done” button. A separate loft surface entity will be created and your original curves will be retained and “turned off” in the Group palette.



Use the “Shift Right” button.



Hit the “Next” button.



Hit the “Done” button.

Using the Tool Info palette to control the Loft Tool

Previous

Shifts the preview of the loft lines to the pair of curves that immediately precedes the current pair, according to the click order that was used to select the curves.

Next

Shifts the preview of the loft lines to the pair of curves that immediately follows the current pair, according to the click order that was used to select the curves.

Shift Left **Shift Right**

Moves the connection of one end of the loft lines one vertex to the left or right on one of the curves, which changes the apparent angle of the loft lines.

Reverse

Switches the connections of the loft lines on one of the curves to vertices on the “other side” of that curve.

Evenly

Tries to do its best to make a “clean” connection of the loft lines between the two curves. It tries to match up points that will result in a simple surface that is not twisted or buckled on itself.

Done

Completes the operation by putting the lofted surface through all the curves as you have specified.

Circular

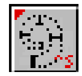
Putting an “X” in this check-box loops the lofted surface from the last curve back to the first curve to make a closed loft surface.

Smooth

Putting an “X” in this check-box makes the lofted surface go through a smooth transition from curve to curve, resulting in curved loft splines. Leave it unchecked if you want the loft lines to be straight-line segments which will result in sharp-cornered transitions from curve to curve.

The Loft Tool Info palette

The Sweep Tool



The Sweep Tool creates a new surfaced object by taking the currently selected objects and rotating them away from their current position along a circular or spiral path, resulting in a surface that is defined by the “sweep lines” of the selected objects. Only lines, arcs, polygons, ellipses, and splines can be swept.

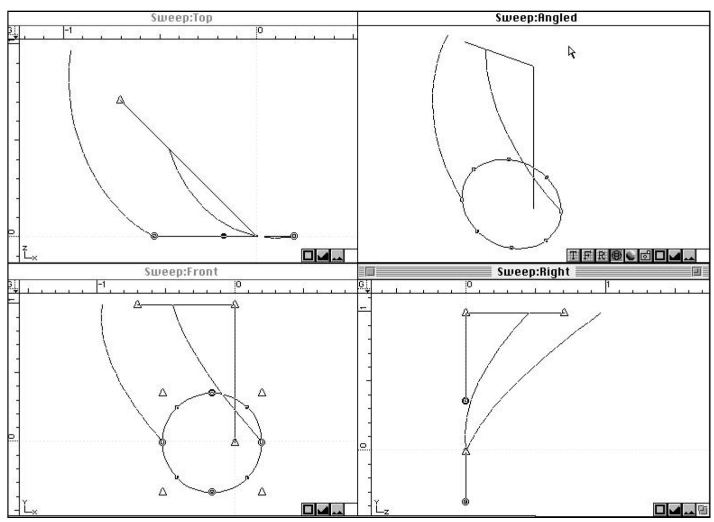
The objects can also be resized and rotated at the end of the sweep, resulting in a shape that is transformed along the length of the sweep.

As soon as you have at least one object selected and you click on the Sweep tool, the selected object(s) becomes enclosed by a bounding rectangle, and a default set of sweep lines appear that originate from the bounding rectangle.

The corner points of the bounding rectangle are marked by small triangles. These triangles are used for resizing the bounding rectangle, resulting in a reshaped and resized object at the end of the sweep.

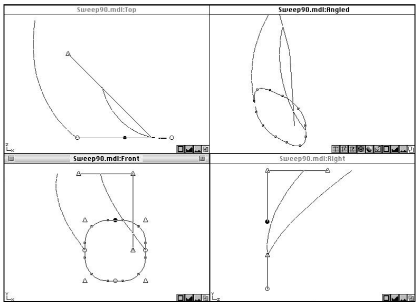
The midpoints of the sides of the bounding rectangle are marked by small circles. These circles are used to rotate the bounding rectangle, resulting in a rotated orientation of the object at the end of the sweep. Note: One of these midpoints is filled in as solid black. This is done to designate the “top” of the initial bounding rectangle so that if you rotate it, you have something to reference the amount of rotation with.

In addition to the bounding rectangle, there is a separate set of three control points that rest on the sweep control lines. These small triangular control points are used to define the center, the offset, the radius, and the angle of the sweep.

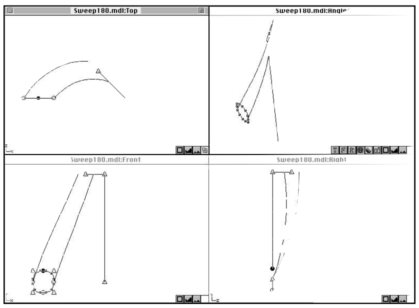


Sweep Tool Control Points

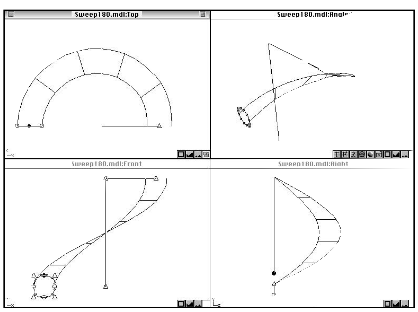
The first control point is the Center control point. Its location defines the center point of the sweep operation. The Center control point is always fixed at a depth that is at the center of the bounding rectangle along the axis whose radio button is highlighted in the Tool Info palette. For example, if the “ΔY” radio button is picked, then the Center control point is fixed at a “Y” depth that is at the center of the bounding rectangle. Its location in “X” and “Z” space is allowed to change.



Select the object and the Sweep tool.



Move center control point and raise off-set control point.



Move to 180º position.

Hint: Another way to activate the Sweep operation, is to double-click in any View window.

Hint: If you click in the By Seg box in the Tool Info palette, then the height and other parameters set here will apply to each segment, and the sweep will go higher as it rotates instead of staying at a constant height.

The axis of rotation goes through the Center control point, and the direction of the axis is determined by the Radio button that is selected (ΔX, ΔY, or ΔZ).

The second control point is the Offset control point. The change in 3-D space between the Center control point and the Offset control point is what gets displayed in the ΔX, ΔY, and ΔZ data fields in the Tool Info palette. This offset "vector" defines a displacement through which all of the copies get distributed. It is through the usage of an offset vector that a spiral and/or nautilus type of rotation can be achieved.

The third control point is the Angle control point. Its job is to control what amount of angular rotation the selected object(s) goes through about the axis of rotation. It also acts to define the ending Radius of the sweep. The distance between the Angle control point and the Offset control point is the ending radius of the sweep. The beginning radius of the sweep is the distance between the Center control point and the center of the bounding rectangle.

