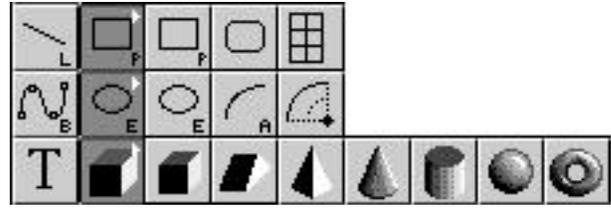


Object Creation Tools

The six Object creation tools are perhaps the most important in ModelPro, for it is with these tools and their sub-tools that all of the objects in the file get created. These are the starting points of all the work that is to come.



Objects can be created with these tools in any of the three orthogonal view windows. The Creation tools are inactive in the Angled view window except in those cases where you lock onto existing points.

Remember to think in 3-D. Although some of these tools are 2-D in nature, the space in which you are working is 3-D. To help you work in 3-D, with 2-D drawing tools, ModelPro employs something called Depth Lock.

Using the Depth Lock with the Creation tools

When creating an object in one of the Orthogonal windows, it is not immediately obvious where the object comes to rest with regard to the depth of view in and out of that view window. Is it close to you or far away from you? And how can you control this?

When you create a planar object in the Top view, it will be seen as a horizontal line in the Front view since you are looking at it edge-on. You will also see that right after creating this object, the line that is seen as its “edge” view in the Front view lies right on the current horizontal Depth Line. That is to say, the object will always be placed on the plane defined by the Depth Lines. The only way that this can be overruled is if you pick a snap mode and define the beginning of the object to be snapped to another object that is at some depth other than the Depth Line.

So, when placing an object in one of the orthogonal views, you control its “elevation” in the other two views by setting the Depth Origin ahead of time.



To set the Depth Lines, put the cursor at a desired location in one of the orthogonal view windows. Press and release the Space Bar on the keyboard once. You will hear a mechanical voice say the word “Lock” and the Depth Lines will jump to your cursor location. By doing this, you have set a location for the Depth Origin in two of the three axis. To set the third axis, move the cursor into one of the other orthogonal view windows and position the cursor to the desired location and press the Space Bar again.

By setting this Depth Origin, you are telling ModelPro to place the next object “Here!” with regard to depth.

You can use the Snap options to help you locate the Depth Origin exactly on existing objects. It doesn’t matter what tool you have when you do this. You do not have to have the Selector tool.

Using the Snap Options to help set the Depth Origin

- 1) Click on a snap option in the Snap palette or type the key for the corresponding snap option.
- 2) Position the cursor near the point where you want to snap the Depth Origin. Be near, and be on the snap object.
- 3) Hit the Space Bar on the keyboard and the Depth Option will jump to the snap point on the object you have specified. (This fixes the Depth Origin in two of the three axis.)
- 4) If necessary, do the same thing in one of the other orthogonal views to set the Depth Origin in the third axis.

For more information on Depth Lock, consult the ModelPro Windows chapter in this manual.

The Line Tool

The Line tool creates a straight line segment between any two points in 3-D space.

Using the Line tool

- 1) Select the Line tool from the tool palette (or press “L” on the keyboard).
- 2) Press and hold the mouse at a point to be the beginning point of the line.

Use the snap options and the depth lock to help you locate the point.

- 3) Drag the mouse and release it at the point you want to be the end point of the line. Again, use the snap options and the depth lock if you want.

Creating a Connected Line

As soon as a line object is drawn, notice that the ending point is still highlighted with a solid black square. This is telling you that ModelPro is ready for you to continue with the line tool to add another line segment and make a connected line. If you immediately begin a new line segment by clicking within five pixels of the previous ending point, the new line segment will automatically be connected to the first and the new line will start there. You don’t need to use the vertex snap to accomplish this. You can end the line wherever you want after that. The two lines are connected into a single line object. And in fact, this is true for the Spline and Arc tools as well. That is to say that you can make a single continuous multi-line object with any combination of lines, splines, and arcs simply by starting a new object within five pixels of the highlighted vertex of the previous object.

Using the Snap options with the Line tool

- 1) Select the Line tool from the tool palette (or press “L” on the keyboard).
- 2) Click on a snap option in the Snap palette, or type the key for the corresponding snap option.
- 3) Press and hold the mouse very near the desired snap point.

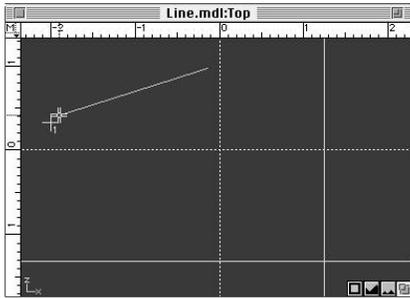
The important thing here is to press the mouse on the object that you are trying to snap to. Don’t just be near the snap point. Be near, and be on the snap object.



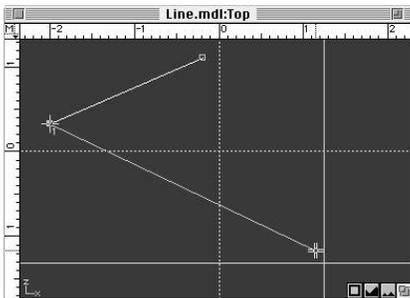
Note: This tool can be selected directly by typing “l”.

Note: If you don’t drag the mouse after initially pressing the mouse button, that is, if you press and release the mouse button at the same location, you will create a line object with zero length. In essence, this is a Point. You must be careful with this because these “points” have little or no value in ModelPro and can actually get in the way.

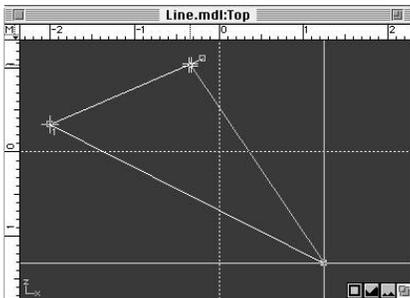
Note: Snapping options will work in all of the View windows, including the Angled view window.



Drag to first snap point (Marker).



Drag to second snap point (Guide).



Drag to last snap point (Vertex).

- 4) While dragging the mouse, take the cursor over to the Snap palette and release the mouse on the snap option you want. The line will remain attached to the mouse as if you were still holding the mouse button down. You can move the mouse around the screen and the line will follow; or, while dragging the mouse, press the key that corresponds to the snap option you want.
- 5) Still dragging the mouse (if you released on the snap palette, the mouse button is not being held down), take it to the snap point where you want to end the line. Be sure you are near the snap point and on the snap object, then release the mouse.

Using the Tool Info palette to control a Line

The two attributes that you can specify to define a line are Length and Angle.

To specify a length for a line, put an "X" in the check box next to the Length field and type in a value for length. This tells ModelPro to make a line of that length regardless of mouse actions. Then, after the beginning point is defined, as the mouse is dragged around, a line of the specified length follows.

To control the Angle of the line, the rest of the buttons and fields in the Tool Info bar are used.

By default, the "None" radio button is highlighted to indicate that no angular constraints are in effect. You are free to drag the mouse around the screen and the line will follow at the arbitrary angles define by those movements.

To define an angular constraint, click on the "Fixed" radio button and enter a value for the Angle you want. (If you leave the field blank, then ModelPro will assume an angle of zero degrees.) When you press a point to begin the line, and then move the mouse around, the line will appear

Line Tool Info palette.



only at the positive and negative values of the angle you specify. (Remember, zero degrees is defined to be at the twelve o'clock position.) Each of the three orthogonal views behave the same with respect to this angle. The angle of the line is always defined with respect to the view window that the cursor is currently in. It's like each of the orthogonal view windows become purely 2-D windows for a moment. This angle value has no meaning in a 3-D sense.

