

## Parametric Modeling with PTC's ProDesktop 2000i<sup>2</sup>

This tutorial is designed to introduce the 3D solid modeling program ProDesktop (version 2000i<sup>2</sup>) to users who have little to no CAD experience.

### ***ProDesktop***

ProDesktop (Pro-D) is a parametric modeling program. Similar products on the market include AutoDesk Inventor, SolidWorks, and Pro Engineer. Pro-D is a lighter version of Pro Engineer.

This tutorial will walk the user through the creation of the 3 hole part from the BEES (BattleBots Educational Engineering System) kit. Creating a part in a parametric modeling program involves a few basic operations, not all of which might be applicable to your particular model. Some of these operations will be explained in detail later.

- Creating a workplane/sketchplane
- Creating a sketch
  - Construction geometry
  - Relations/Constraints
  - Dimensions
- Extruding a sketch
  - Solid
  - Thin (sheet metal)
- Cutting with a sketch
  - Blind
  - Up to Next
- Revolving a sketch
  - Centerlines
- Cutting with a revolved sketch
- Creating fillets and chamfers
- Shell operation

These operations form the basis for most parts that you will need to design and draw, and have similar names throughout most parametric programs.

### ***Assemblies***

Parametric programs also let the user create assemblies from the parts that they have drawn or downloaded. Using models of the BEES kit parts, you can design, assemble, and experiment with your robot designs without ever picking up a single piece of metal. By creating a library of parts, you can try new, complex designs without the worry of spending a large amount of time on something that might not work. Computer modeling reduces risk when creating a new design.



## Open Pro-D

The first thing to do is to create a blank file. You can do this by clicking the **New...** button, or by selecting **New** from the **File** menu, or by pressing **Ctrl + N** (Control and N together).

Pro-D will open a dialog box asking what kind of file you want to create. Select **Design** and click **OK**.

Maximize the new drawing window, and maximize Pro-D.

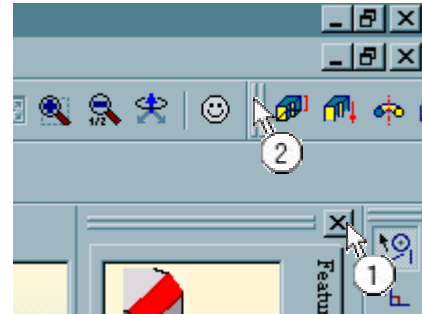


## Setup



If this is your first time using Pro-D, we recommend that you change some things around in the interface, to help reduce confusion. If these settings have already been changed on the computer you are using, you don't need to follow this step. We recommend that you

- Close the Palette on the right side of the screen by choosing **Palette** from the **Tools** menu, or by clicking the **Close** box in the Palette (1).
- Move the **Feature** toolbar (2) from the top right corner of the screen (where it is being cut off) to somewhere in the center of the screen, then move it again to a good location in the second row of toolbars.
- Change the default units from centimeters to inches. Choose **Options...** from the **Tools** menu. Switch to the **Units** tab. Change both **Model distances** and **Paper distances** to **Inches (in)**. Click **OK**.




## Starting a sketch

Click **View Onto Workplane** and the view will rotate so that the sketch plane is facing you.



A sketch plane is required for each new operation within the part that you are drawing. Pro-D automatically created this sketch plane (called initial) when we created the file.

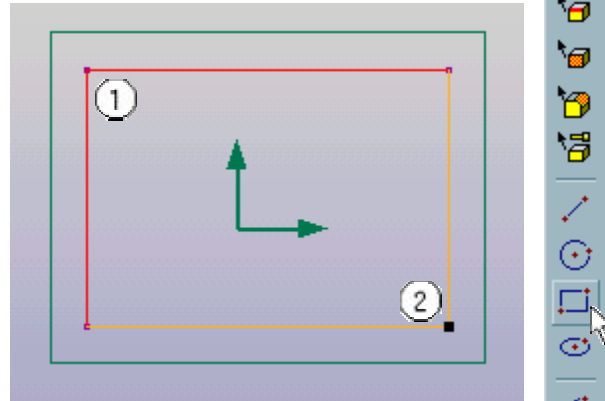
Click  **Rectangle** on the **Design** toolbar. Starting from a where you want one corner of the rectangle to be located (1), Click and Hold the mouse button to start the rectangle on the workplane. Move the mouse (drag) to the location of the opposite corner of the rectangle (2), and then release the mouse button.





The size of the rectangle doesn't really matter. We will be change the size of the rectangle in a later step.



The sketch plane will automatically resize to be just larger than the largest part of your sketch.