If the original selected objects are closed curves and they are "filled" (the small ellipse in the Draw-Options portion of the Tool palette was filled in with grey when the objects were created), then the resulting sweep surface object will have its ends "capped." If the original selected objects are closed curves but they are "open" or "not-filled" (the small ellipse was white), then the resulting sweep surface object will have ends that are open, or un-capped.

Using the Sweep tool

1) Select the Object(s) that you want to sweep. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Sweep tool from the Tool palette. (Or press Shift-S on the keyboard.)

You may reverse step 1 and 2.

3) Move the center control point to the right, so it is clear of the Object in the Front View. This will set the radius for the sweep.
4) Raise the off-set control point to the height you want the sweep to reach in the Front View.

Look in the **ΔY** box in the Sweep Tool Info palette. This is the value of the height you set. You can change it by double-clicking in the box and entering new value.

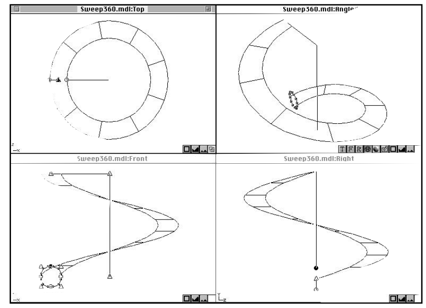
5) Click-hold on the angle control point in the Top View. Slowly move to the right and down till you reach the 180° position.

Notice the height remains the same even though the sweep is going up. If the sweep° box in the Info palette is not exactly 180°, you can enter 180 in the data field.

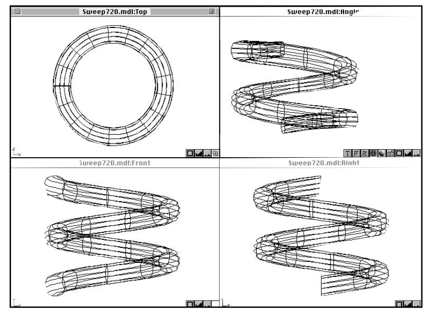
6) Click-hold on the angle control point and move down, left, and then up, in the Top View, till the end of the sweep meets the beginning.

The angle in the sweep° box should be close to 360°.
Notice that the height is still at the preset level.

7) Click-drag another full turn till the end meets the beginning again. The height will still be the same and the sweep° should be near 720°.
8) Hit the “Sweep” button in the Tool Info palette.



Move to 360º position.



Move to 720º position and hit the Sweep button.

The Selected object(s) will be “swept” around the axis of rotation line and transformed in a gradual fashion according to the ending bounding rectangle. A new, surfaced object will be created along that sweep. The original selected object is saved separately and “turned-off” in the Group palette, unless otherwise specified in your Preferences. They can be restored at anytime.

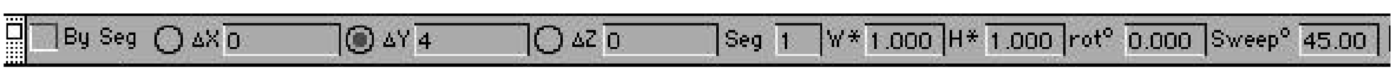
Controlling the Sweep tool with the Tool Info palette

By Seg (Check-box)

Put an “X” in this check-box to use the “Sweep °” for the rotation of each segment of the sweep, rather than the rotation of all the segments as a whole.

ΔX, ΔY, ΔZ (Radio Buttons)

Controls the displacement of the Offset control point relative to the Center control point. This displacement defines a vector that is the axis of revolution. Click on one of the Radio Buttons next to ΔX, ΔY, or ΔZ to indicate which axis the sweep angle will be measured about. Enter values from the keyboard for ΔX, ΔY, and ΔZ, or drag the small triangular Offset control point to set the location of the Offset control point graphically.



Sweep Tool Info palette

Segments

Defines how many copies of the original shape(s) get distributed through the final object as “ribs” to support the surface.

ModelPro automatically adjusts this value to compensate for larger or smaller sweep angles; roughly 3 segments for every 90° of sweep rotation.
You can override the automatic value and put one in of your own.

W*, H*

Defines the percentage in change of width and height of the object(s) from the beginning to the end of the sweep, thereby resizing and reproportioning the object(s) along the way.

Enter a value to indicate the ratio of the ending size compared to the beginning size. Values less than 1 shrink the object. Values larger than 1 expand the object. Or, drag the small triangular control points at the corners of the bounding rectangle to graphically change the width or height.

Rot °

Defines the rotation of the copy of the object at the end of the sweep line compared to the orientation of the object at the beginning.

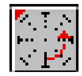
Enter a value between 0° and 360°. Or, drag the small circular control points on the sides of the bounding rectangle to graphically set the rotation angle.

Sweep °

Defines the angular rotation of the selected shape(s) about the axis of rotation.

Enter a value between 0° and 360°. Or, drag the small triangular Angle control point to graphically set the rotation angle.

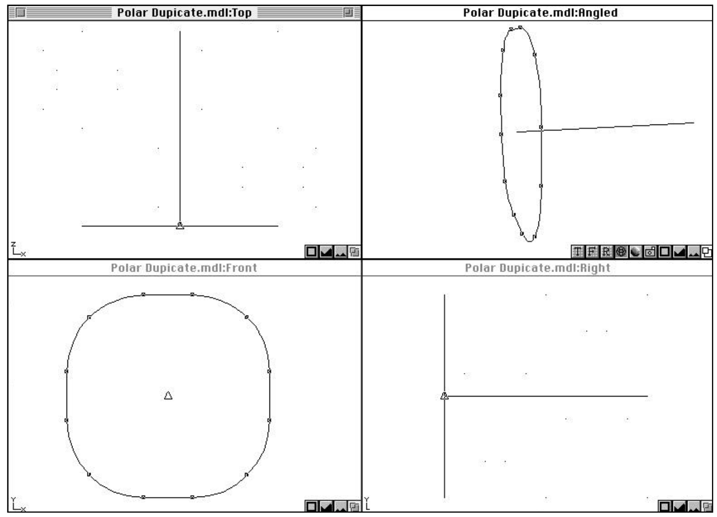
The Polar Duplicate Tool



Note: The Polar Duplicate tool is on the Sweep tool pop-up.

The Polar Duplicate Tool creates a user-defined number of copies of the currently selected objects by taking those objects and rotating them away from their current position along a circular or spiral path and distributing the copies along that path. Any object can be polar duplicated.