The Polygon Tool

The Polygon tool creates a closed multi-line object that has straight sides of equal length. Most commonly it is used for creating rectangles, but it can create polygons of any number of sides. The polygon object itself is created inside a bounding rectangle whose width and height can be defined by the user numerically.

A polygon can only be created in the three orthogonal planes. It will be automatically be placed at the Depth location as defined by the Depth Origin and Depth Lines unless the Snap Options are used to snap the polygon to other objects already on the screen. It can be created normal at first, and then rotated as a separate operation using the Rotate tool.

Using the Polygon tool

- 1) Select the Polygon tool from the tool palette (or press "p" on the keyboard).
- 2) Press and hold the mouse at a point to be the beginning point of the polygon bounding rectangle.
- 3) Drag the mouse and release it at the point you want as the end point of the bounding rectangle. (Hold the Shift key to constrain the bounding rectangle to a square.)



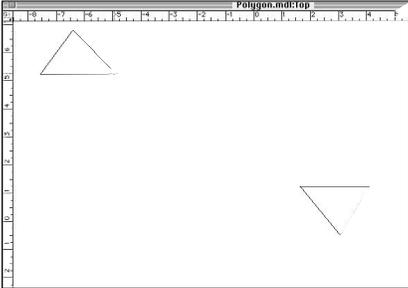
Note: The Polygon tool can be selected directly by typing "p."

Note: For Polygons with an odd number of sides, use the Control-Shift key combination to create sides of equal length.

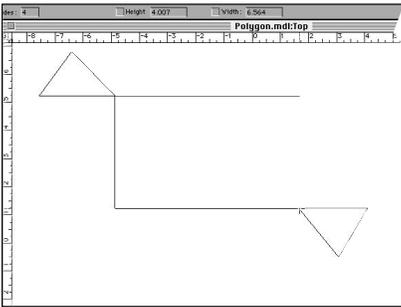
Using the Snap options with the Polygon tool

- 1) Select the Polygon tool from the tool palette (or press “p” on the keyboard).
- 2) Click on a snap option in the Snap palette or type the key for the corresponding snap option.
- 3) Press and hold the mouse very near the desired snap point.

Note: The important thing here is to press the mouse on the object that you are trying to snap to. Don't just be near the snap point. Be near, and be on the snap object.



Move to first snap point.



Drag to second snap point and release.

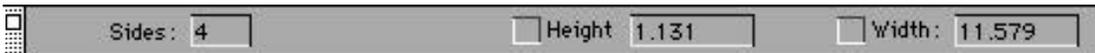
- 4) While dragging the mouse, take the cursor over to the Snap palette and release the mouse on the snap option you want. The indicated snap option on the palette will highlight and the polygon will remain attached to the mouse as if you were still holding the mouse button down. You can move the mouse around the screen and the polygon will follow; or, while dragging the mouse, press the key that corresponds to the snap option you want.
- 5) Still dragging the mouse (of course, if you released on the snap palette, the mouse button is not being held down), take it to the snap point where you want to end the polygon bounding rectangle. Be sure you are near the snap point, and on the snap object, and then release the mouse. The polygon will finish directly on that point.

Using the Tool Info palette to control a Polygon

The attributes that you can specify to define a polygon are: Number of Sides, Height, and Width.

By default, the value for “Sides” is 4. To change this, click in this field to activate it and type in a new value.

To specify a value for height and width of the bounding rectangle, put an “X” in the check box next to one or both of the fields. This tells ModelPro to pay attention to the value that is in the fields and to build the bounding rectangle according to those dimensions regardless of mouse actions. Type in values that you want. Then, press and hold the mouse to define the beginning point. As the mouse is dragged around, a polygon of the specified dimensions follows. The polygon is attached to the cursor by the lower left corner of the bounding rectangle.



Polygon Tool Info palette

Using Draw Options with the Polygon tool

The Draw Options portion of the Tool palette controls the manner in which the bounding rectangle of the Polygon is constructed. At the very bottom of the Tool palette there are three buttons with small arrows. These arrows dictate how the polygon is created relative to the two points that are used to define it. One of these three buttons will always be highlighted, thereby defining the method that is used for all new polygons.

The first button sets ModelPro so that the bounding rectangle will be created by dragging across its diagonal.

The middle button sets ModelPro so the the bounding rectangle will be created by dragging from one of the corners to its center.

The last button sets ModelPro so the the bounding rectangle will be created by dragging from its center to one of its corners.

One of these three methods may have advantages over the others in different situations, depending on surrounding geometry and the Snap Options that will work best for the current construction.

A polygon can be created as a hollow wireframe object or as a solid filled object. This is controlled by the “Filled/Open” button in the Draw Options portion of the Tool palette. If the little oval has a grey filled-in pattern, then the new polygon will be created as a filled object. If the oval is plain white, then the polygon will be open. To change from one mode to the other, simply click on the small oval button. It acts as a toggle, switching between Filled and Not-filled. The color that is selected in the Color indicator will be used for the new polygon. To pick a new color, simply press and hold the cursor on the Color indicator; a color palette pops up. You can then drag out onto the colors to select one. If you drag off the palette to the right, the color palette will “tear off,” allowing you to place it on the screen and leave it open.

These options affect how the polygon will be displayed in the Angled View window when the Quick-Render button is pressed. If it is a filled object, it will be displayed as a colored surface. If it is an open object, it won't display at all since wireframe edges are not shown in this mode. The open object will be invisible. At any time, you can change the "Filled/Open" attribute of a polygon by double-clicking on the object and clicking on the "Filled/Capped" option in the resulting dialog box.

Using Modifier keys with the Polygon tool

The Shift key can be used to constrain the bounding rectangle so that its width and height are the same, resulting in a proportional polygon. For example, a four-sided polygon will be constrained to a square.

All of the standard keyboard modifiers are in effect.



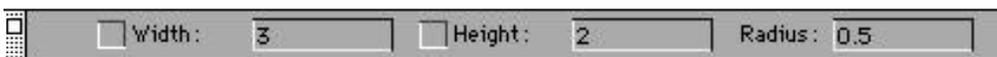
Note: The Rounded Rectangle Tool is on the Line pop-up.

The Rounded Rectangle Tool

The Rounded Rectangle Tool works exactly like the Polygon tool, except for two things: You can only make rectangles with it, and the corners of the rectangle are rounded.

Everything in the previous section about the Polygon tool applies for this tool as well, except for the part where you can define the number of sides. In this tool the number of sides is always four.

To control the radius of the rounded corners, use the Tool Info palette. The default radius is 0.5. Click in that field to activate it and type in a new value if you so desire.



Rounded Rectangle Tool Info palette

The Spline Mesh Tool

The Spline Mesh tool creates a rectangular spline surface that is planar. It is made up of a series of mesh lines with vertices at the intersections of the mesh lines. The Spline Mesh object is most often used as the starting point for a freeform surface. The original planar surface is created, and then the individual vertices are edited in 3-D space to create a new shape such as a terrain model or a warped surface.

A Spline Mesh object can be created initially only in the three orthogonal planes. It will be automatically be placed at the Depth location as defined by the Depth Origin and Depth Lines unless the Snap Options are used to snap the mesh to other objects already on the screen.

Using the Spline Mesh tool

- 1) Select the Spline Mesh tool from the tool palette.
- 2) Press and hold the mouse at a point to be the beginning point of the Spline Mesh.
- 3) Drag the mouse and release it at the point you want as the end point of the Spline Mesh. (Hold the Shift key to constrain the Spline Mesh to a square.)

Using the Snap options with the Spline Mesh tool

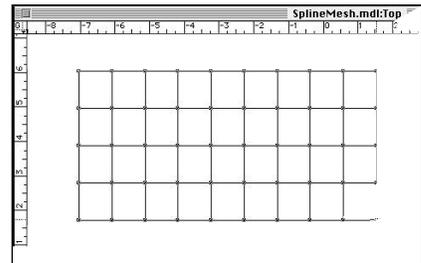
- 1) Select the Spline Mesh tool from the tool palette.
- 2) Click on a snap option in the Snap palette, or type the key for the corresponding snap option.
- 3) Press and hold the mouse very near the desired snap point to begin the Spline Mesh object.

Note: The important thing here is to press the mouse on the object that you are trying to snap to. Don't just be near the snap point. Be near, and be on the snap object.

- 4) While dragging the mouse, take the cursor over to the Snap palette and release the mouse on the snap option you want. The indicated snap option on the palette will highlight and the Spline Mesh will remain attached to the mouse as if you were still holding the mouse button down. You can move the mouse around the screen and the Spline Mesh will follow; or, while dragging the mouse, press the key that corresponds to the snap option you want.
- 5) Still dragging the mouse (if you released on the snap palette, the mouse button is not being held down), take it to the snap point where you want to end the Spline Mesh . Be sure you are near the snap point and on the snap object, and then release the mouse. The Spline Mesh will finish directly on that point.



Note: The Spline Mesh tool is on the line pop-up.



Spline Mesh

Using the Tool Info palette to control a Spline Mesh

The attributes of a Spline Mesh that can be controlled by the Tool Info palette are two. You can control the approximate size of each of the mesh cells, and you can control whether the mesh lines are Splines, or simply straight line segments.



Spline Mesh Tool Info palette

A Spline Mesh will try to maintain a constant mesh cell size as the mesh gets larger and smaller when the mouse is dragged in the window. The mesh will always have an integer number of square cells in each direction, (there will never be a partial-cell) and to do this the cells adjust larger and smaller within one-half the value for cell width that is called out in the Tool Info Palette. By default, the approximate cell size is set to 1 inch. Simply click in the field labeled “Spacing approximately [1 in] per Section” to activate it and type in the value you want. This label reflects the current units as well. If you switch to millimeters as the default units in the Edit/Preferences/Drawing dialog box, the Tool Info palette will now show 25.4 mm as the current spacing.

The other option in the Tool Info palette is a check box labeled “As Lines.” This is used to define whether the spline mesh is going to be created as a series of splines, or as a series of straight-line segments.

If the box is left unchecked (its default condition), then the mesh will consist of interconnected spline objects. If one or more vertices are selected and moved, then the surrounding mesh lines will distort in a smooth curvy manner to maintain a continuous curvature between all the vertices.

If you put an “X” in the “As Lines” check box, then the mesh will consist of interconnected straight-line objects. If one or more vertices are selected and moved, then the surrounding mesh lines will distort in a sharp-cornered manner. Each vertex acts as a “pin” for the four straight segments that join there. As a vertex moves, its four lines hinge freely, creating a sharp corner there and at the neighboring vertices.

Using Draw Options with the Spline Mesh tool

The Draw Options portion of the Tool palette controls the manner in which the Spline Mesh is constructed. At the very bottom of the Tool palette there are three buttons with small arrows. These arrows dictate how the Spline Mesh is created relative to the two points that are used to define it. One of these three buttons will always be highlighted, thereby defining the method that is used for all new polygons.

The first button sets ModelPro so that the Spline Mesh will be created by dragging across its diagonal.

The middle button sets ModelPro so the the Spline Mesh will be created by dragging from one of the corners, to its center.

The last button sets ModelPro so the the Spline Mesh will be created by dragging from its center, to one of its corners.

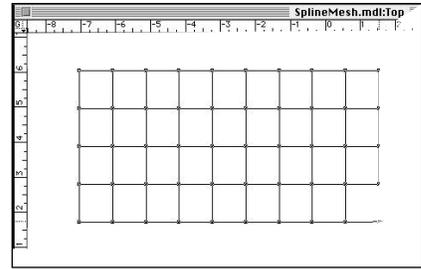
One of these three methods may have advantages over the others in different situations, depending on surrounding geometry and the Snap Options that will work best for the current construction.

The color that is selected in the Color indicator will be used for the new Spline Mesh. To pick a new color, simply press and hold the cursor on the Color indicator; a color palette pops up. You can then drag out onto the colors to select one. If you drag off the palette to the right, the color palette will “tear off,” allowing you to place it on the screen and leave it open.

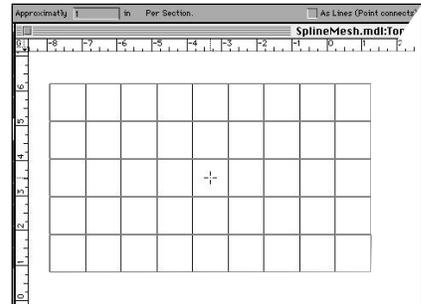
Using Modifier keys with the Spline Mesh tool

The Shift key can be used to constrain the rectangular mesh so that its width and height are the same, resulting in a square spline mesh.

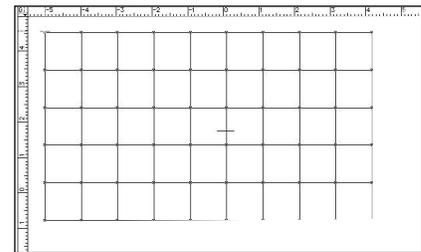
All of the standard keyboard modifiers are in effect.



Spline Mesh created by dragging across diagonal.



Spline Mesh created by dragging from corner to center.



Spline Mesh created by dragging from center to corner,



The Spline Tool

The Spline tool is used to place a freeform curve through any series of points through 3-D space. The curve can be made of curved segments and/or straight-line segments, all connected together in a continuous object.

When placing points that define the spline you are creating, you have the option of either clicking or dragging to establish the points. If you click at a location, the vertex that is defined there will have a sharp corner between the two spline segments. No control handles are created. If you press, hold, and drag the mouse, the vertex will have a smooth tangent joint between the two segments. As you drag, a pair of control handles are pulled off the vertex, with which you control the curvature of the two segments.

Using the Spline Tool

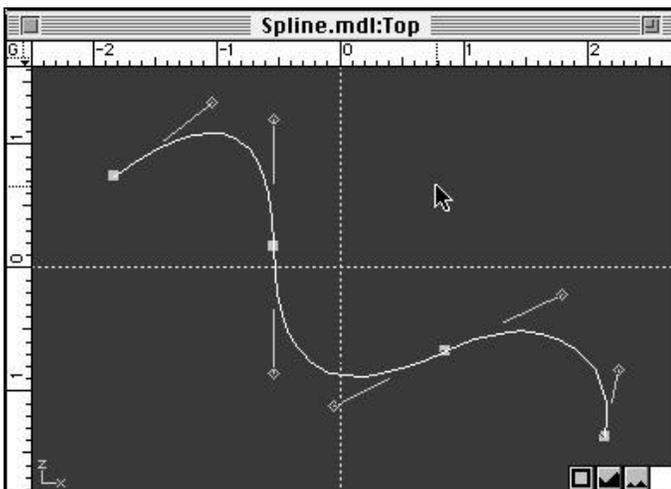
- 1) Select the Spline tool from the tool palette. (Or press the “b” key.)
- 2) **To create the first point of the spline**
Click at the desired location. A black vertex will appear to mark the point. There will be no Control Handles available at this point for later editing.

(Or press and hold the mouse at the desired location and drag away in the general direction that you want the spline to go.)