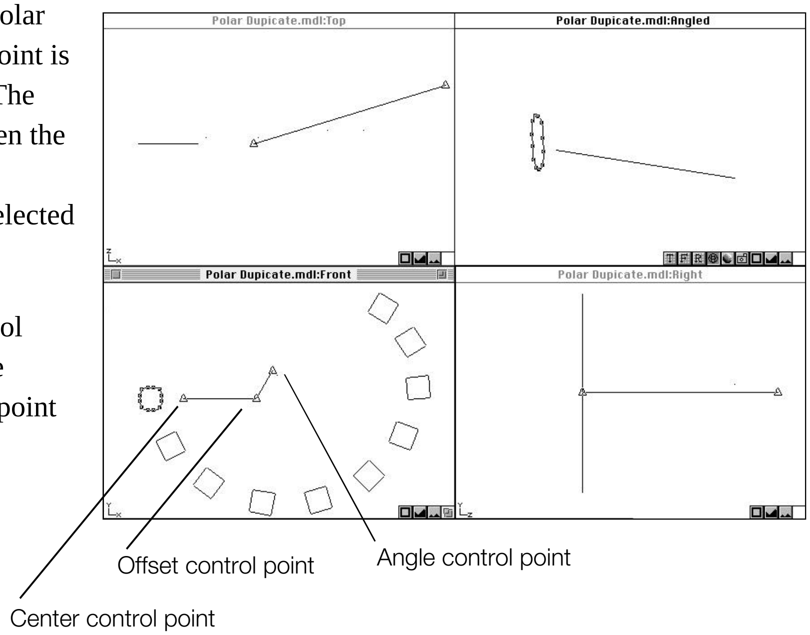
As soon as you have at least one object selected and you click on the Polar Duplicate tool, a set of three small triangular control points appear. (By default, some or all of these control points may initially overlap each other, thereby appearing as a only one or two small triangles, instead of three. But if you drag on it/them, they will separate from each other, and all three control points will become visible.) These small triangular control points are used to define the center, the offset, the radius, and the angle of the polar duplication.

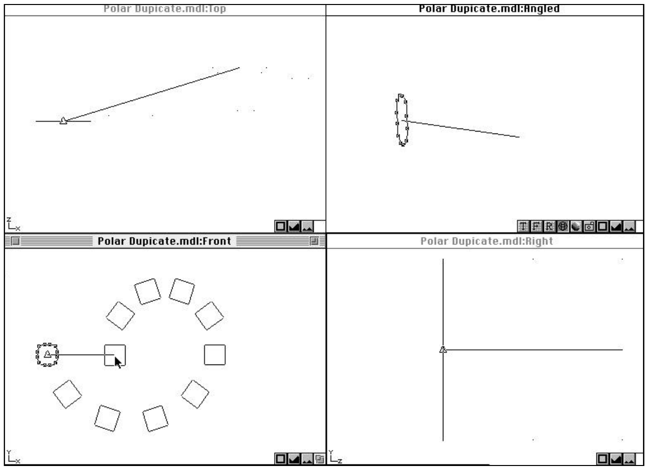


Overlapping control points.

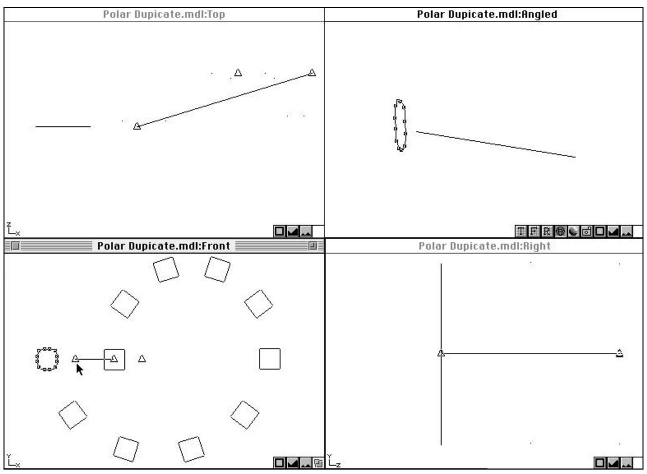
The first control point is the Center control point. Its location defines the center point of the polar duplication operation. The Center control point is free to move to any location in 3-D space. The Radius of the rotation is the distance between the Center control point and the center of an imaginary bounding rectangle around the selected objects.

The second control point is the Offset control point. The change in 3-D space between the Center control point and the Offset control point defines an offset vector that defines a





Drag the Offset control point to the right.



Drag the Center Control point to the right.

displacement through which all of the copies get distributed as they go through their angular rotation. This would be useful for creating an effect like a spiral staircase. Each tread get rotated in a horizontal plane, but each step also gets an incremental offset in the vertical direction.

The third control point is the Angle control point. Its job is to control what amount of angular rotation the selected object(s) goes through about the axis of rotation.

Hints for using the Control Points

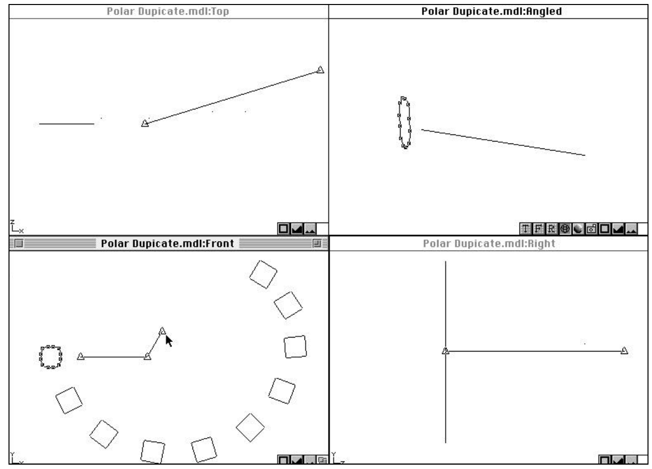
Generally, the most common polar duplication that you will want to create is one where the selected objects are rotated through an angle that you define, about a center point that you define, but with no offset. The way to achieve this is to first locate the center of the rotation by dragging the Center control point to that spot. Then drag the Offset control point right on top of the Center control point (the two triangles will exactly overlap, and even disappear), thereby defining zero offset. (Make sure you match the two points in all of the orthogonal views.) Then you can drag the Angle control point to define the rotation amount about that center point.

Using the Polar Duplicate tool

1) Select the Object(s) that you want to duplicate. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Polar Duplicate tool from the Tool palette.

You may reverse step 1 and 2.