A black vertex will appear at the beginning point and a line will grow between this point and the cursor as you drag away. This line indicates the tangent direction that the spline will go away from the beginning point. There is a Control Handle available at the end of this line for later editing of the curvature of the spline segment leaving this point. Release the mouse.

- 3) **To create the next point of the spline**

Click at the next desired location. A new black vertex will appear and a spline segment will connect this point and the previous point. This vertex will have a “sharp” corner. There will be no Control Handles available at this point for later editing.



Spline with all control points selected.

If the previous spline point was a “click” point, then this point and the previous point will be connected with a straight spline segment.

If the previous spline point was a “drag” point, then the spline will start out curved at the previous point, and end up straight at this new point.

(Or press and hold the mouse at the desired location and drag away in the general direction that you want the spline to continue.)

Using the Snap options with the Spline tool

- 1) Select the Spline tool from the tool palette. (Or press the “b” key.)
- 2) Click on a snap option in the Snap palette.
or type the key for the corresponding snap option.
- 3) Click or drag the mouse from very near the desired snap point to begin the Spline object.

The important thing here is to put the mouse on the object that you are trying to snap to. Don’t just be near the snap point. Be near, and be on the snap object.

- 4) Repeat steps 2 and 3 for each spline point that you add where snap alignment is needed.

Creating successive joined multi-line objects

Use Vertex Snapping to continue drawing on an existing open spline path. When the new spline segments are finished, select both the old and new paths and use the Join Elements command under the Operations Menu to make a single spline path.

Using Modifier keys with the Spline tool

The Shift key can be used to constrain the tangency Control Lines to horizontal, vertical, and 45 degrees.

All of the standard keyboard modifiers are in effect.

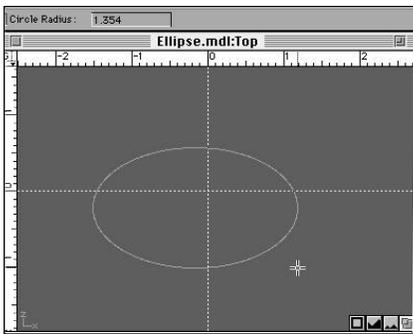


The Circle/Ellipse Tool

The Circle/Ellipse Tool creates planar circles and ellipses. In the case of a circle, the radius can be defined numerically with the Tool Info palette.

For the purposes of this discussion, this tool will be referred to as the Ellipse tool, since a circle is just a special form of an ellipse.

An ellipse can only be created in the three orthogonal planes. It will be automatically be placed at the Depth location as defined by the Depth Origin and Depth Lines unless the Snap Options are used to snap the ellipse to other objects already on the screen. It can be created normal at first, and then rotated as a separate operation using the Rotate tool.



Ellipse

Using the Ellipse tool

- 1) Select the Ellipse tool from the tool palette (or press “e” on the keyboard).
- 2) Press and hold the mouse at a point to be the beginning point of the ellipse bounding rectangle.
- 3) Drag the mouse and release it at the point you want as the end point of the bounding rectangle. (Hold the Shift key to constrain the ellipse to a circle.)

Using the Snap options with the Ellipse tool

- 1) Select the Ellipse tool from the tool palette (or press “e” on the keyboard).
- 2) Click on a snap option in the Snap palette.
or type the key for the corresponding snap option.
- 3) Press and hold the mouse very near the desired snap point.

The important thing here is to press the mouse on the object that you are trying to snap to. Don't just be near the snap point. Be near, and be on the snap object.

- 4) While dragging the mouse, take the cursor over to the Snap palette and release the mouse on the snap option you want. The indicated snap option on the palette will highlight and the ellipse will remain attached to the mouse as if you were still holding the mouse button down. You can move the mouse around the screen and the ellipse will follow.

(Or, while dragging the mouse, press the key that corresponds to the snap option you want. There are several other ways to accomplish this function; consult the appropriate reference section for more information.)

- 5) Still dragging the mouse (of course, if you released on the snap palette, the mouse button is not being held down), take it to the snap point where you want to end the ellipse bounding rectangle. Be sure you are near the snap point, and on the snap object, and then release the mouse. The ellipse bounding rectangle will finish directly on that point.

Using the Tool Info palette to control a Ellipse

The only attribute that you can specify in the Tool Info palette for the Ellipse tool is Radius, for defining the size of a circle.

By default, the value for “Radius” is 1. To change this, click in this field to activate it and type in a new value.

To activate the value for radius of the circle, put an “X” in the check box next to the field. This tells ModelPro to pay attention to the value that is in the field and to build the circle to that size regardless of mouse actions. Type in value that you want. Then, press and hold the mouse to define the beginning point. As the mouse is dragged around, a polygon of the specified dimensions follows. The polygon is either attached to the cursor by the lower left corner of the bounding rectangle or by the center of the circle depending on the settings of the Draw Options.

Using Draw Options with the Ellipse tool

The Draw Options portion of the Tool palette controls the manner in which the bounding rectangle of the Ellipse is constructed. At the very bottom of the Tool palette, there are three buttons with small arrows. These arrows dictate how the ellipse is created relative to the two points that are used to define it. One of these three buttons will always be highlighted, thereby defining the method that is used for all new ellipses.

The first button sets ModelPro so that the bounding rectangle will be created by dragging across its diagonal.

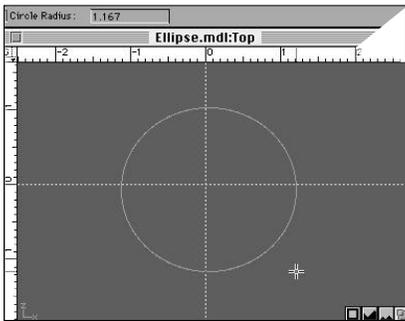
The middle button sets ModelPro so the the bounding rectangle will be created by dragging from one of the corners, to its center.

The last button sets ModelPro so the the bounding rectangle will be created by dragging from its center, to one of its corners.

One of these three methods may have advantages over the others in different situations, depending on surrounding geometry and the Snap Options that will work best for the current construction.

An Ellipse can be created as a hollow wireframe object or as a solid filled object. This is controlled by the “Filled/Open” button in the Draw Options portion of the Tool palette. If the little oval has a grey filled-in pattern, then the new ellipse will be created as a filled object. If the oval is plain white, then the ellipse will be open. To change from one mode to the other, simply click on the small oval button. It acts as a toggle, switching between Filled and Not-filled.

The color that is selected in the Color indicator will be used for the new ellipse. To pick a new color, simply press and hold the cursor on the Color indicator; a color palette pops up. You can then drag out onto the colors to select one. If you drag off the palette to the right, the color palette will “tear off”, allowing you to place it on the screen and leave it open.



Create circle from ellipse using Shift key.

Note: All of the standard keyboard modifiers are in effect.

These options affect how the ellipse will be displayed in the Angled View window when the Quick-Render button is pressed. If it is a filled object, it will be displayed as a colored surface. If it is an open object, it won't display at all since wireframe edges are not shown at all in this mode. The open object will be invisible. At any time, you can change the “Filled/Open” attribute of an ellipse by double-clicking on the object and clicking on the “Filled/Capped” option in the resulting dialog box.

Using Modifier keys with the Ellipse tool

The Shift key can be used to constrain the bounding rectangle so that its width and height are the same, resulting in a Circle.

The 90° Arc Tool

The 90° Arc Tool creates planar elliptical arcs that go through 90 degrees of rotation. That is, they always begin and end in a vertical or horizontal direction. The radius of the arc can be defined numerically in the Tool Info palette.

When using the 90° Arc tool, each new arc that is created will be formed in a “clockwise” direction. If you need one that goes in a counter-clockwise direction, hold the Control key while drawing.

A 90° Arc can only be created in the three orthogonal planes. It will be automatically be placed at the Depth location as defined by the Depth Origin and Depth Lines unless the Snap Options are used to snap the arc to other objects already on the screen. It can be created normal at first, and then rotated as a separate operation using the Rotate tool.

Using the 90° Arc tool

- 1) Select the 90° Arc tool from the tool palette (or press “a” on the keyboard).
- 2) Press and hold the mouse at a point to be the beginning point of the arc bounding rectangle.
- 3) Drag the mouse and release it at the point you want as the end point of the bounding rectangle. (Hold the Shift key to constrain the ellipse to a circle.)

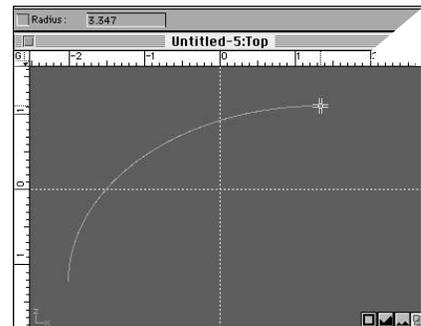
Using the Snap options with the 90° Arc tool

- 1) Select the 90° Arc tool from the tool palette (or press “a” on the keyboard).
- 2) Click on a snap option in the Snap palette.
or type the key for the corresponding snap option.
- 3) Press and hold the mouse very near the desired snap point.

The important thing here is to press the mouse on the object that you are trying to snap to. Don't just be near the snap point. Be near, and be on the snap object.



Note: The 90° Arc tool is on the Ellipse tool pop-up.



Draw spline arc.

Note: Holding the Shift key constrains the elliptical arc into a circular arc.

- 4) While dragging the mouse, take the cursor over to the Snap palette and release the mouse on the snap option you want. The indicated snap option on the palette will highlight and the arc will remain attached to the mouse as if you were still holding the mouse button down. You can move the mouse around the screen and the arc will follow.

(Or, while dragging the mouse, press the key that corresponds to the snap option you want.)

- 5) Still dragging the mouse (if you released on the snap palette, the mouse button is not being held down), take it to the snap point where you want to end the arc bounding rectangle. Be sure you are near the snap point, and on the snap object, and then release the mouse. The arc will finish directly on that point.

Using the Tool Info palette to control a 90° Arc

The only attribute that you can specify in the Tool Info palette for the 90° Arc tool is Radius, for defining the size of a circular arc.

By default, the value for “Radius” is 1. To change this, click in this field to activate it and type in a new value.

To activate the value for radius of the arc, put an “X” in the check box next to the field. This tells ModelPro to pay attention to the value that is in the field and to build the arc to that size regardless of mouse actions. Type in value that you want. Then, press and hold the mouse to define the beginning point. As the mouse is dragged around, a 90° Arc of the specified size is anchored at the beginning point and follows the cursor in 90 degree rotational increments.

Creating successive joined multi-line objects

As soon as a 90° Arc object is drawn, notice that the ending point is still highlighted with a solid black square. This is telling you that ModelPro is ready for you to add on to this arc with another 90° Arc. If you choose, the Line tool or the Spline tool can be used instead. If you immediately begin a new line, spline, or arc segment by clicking within five pixels of the 90° Arc ending point, the new object will automatically be joined to the 90° Arc. You don't need to use the Vertex-snap to accomplish this. The two objects are connected into a single multi-line object.

Using Modifier keys with the 90° Arc tool

The Shift key can be used to constrain the bounding rectangle of the 90° Arc so that its width and height are the same, resulting in a Circular arc.

The Control key can be used to draw an arc counter-clockwise.

All of the standard keyboard modifiers are in effect.

The Circular Arc Tool

The Circular Arc Tool creates planar circular arcs of any radius that go through any amount of angular rotation. The starting point and ending point of the arc can be at any angular location around the center point. The user can choose whether to begin the arc from the center or from a radius point, by selecting the appropriate option on the Tool Info palette.

When using the Circular Arc tool, each new arc that is created can be formed either in a “clockwise” or “counter-clockwise” direction. It is determined by which direction you drag it when you create it.

A Circular Arc can only be created in the three orthogonal planes. It will be automatically be placed at the Depth location as defined by the Depth Origin and Depth Lines unless the Snap Options are used to snap the arc to other objects already on the screen. It can be created normal at first, and then rotated as a separate operation using the Rotate tool.

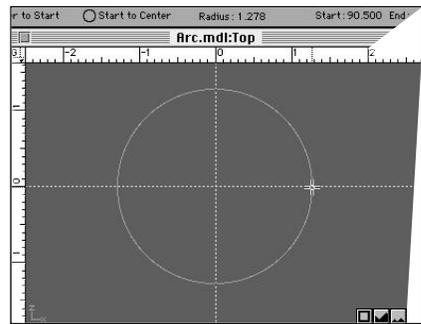
Using the Circular Arc tool

With the “Center to Start” option picked in the Tool Info palette

- 1) Select the Circular Arc tool from the tool palette.
- 2) Press and hold the mouse at a point to be the center of the arc.
- 3) Drag the mouse away from the center point the distance that you want to be the radius of the arc, and release it at the point where you want the arc to start. (Hold the Shift key to constrain the angle of the beginning point to be a multiple of 45°.)
- 4) Move the mouse away from the beginning point in the direction you want the arc to go, and click the mouse button at the point where you want the arc to end. (Hold the Shift key to constrain the delta angle of the arc to be a multiple of 45°.)



Note: The Circular Arc tool is on the Ellipse tool pop-up.



Select arc size.



Draw arc.

Using the Snap options with the Circular Arc tool

Before defining each of the three points that make up the Circular Arc, click on a snap option in the Snap palette, or type the key for the corresponding snap option to activate the snap mode. Then click on the snap point and the arc will snap to that point.

If you are dragging the mouse and you want to activate a snap mode, take the cursor over to the Snap palette and release the mouse on the snap option you want. The indicated snap option on the palette will highlight and the cursor will remain attached to the arc as if you were still holding the mouse button down. You can move the mouse around the screen and the arc will follow. (Or, you can just press the key that corresponds to the snap option you want.) Then you can take the cursor over to the snap point you want and release the mouse there. The arc will snap right to that point.

Using the Tool Info palette to control a Circular Arc

The Tool Info palette lets you see the values for Radius, Start angle and End angle for the arc you are creating, but you cannot edit these values. Having them displayed here is useful to confirm that the arc is being created as you want. (For example, you can see that the arc is snapping to increments of 45° when you hold the shift key down.)

Creating successive joined multi-line objects

As soon as a Circular Arc object is drawn, notice that the ending point is still highlighted with a solid black square. This is telling you that ModelPro is ready for you to add on to this arc with another arc or line or spline. If you immediately begin a new line, spline, or arc segment by clicking within five pixels of the Circular Arc ending point, the new object will automatically be joined to the Circular Arc. You don't need to use the Vertex-snap to accomplish this. The two objects are connected into a single multi-line object.

Using Modifier keys with the Circular Arc tool

The Shift key can be used to constrain the beginning and ending angles of the Circular Arc to increments of 45°.

All of the standard keyboard modifiers are in effect.

The Text Tool

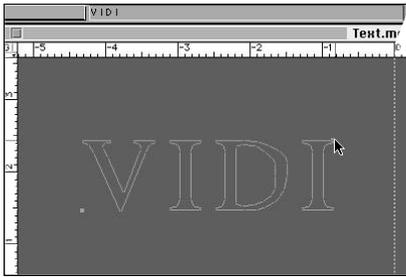


The text tool is used to import the outline paths of TrueType fonts into the ModelPro work space for the purposes of creating 3-D modeled objects of user-defined character strings.