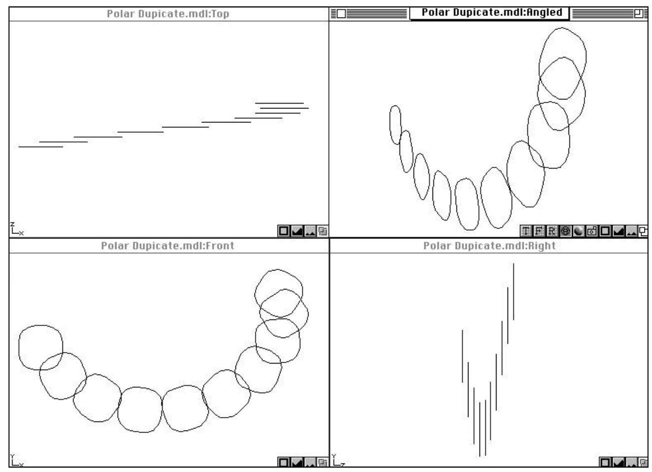
3) Enter 10 for the number of copies that you want to create in the “Total Copies” field in the Tool Info palette.
4) Make modifications to duplication “preview” by dragging the off-set control point to set up the duplication you want. Move this point around to see what effect it has.
5) Drag the center control point to the right and around to notice the effect.



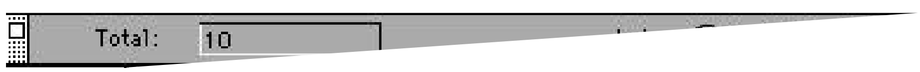
Move the Angle control point.

The angle control point should now be visible.

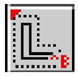
6) Drag the angle control point around to see what happens to the duplicated objects.
7) When everything is as you want, hit the “Duplicate” button in the Tool Info palette, or double-click in any View window.



Hit the Duplicate button.

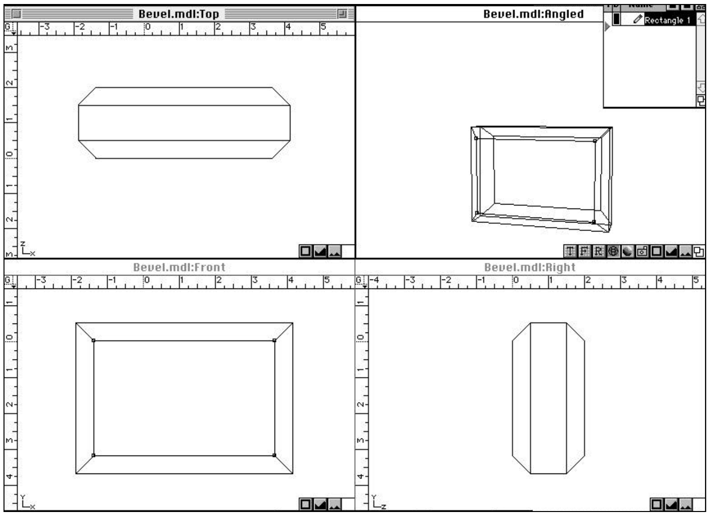


Polar Duplicate Info palette



The Bevel Extrude Tool

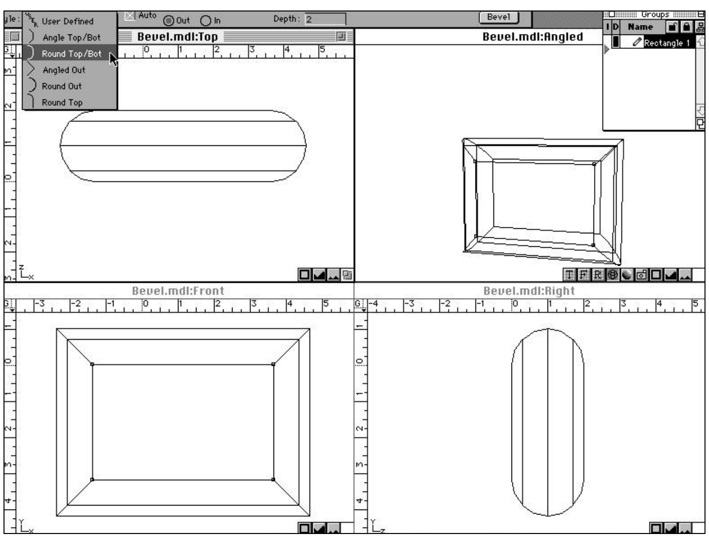
The Bevel Extrude Tool creates a new surface mesh object by doing a perpendicular extrusion of selected objects, while shaping the extrusion lines according a user-defined contour, thereby making the sides of the extruded form “bulge-out” or “draw-in” to follow the contour. Lines, arcs, polygons, ellipses and splines may be used.



Select the object and pick Bevel Extrude.

It is best to keep the original shape(s) planar. And it is best to keep the orientation of the shapes in one of the orthogonal planes. And, it is best that all of the selected objects have the same planar orientation. Otherwise, you are likely to get very unpredictable results.

If the original selected objects are closed curves and they are “filled,” then the resulting extruded surface object will have its ends “capped.” If the original selected objects are closed curves but they are “open” or “not-filled” (the small ellipse was white), then the resulting extruded surface object will have ends that are open, or un-capped.



Pick the Round Top/Bot style.

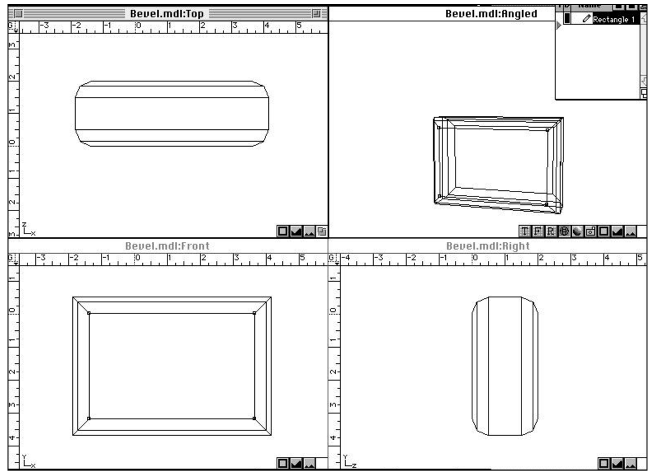
Using the Bevel Extrude tool

1) Select the Object(s) that you want to extrude. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Bevel Extrude tool from the Tool palette. Or press Shift-B.

You may reverse step 1 and 2.

3) Pick the “style” of bevel you would like from the pop-up list in the Tool Info palette.
4) Pick whether you want the extrusion lines to “bulge out” or “shrink-in” on the finished shape by clicking either the “Out” or the “In” radio buttons.

5) Enter the length of the extruded object that you want to create by typing in a desired value in the “Depth” field in the Tool Info palette.
6) Move the small arrow “slider” to define the effective radius of the final object.
7) Click on the “Auto” check-box, putting an “X” in it, if you want the extrusion to close on itself.
8) When everything is as you want, hit the “Bevel” button in the Tool Info palette, or double-click in any View window..