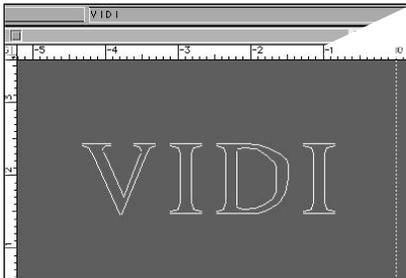
The text is brought in and converted to a series of multi-line objects. A series of hierarchical group folders is automatically created in the Group palette to contain the text objects. Each letter gets its own group, or its own folder if it is made up of more than one spline.

Only a single line of text can be entered at a time.

A Text object can only be created in the three orthogonal planes. It will automatically be placed at the Depth location as defined by the Depth Origin and Depth Lines. It can be created normal at first, and then rotated as a separate operation using the Rotate tool. The Snap Options have no effect when creating with the Text tool. Once the text is placed, it becomes a series of multi-line objects, and the Snap Options behave as normal with the spline vertices of the characters.



Text placed and resized.



Text "engaged".

Hint: wider text spacing helps if the text is to be beveled.

Using the Text tool

- 1) Select the Text tool from the tool palette. As soon as you do so, the cursor changes to a "T" with an arrow, indicating that you should pick the point where you want the text to start, and in which view window. Also, the Tool Info palette changes to show a pop-up menu for choosing which font should be used, and a data field for typing in the character string that you want.
- 2) Press and hold on the pop-up menu in the Tool Info palette to choose the font you would like to use.
- 3) Type in the character string you want. There is room for 50 characters in the data field.
- 4) Click in the orthogonal view window of your choice to indicate where the text should appear. A preview of the text string appears. You can resize the bounding rectangle by grabbing and moving one of it's corners with the cursor. The text inside the rectangle will be re-sized and re-proportioned accordingly. (To ensure that the text is proportioned correctly for that font, hold down the Shift key when re-sizing the bounding rectangle.)
- 5) When the text looks the way you want, hit the "Engage" button in the Tool Info palette. The text string will be converted to spline objects and will be placed in the work space just as the preview showed.

Using the Tool Info palette to control a Text string

The only setting you can change in the Tool Info palette besides the Font is a small slider next to the "Engage" button. This slider sets the horizontal spacing and kerning for the characters. When the slider is all the way to the left, the characters are "fatter" horizontally, and the spacing between letters is at a minimum. When the slider is all the way to the right, the characters are "skinnier" horizontally, and the spacing between letter is at a maximum. This slider does not affect the vertical scaling of the characters.

Using Draw Options with the Text tool

At the very bottom of the Tool palette there are three buttons with small arrows. These arrows control the manner in which the text is placed on the screen. One of these three buttons will always be active, thereby defining the method that is used for all new text.

The first button sets ModelPro so that the text will be created such that your click point defines the left edge of the text.

The middle button sets ModelPro so that your click point will define the center of the text object.

The last button also sets ModelPro so that your click point will define the center of the text object.

The text can be created as a hollow wireframe object, or as a solid filled object. This is controlled by the “Filled/Open” button in the Draw Options portion of the Tool palette. If the little oval has a grey filled-in pattern, then the new text will be created as a filled object. If the oval is plain white, then the text will be open. To change from one mode to the other, simply click on the small oval button. It acts as a toggle, switching between Filled and Not-filled.

The color that is selected in the Color indicator will be used for the new text. To pick a new color, simply press and hold the cursor on the Color indicator; a color palette pops up. You can then drag out onto the colors to select one. If you drag off the palette to the right, the color palette will “tear off,” allowing you to place it on the screen and leave it open.

These options affects how the text will be displayed in the Angled View window when the Quick-Render button is pressed. If it is a filled object, it will be displayed as a colored surface. If it is an open object, it won't display at all since wireframe edges are not shown at all in this mode. The open object will be invisible.



The 3-D primitives.

The 3-D Primitives

The 3-D Primitives are all special objects in ModelPro. They are special because they are the only objects that are not made up of either multi-lines or a surface mesh. Instead, the 3-D Primitives are mathematically simple geometric forms that are surfaced. They are much simpler than if they were made from multi-lines, therefore they take up less space in the file.

Each of these objects is created by dragging out its shape in the orthogonal view windows. When they are created, they originate on the current Depth lines.

A 3-D Primitive object does not have individual vertices. It only has bounding rectangle control points. This is the way that the objects are kept simple. The only editing possible is done by editing their overall size by moving their bounding box corners. Individual points cannot be changed since they don't exist.

The good news is that any 3-D Primitive (except a cube) can be converted to a spline mesh object simply by double-clicking on it and hitting the "Convert to Spline" button in the resulting dialog box. The details about the sizing of each of these objects can be controlled from this same dialog box.

This sizing-information dialog box is important because when the object is first created, ModelPro is forced to make an assumption about the depth of the object since you are dragging out its 3-D shape in a 2-D window. It is likely that the sizing of the object is initially not what you want. So defining the sizing you do want is made easy in this dialog box.

Using Draw Options with the 3-D Primitives

The Draw Options portion of the Tool palette controls the manner in which the bounding box of each object is constructed. At the very bottom of the Tool palette there are three buttons with small arrows. These arrows dictate how the object is created relative to the two points that are used to define it. One of these three buttons will always be highlighted, thereby defining the method that is used for all new objects.

The first button sets ModelPro so that the bounding box will be created from corner-to-corner.

The middle button sets ModelPro so the the bounding box will be created from one of the corners, to the object center.

The last button sets ModelPro so the the bounding rectangle will be created from the center of the object, to one of its corners.

One of these three methods may have advantages over the others in different situations, depending on surrounding geometry and the Snap Options that will work best for the current construction.

An 3-D Primitive can only be created as a surfaced object. It cannot be drawn as a hollow wireframe object. Therefore, 3-D Primitives ignore the “Filled/Open” button in the Draw Options portion of the Tool palette.

The color that is selected in the Color indicator will be used for the new object. To pick a new color, simply press and hold the cursor on the Color indicator; a color palette pops up. You can then drag out onto the colors to select one. If you drag off the palette to the right, the color palette will “tear off”, allowing you to place it on the screen and leave it open.



Cube

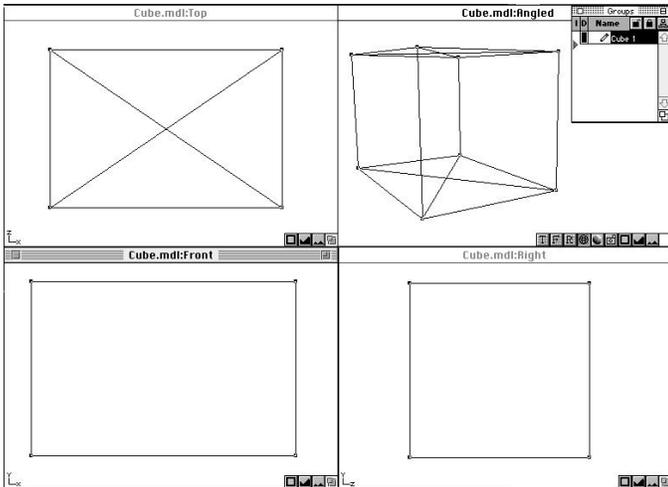
This tool creates a rectangular prism. When you drag out a rectangle in one of the three orthogonal views to define the object, ModelPro automatically defines the depth of the object in the other

windows. The object is forced to have a square shape in one of the other windows, which gives all the additional information needed to define its size in the third window.

You can hold the Shift key to force the object to have the same size in all three dimensions. In other words, it will be an exact cube.