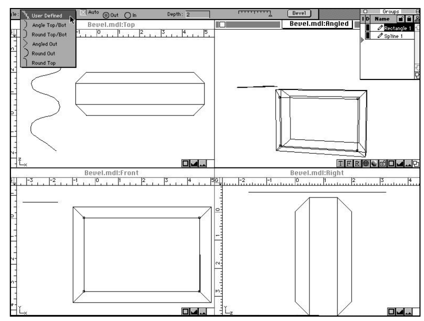


Hit the Bevel button.

Using the Bevel Extrude tool with a "User Defined" bevel

The “User Defined” style requires that a path exists and is defined by a separate line, spline or arc object. Things work best if the path is planar, but it does not have to be.

1) Select the Object(s) that you want to extrude. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Bevel Extrude tool from the Tool palette. (Or press Shift-B on the keyboard.)
3) Pick the "User Defined" style option from the pop-up list in the tool info palette.
4) Holding the Option key, click on the object you want to act as the path.
5) Hit the "Bevel" button.



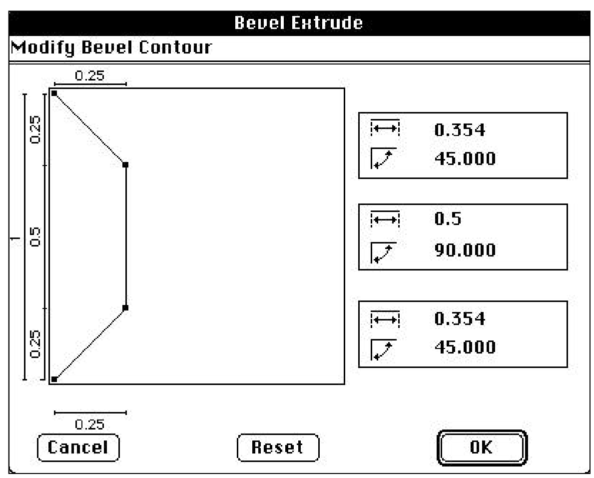
Select the object pick Bevel Extrude and choose the "User Defined" style.

Controlling the Bevel Extrude tool with the Tool Info palette

Edit

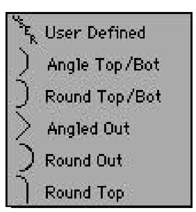
This button allows the user to customize the aspect ratios of each of the different bevel styles.

When you press this button, a dialog box pops up that shows the layout of the current bevel style. You can drag the vertex points of the curve to edit the way it looks. The numbers you see relate to the “Depth” value that is set here in the Tool Info palette.



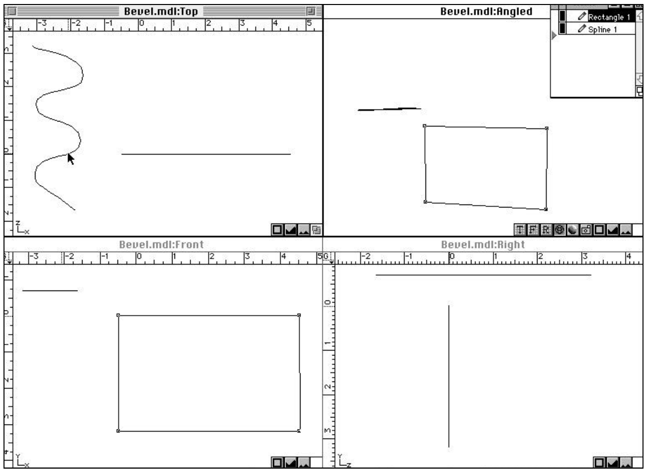
Bevel Extrude Edit box

Style Pop-up Menu

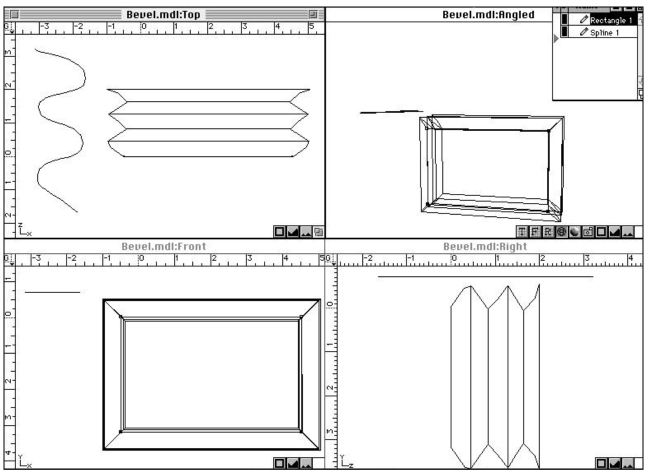


Shows a list of default bevel styles. The “User Defined” option allows you to pick a separate spline object to act as the bevel shape.

Press and hold the mouse on the pop-up menu, drag down to select one of the choices, and release the mouse. The style you select will be displayed in a “preview” mode on the screen. (You may need to give the command “Re-Display” in the “Windows” menu to force ModelPro to re-draw the screen.)



Hold the Option key and select the path.



Hit the Bevel button.

Auto

If two objects are extruded at the same time with one completely inside of the other, then this function makes the bevel of the two objects go opposite directions.

If the Auto check-box is checked, then the two shapes will bevel in opposite directions. The inner shape will bevel “in” and the outer shape will bevel “out,” or vice versa. This way the “intent” of the overall part is maintained. If there is no check, then the two shapes will both bevel in the same direction.

Out, In

These radio button are used to control whether the bevel lines curve away from the center of the part or towards the center.
Just click on the one you like.

Depth

This defines the length of the extrusion.

Hit the Tab key enough times until this field highlights, or double-click in the field to select its contents and type in a new value.

Width Slider

This slider controls the width of the finished object relative to the width of the original object.

With the slider all the way to the right, the width of the original object is used to define the width of the end-caps of the extruded object.

With the slider all the way to the left, the width of the original object is used to define the width of the widest or narrowest part along the length of the extrusion.

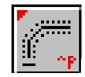
Bevel

Completes the operation.

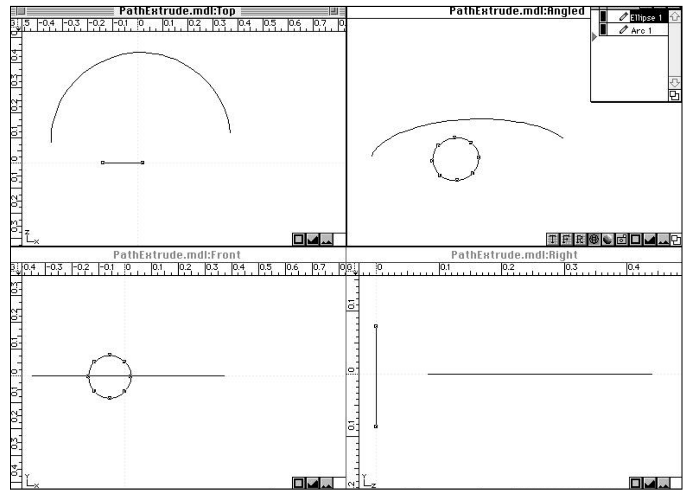
Click and you’re done. A separate surfaced object is created. The original object is maintained, though “turned-off” in the Group palette, unless otherwise specified in Preferences.

Bevel Extrude Tool Info palette.

The Path Extrude Tool



The Path Extrude tool creates a new, surfaced object by taking the currently selected objects and projecting them along a user-defined path, resulting in a surface that is defined by the "extrusion lines" of the selected objects. Only objects made up of lines, arcs, ellipses, and splines can be Path Extruded.

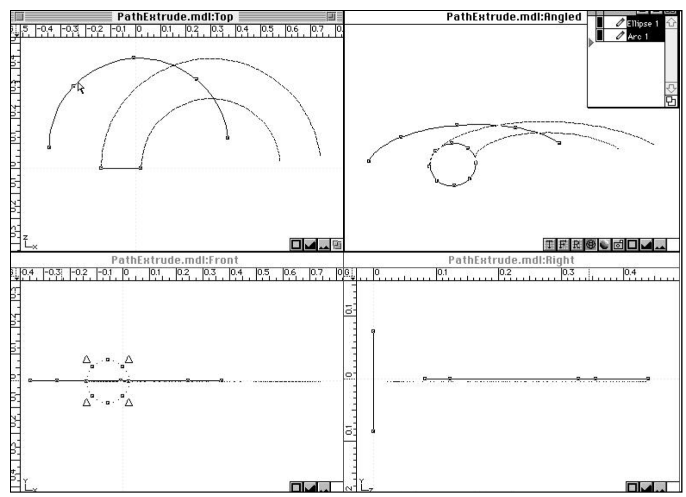


Select the object and Path Extrude.

The objects can also be resized at the end of the extrusion, resulting in a shape that is transformed along the length of the extrusion.

The one limitation with this tool is that the objects that are going to be extruded must be drawn in the Front view window. That is, they must in the X-Y plane (therefore perpendicular to the Z axis). The path will therefore lie in either the top or the right side view windows.

If the original selected objects are "filled" (the small ellipse in the Draw-Options portion of the Tool palette was filled in with grey when the objects were created), then the resulting extruded surface object will have its ends "capped."0 If the original selected objects are "open" (the small ellipse was white), then the resulting extruded surface object will have ends that are open, or un-capped.



Hold the Option key and select the path.

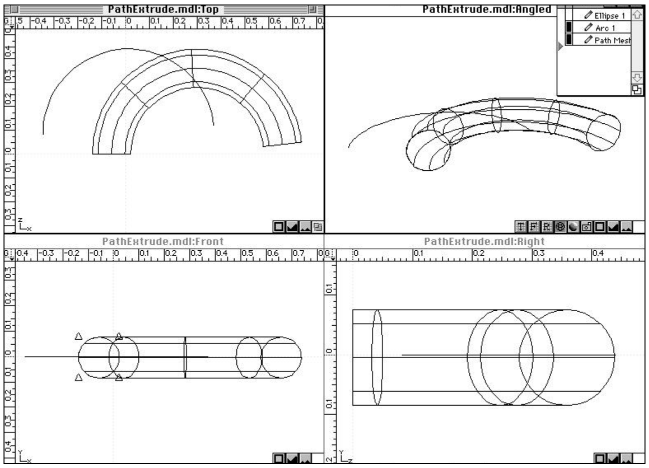
Using the Path Extrude tool

1) Select the Object(s) that you want to path-extrude with the Selector tool. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Path Extrude tool from the Tool palette, or press Shift-P.

You may reverse step 1 and 2.

3) Hold the Option key and click on the path along which the object will be extruded. As soon as you select the path, a "preview" of the new extruded object appears.
4) Make modifications to the slider, buttons, and data fields shown in the Tool Info palette by typing in new values to make the preview match what you want. When everything is set, hit the "Path Extr" button, or double-click in any View window.

Note: The Path is defined by a separate line, spline or arc object. Things work best if the path is planar, but it doesn't have to be. The path can be a 3-D spline. The path doesn't have to be hooked to, or share a point with the object to be extruded.



Hit the Path Extr. button.

The selected shape(s) will be placed at the end of the path, and made perpendicular to the path, and then “extruded” along the path and transformed in a linear fashion according to the ending width and height that was specified. A new, surfaced object will be created along that extrusion. The original objects are saved separately and “turned-off” in the Group palette unless otherwise specified in Preferences. They can be restored at anytime.

6) Hit ⌘-z to undo the path extrude. Repeat steps 1 and 2.
7) Select "On Path" in the Tool Info palette at the top of the screen. Repeat steps 3, 4, and 5.

Controlling the Path Extrude tool with the Tool Info palette

Position Slider

With the slider all the way to the right or all the way to the left, one edge or the other of the selected objects will follow the path. With the slider in the middle, then the center of the selected objects will follow the path.

On Path (radio button)

With this button “on,” the object is moved to the path and the extrusion occurs on the path at its current location. Click this button and the preview will show the corresponding results.

On Contour (radio button)

With this button“on,” the path is moved to the contour (the selected object) and the extrusion occurs on the contour at its current location. Click this button and the preview will show the corresponding results.

W*, H*

Defines the percentage in change of width and height of the object(s) from the beginning to the end of the path extrude, thereby resizing and reproportioning the object(s) along the way.

Enter a value to indicate the ratio of the ending size compared to the beginning size. Values less than 1 shrink the object. Values larger than 1 expand the object. Or, drag the small triangular control points at the corners of the bounding rectangle to intuitively change the width or height.

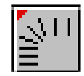
Path Extrude (button)

Completes the operation.

Click and you’re done. A separate surfaced object is created. The original object is maintained, though “turned-off” in the Group palette, unless otherwise specified in Preferences.

Path Extrude Tool Info palette.

The Path Duplicate Tool



The Path Duplicate tool takes the currently selected objects and distributes a user-defined number of copies of those objects along a user defined path.

The Path is defined by a separate line, spline or arc object. Things work best if the path is planar, but it doesn't have to be. The path can be a 3-D spline. The path doesn't have to be hooked to or share a point with the object to be duplicated.