Using the Tool Info palette to control a Cube

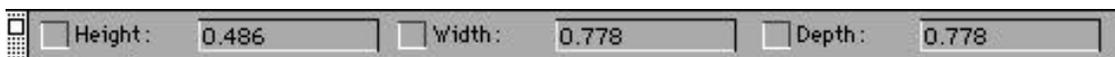
You can specify values for all three dimensions in the Tool Info palette. To enter values into these data fields, click in them to activate them (or hit the tab key until the box you want is



Cube

highlighted) and type in new values.

To activate the values for Height, Width or Depth, put an “X” in the check box next to one, two, or all of the fields. Whatever value is currently in each box you check will be frozen. This tells ModelPro to pay attention to those values and build the cube according to those numbers, regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a cube of the specified size is anchored at the beginning point and follows the cursor. If you only check one box, then only that dimension is frozen. The other two dimensions are defined by how you drag the mouse.



Cube Tool Info palette

Prism

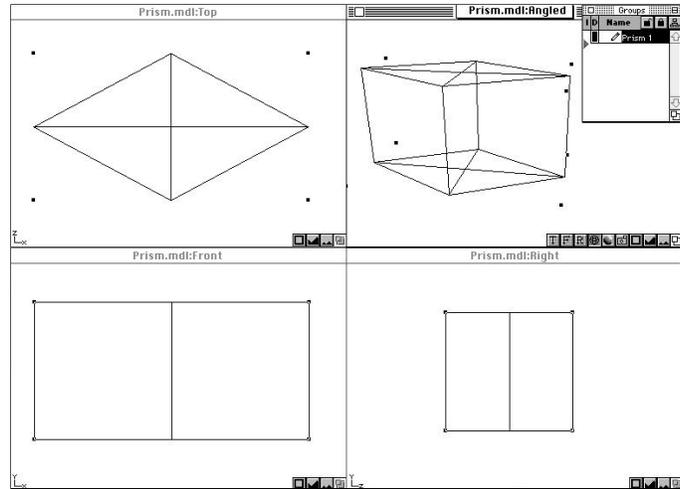


This tool creates a polygonal prism. When you drag out a polygon in one of the three orthogonal views to define the object, the face of the polygon is drawn in the plane of that view, and ModelPro automatically defines the depth of the object in the other windows. The object is forced to have a 2-to-1 width-to-height ratio in one of the other windows, which gives all the additional information needed to define its size in the third window.

You can hold the Shift key to force the object to have the same size for the width and height of the bounding box of the polygon. In other words, the 2-D face of the polygon will be proportional.

Using the Tool Info palette to control a Prism

You can define the number of sides for the polygon just by typing a value in the “Sides” data field.



Prism



Prism Tool Info palette

You can also specify values for all three dimensions in the Tool Info palette. To enter values into these data fields, click in them to activate them (or hit the tab key until the box you want is highlighted) and type in new values.

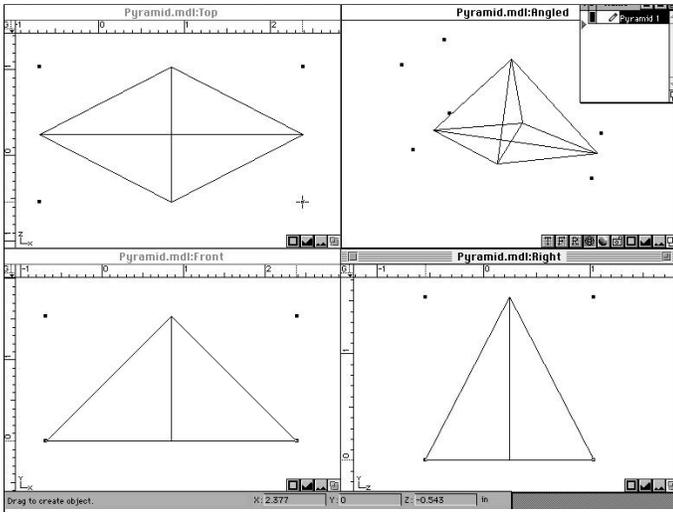
To activate the values for Height, Width or Depth, put an “X” in the check box next to one, two, or all of the fields. Whatever value is currently in each box you check will be frozen. This tells ModelPro to pay attention to those values and build the prism according to those numbers regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a prism of the specified size is anchored at the beginning point and follows the cursor. If you only check one box, then only that dimension is frozen. The other two dimensions are defined by how you drag the mouse.



Pyramid

This tool creates a polygonal pyramid. When you drag out a polygon in one of the three orthogonal views to define the object, the face of the polygon is drawn in the plane of that view, and the point of the pyramid is perpendicular to that face. ModelPro automatically defines the depth of the object in the other windows. The object is forced to have a 2-to-1 width-to-height ratio in one of the other windows, which gives all the additional information needed to define its size in the third window.

You can hold the Shift key to force the object to have the same size for the width and height of the bounding box of the polygon. In other words, the 2-D face of the polygon will be proportional.

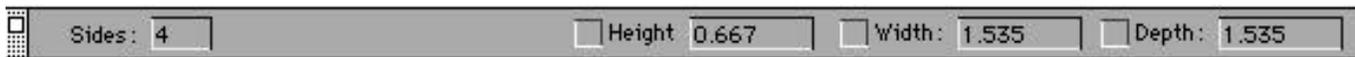


Pyramid

Using the Tool Info palette to control a Pyramid

You can define the number of sides for the polygon just by typing a value in the “Sides” data field.

You can also specify values for all three dimensions in the Tool Info palette. To enter values into these data fields, click in them to activate them (or hit the tab key until the box you want is highlighted) and type in new values.



Pyramid Tool Info palette

To activate the values for Height, Width or Depth, put an “X” in the check box next to one, two, or all of the fields. Whatever value is currently in each box you check will be frozen. This tells ModelPro to pay attention to those values and build the pyramid according to those numbers regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a pyramid of the specified size is anchored at the beginning point

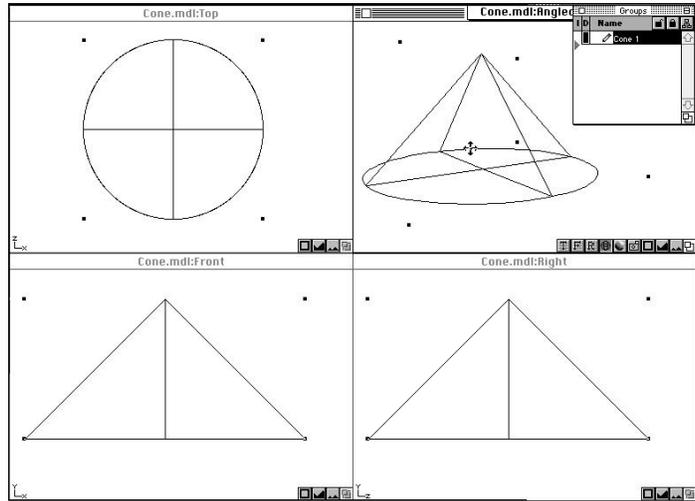
and follows the cursor. If you only check one box, then only that dimension is frozen. The other two dimensions are defined by how you drag the mouse.

Cone



This tool creates an elliptical cone. When you drag out an ellipse or circle in one of the three orthogonal views to define the object, the face of the ellipse is drawn in the plane of that view, and the point of the cone is perpendicular to that face. ModelPro automatically defines the depth of the object in the other windows. The object is forced to have a 2-to-1 width-to-height ratio in one of the other windows, which gives all the additional information needed to define its size in the third view window.

You can hold the Shift key to force the object to have the same size for the width and height of the bounding box of the ellipse. In other words, the 2-D face of the cone will be a circle.