Using the Path Duplicate tool

1) Select the Object(s) that you want to path-duplicate with the Selector tool. It does not matter if the object is selected with its vertices visible or if its bounding box corners are showing.
2) Pick the Path Duplicate tool from the Tool palette.

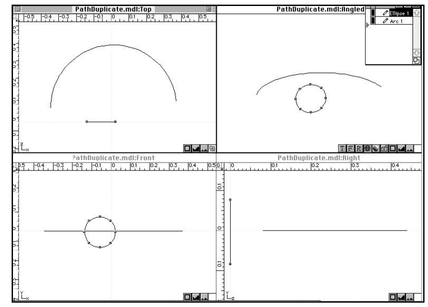
You may reverse step 1 and 2.

3) Hold the Option key and click on the object that you want to act as the path.

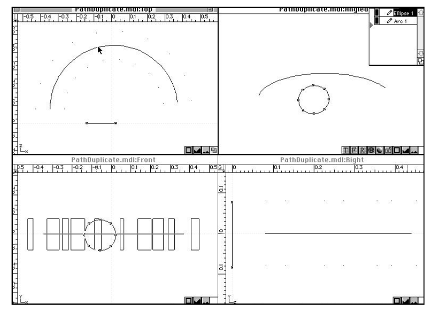
As soon as you select the path, a "preview" of the new duplicated objects appears.

4) Indicate how many copies of the object you want by typing in a new value to make the preview match what you want.
5) Indicate whether you want the new copies to rotate about either the "X," "Y," or "Z" axis as they follow along the path by clicking on the appropriate radio button.
6) When everything is set, hit the "Duplicate" button in the Tool Info palette, or double-click in any View window.

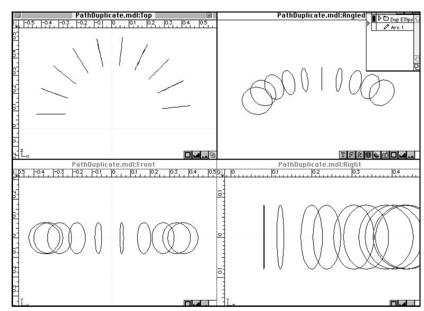
The selected shape(s) will be placed at the end of the path, made perpendicular to the path, and then "duplicated" along the path. The object(s) will run their center along the path.



Select the object and Path Duplicate.

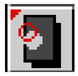


Hold the Option key and select the path.



Select 10 copies and hit the Duplicate button.

Path Duplicate Tool Info palette.



The Punch Tool

The Punch tool is used to cut a hole of a user-defined shape through a surface mesh object. The shape of the hole is defined by a closed planar curve. Only Primitive objects and Spline Mesh objects can be used for Punch operations.

Note: The result of a Punch operation on a 3-D Primitive or spline mesh is a polygonal object.

Note: The punched object is still a surface object that is empty inside.

To create the hole in the surface mesh object, ModelPro makes a cut where the perpendicular projection of the curve intersects the surface mesh object. The “punch-curve” will cut through all portions of the surface mesh object that lies along that perpendicular projection. For example, if a curve is set to punch through a sphere, the hole will appear in both the front and the back of the sphere.

The curve that is used to punch must be a closed curve. It you try to use an open curve, ModelPro will connect the beginning point to the end point with an imaginary line to force the curve to be closed, at it will try to cut with that “closed” curve. The results are likely to be unsatisfactory.

The boundary of the “punch-curve” is allowed to go “outside” of the boundary of the spline mesh object. In other words, you can cut a partial hole in the mesh object.

The position of the punch curve relative to the mesh object is irrelevant. It doesn’t matter if the punch curve is in front of the mesh, or behind it, or right in the middle of it. ModelPro will make sure that the punch curve “sees” every bit of the mesh object that lies within the perpendicular projection of that curve, no matter where the curve itself lies.

You can only punch one object at a time with a curve. ModelPro will just beep at you if you select two or more spline mesh objects and then pick the Punch tool.

However, you can punch a single spline mesh object with more than one separate punch curve in the same operation. And, each curve can punch in a different direction if you want.

Using the Punch tool

1) Select the object that you want to punch. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Punch tool from the Tool palette.

You may reverse step 1 and 2.

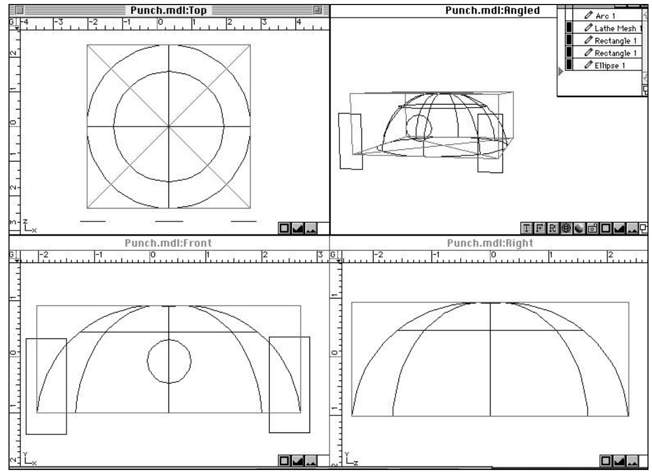
As soon as you select the spline mesh object, it gets a red bounding box around it. This indicates that ModelPro knows THIS is the object to be punched.

3) Click on the curve(s) that you want to act as the punch object. Use the Shift key to click on additional curves.

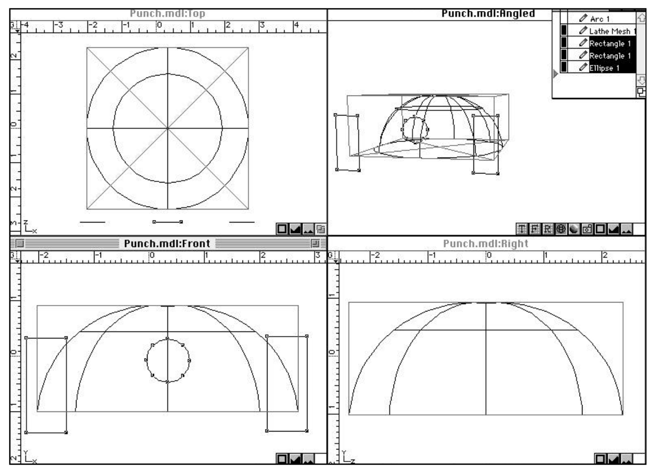
The curves selected as “punches” cut in a direction perpendicular to the window plane. Therefore, they can be placed at any depth (i.e. in front, behind or even inside the object to be cut).

4) Indicate a Mesh Density in the Tool Info palette. The default value is 3.The higher this number gets, the more complex the surface mesh becomes. This leads to far greater file size, but also greater accuracy.
5) When everything is set, hit the “Punch Out” button in the Tool Info palette, or double-click in any View window.

The surface mesh object gets punched according to the punch curve, and the resulting surface mesh is much more detailed.

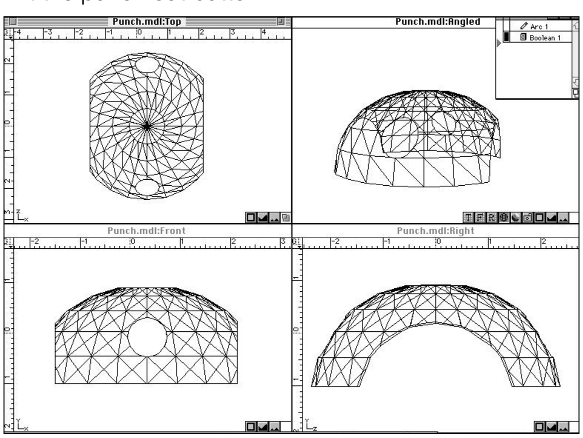


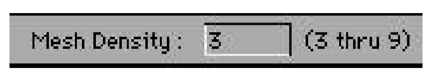
Select the object and Punch.



Click on the two rectangles and the circle acting as punches.

Hit the punch out button.

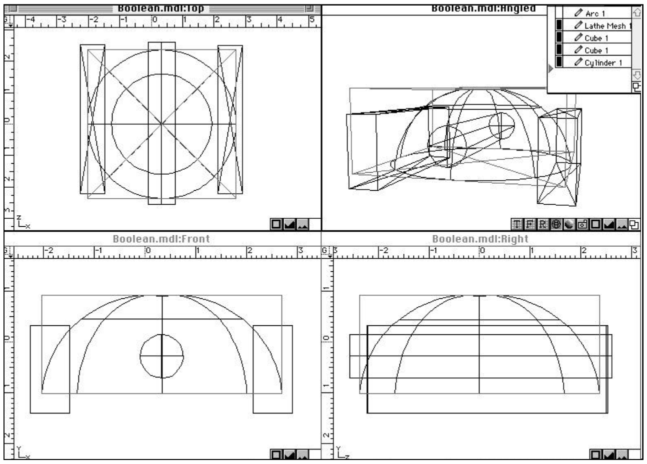




Punch Tool Info palette.

The Boolean Tool

The Boolean tool is used to generate volumetric shapes that result from the intersection or union of two or more 3-D objects. Only Primitive objects, Spline Mesh objects, and Boolean objects can be used for Boolean operations.



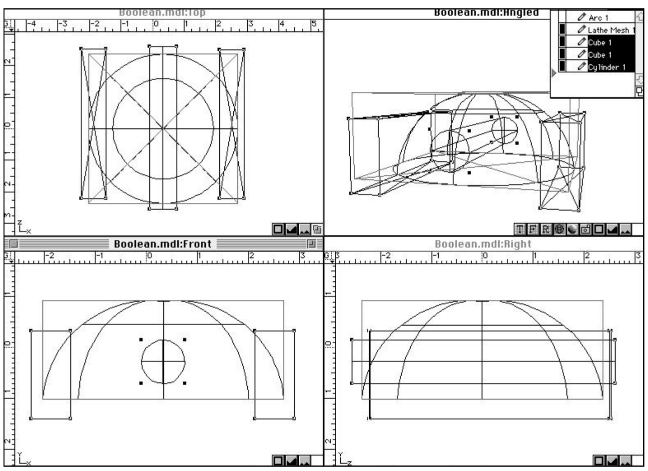
Select the object and the Boolean tool.

Using the Boolean Tool

1) Select the object that you want to operate on. It does not matter if the object is selected with its vertices visible, or if its bounding box corners are showing.
2) Pick the Boolean tool from the Tool palette.

You may reverse step 1 and 2.

As soon as you pick the Boolean tool, the selected object gets a red bounding box around it. This indicates that ModelPro knows that THIS is the object to be operated on.



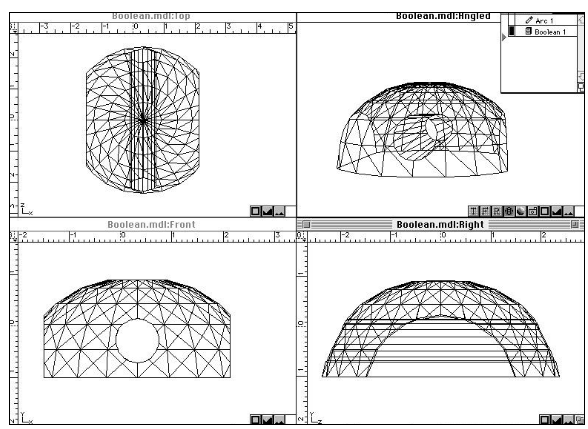
Click on the two rectangular boxes and the cylinder operating on the initial object.

3) Click on the 3-D object(s) that you want to use to operate on the initially selected object. Use the Shift key to click on additional objects.
4) Indicate whether you want to derive the Intersection, Difference, or Union of the objects by clicking on the appropriate radio button in the Tool Info palette.
5) Indicate a Mesh Density in the Tool Info palette. The default value is 3.
6) When everything is set, hit the “Make it So” button in the Tool Info palette or double click in any window.

A new “Boolean” object is created. The original shapes that contributed to this new shape are not preserved in the Group palette.

The result of a Boolean operation on a 3-D primitive or spline mesh is a polygonal object. The Mesh density option allows you to specify the complexity of the resulting surface mesh. The higher this number gets, the more complex the surface mesh becomes. This leads to far greater file size, but also greater accuracy in model geometry, and smoother rendering.

Hit the Make It So button.



Controlling the Boolean tool with the Tool Info palette

Difference (radio button)

This button tells ModelPro to find the portion of the second selected object that intersects with the first, and subtract that portion from the first. This leaves behind the part of the first object that was completely outside of the second (and third, etc.) object. Just click on this button to select this mode.

Intersection (radio button)

This button tells ModelPro to find the portion of all the selected objects that is common to all objects, and delete everything else. That is, the volumetric area that is common to all the selected objects will be retained. Just click on this button to select this mode.

Union (radio button)

This button tells ModelPro to create a new shape that is the outer surface of all the combined forms. That is, the volumetric area that is enclosed by all the selected objects will result. Just click on this button to select this mode.

Mesh Density

This value defines how finely ModelPro breaks up the spline mesh into polygon representation. The default value is 3. The higher this number gets, the more complex the surface mesh becomes. This leads to far greater file size, but also greater accuracy.

Make it So (button)

Completes the operation. Click and you're done. A new, solid object is created.

Boolean Tool Info palette