Cone

Using the Tool Info palette to control a Cone

You can define the depth and the radius of the cone just by typing values into the appropriate data fields in the Tool Info palette. To enter values, click in the fields to activate them (or hit the tab key until the box you want is highlighted) and type in new values.

To activate the values for Depth or Radius, put an “X” in the check box next to one, or both of the fields. Whatever value is currently in each box you check will be frozen. This tells ModelPro to pay attention to those values and build the cone according to those numbers regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a cone of the specified size is anchored at the beginning point and follows the cursor. If you only check one box, then only that dimension is frozen. The other two dimensions are defined by how you drag the mouse.



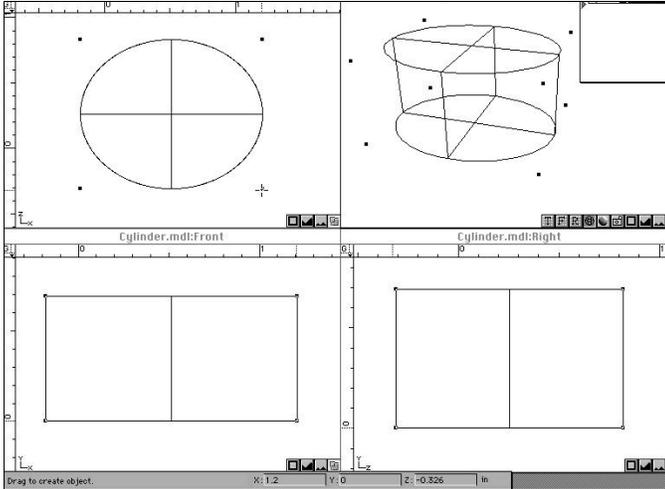
Cone Tool Info palette



Cylinder

This tool creates an elliptical cylinder. When you drag out an ellipse or circle in one of the three orthogonal views to define the object, the face of the ellipse is drawn in the plane of that view, and

cylinder comes out perpendicular to that face. ModelPro automatically defines the depth of the object in the other windows. The object is forced to have a 2-to-1 width-to-height ratio in one of the other windows, which gives all the additional information needed to define its size in the third window.



Cylinder

You can hold the Shift key to force the object to have the same size for the width and height of the bounding box of the ellipse. In other words, the 2-D face of the cylinder will be a circle.

Using the Tool Info palette to control a Cylinder

You can define the depth and the radius of the cylinder just by typing values into the appropriate data fields in the Tool Info palette. To enter values, click in the fields to activate them (or hit the tab key until the box you want is highlighted) and type in new values.



Cylinder Tool Info palette

To activate the values for Depth or Radius, put an “X” in the check box next to one, or both of the fields. Whatever value is currently in each box you check will be frozen. This tells ModelPro to pay attention to those values and build the cylinder according to those numbers regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a cylinder of the specified size is anchored at the beginning point and follows the cursor. If you only check one box, then only that dimension is frozen. The other two dimensions are defined by how you drag the mouse.

Sphere

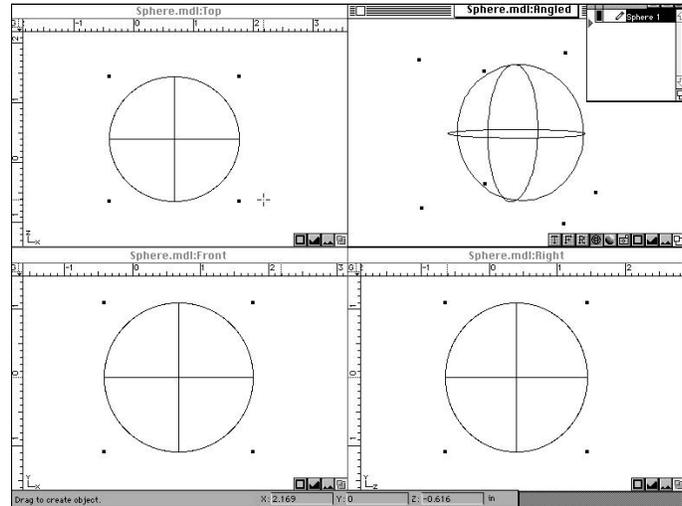


This tool creates both elliptical and circular spheres. When you drag out an ellipse in one of the three orthogonal views to define the object, ModelPro automatically defines the depth of the object in the other windows. The object is forced to have a circular shape in one of the other windows, which gives all the additional information needed to define its size and shape in the third window.

You can hold the Shift key to force the object to have the same size in all three dimensions. In other words, it will be an exact sphere.

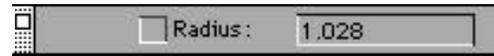
Using the Tool Info palette to control a Sphere

You can specify a value for the radius of the sphere in the Tool Info palette. To enter values into this data field, click in it to activate it (or hit the tab key until the data field is highlighted) and type in a new value.



Sphere

To activate the value for Radius, put an “X” in the check box next to the field. Whatever value is currently in the box after you check it will be frozen. This tells ModelPro to pay attention to that value and build the sphere according to that number regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a sphere of the specified size is anchored at the beginning point and follows the cursor.

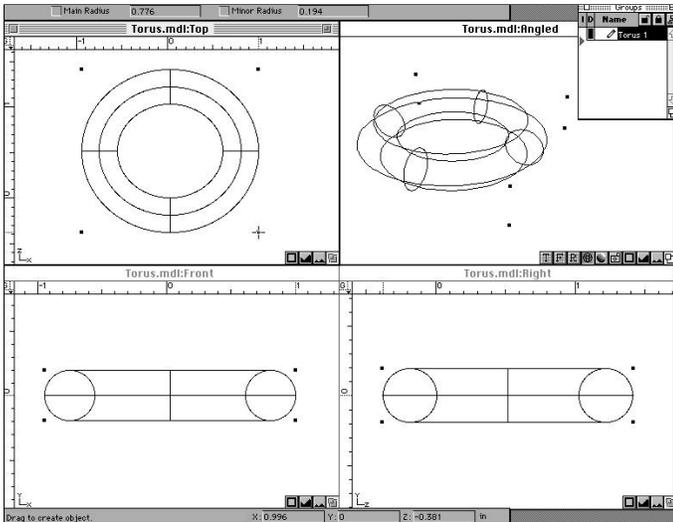


Sphere Tool Info palette



Torus

This tool creates a doughnut shaped object. The torus itself can be circular or elliptical, but the shape of a section cut through the object is always circular. When you drag out an ellipse or circle in one of the three orthogonal views to define the main radius of the object, ModelPro automatically defines the depth of the object in the other windows by making the main radius nine times bigger than the minor radius. This gives all the additional information needed to define its size in the third view window.



Torus

You can hold the Shift key to force the main radius of the torus to be circular.

Using the Tool Info palette to control a Torus

You can specify values for both the Main Radius and the Minor Radius in the Tool Info palette. The Main Radius is measured in a Horizontal

direction across the view window in which you begin the creation of the torus. (The Main Radius is not necessarily the largest dimension across the torus.) To enter values into these data fields, click in them to activate them (or hit the tab key until the box you want is highlighted) and type in new values.



Torus Tool Info palette

To activate the values for Main Radius and Minor Radius, put an “X” in the check box next to one, or both of the fields. Whatever value is currently in each box you check will be frozen. This tells ModelPro to pay attention to those values and build the torus according to those numbers regardless of how you drag the mouse. Then, when you press and hold the mouse to define the beginning point and drag the mouse around, a torus of the specified size is anchored at the beginning point and follows the cursor. If you only check one box, then only that dimension is frozen. The other dimension is defined by how you drag the mouse.