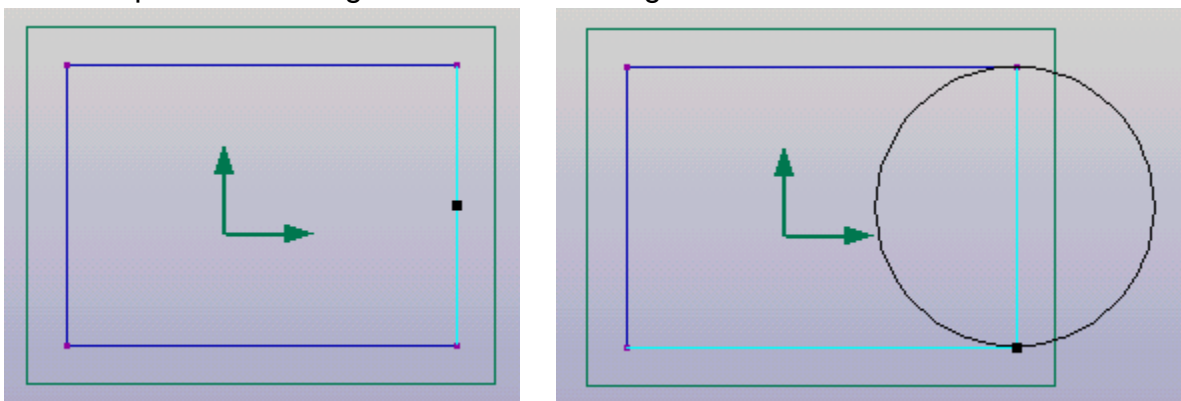



Click  **Circle** on the **Design** toolbar. As you might be able to guess from the icon, the center of the circle is clicked, and then the radius is dragged out to finish the circle. While we are drawing the circle, we can tell Pro-D a little about how different parts of the geometry relate to one another.

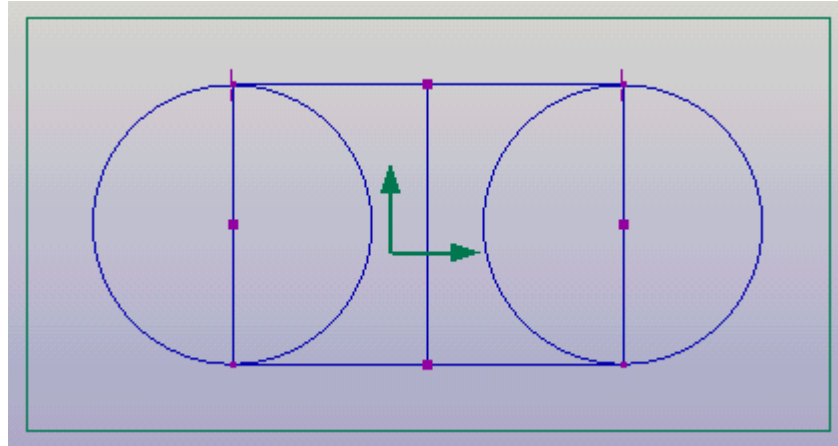
For example, using the  **Circle** tool, hold your mouse pointer over one of the lines of your rectangle. After a short time, the line turns light blue, and draws a black dot on the line. This tells you that the circle, if drawn from that point, would be attached to the line by the circles center point. This is called a *constraint*. If you hold your mouse over an endpoint or midpoint of a line, the black dot gets bigger. Now we are adding two constraints: The point (which can be any part of any sketch tool) will be constrained to the line, and also constrained to the endpoint (or midpoint) of that line.


If we move part of the geometry later, the rest of it will update to keep consistent with the constraints of the drawing. This is the power of parametric drawing. Geometry can be (and should be) *fully constrained* when you are done with the part. It is possible to *over-constrain* a sketch, meaning that Pro-D can't find a solution with the parameters you have given, usually because you have given it conflicting parameters.

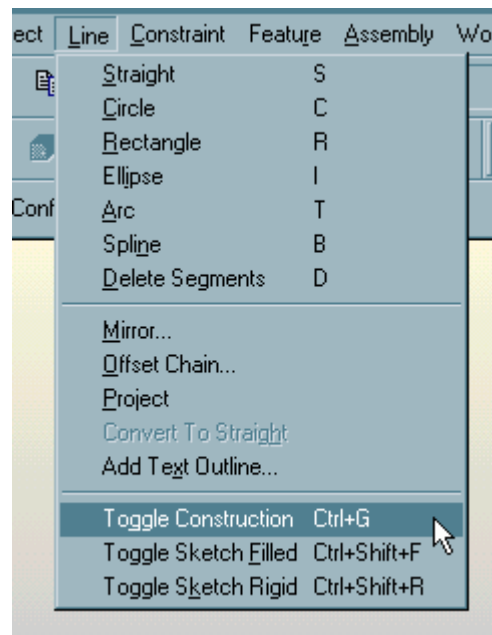
Draw a circle from the midpoint of the right side of the rectangle to one of the endpoints on the right side of the rectangle.




Also draw a circle on the other side of the rectangle, using the same constraints. Click the  **Line** tool on the **Design** toolbar. Draw a line from the midpoint of the top of the rectangle to the midpoint of the bottom of the rectangle.

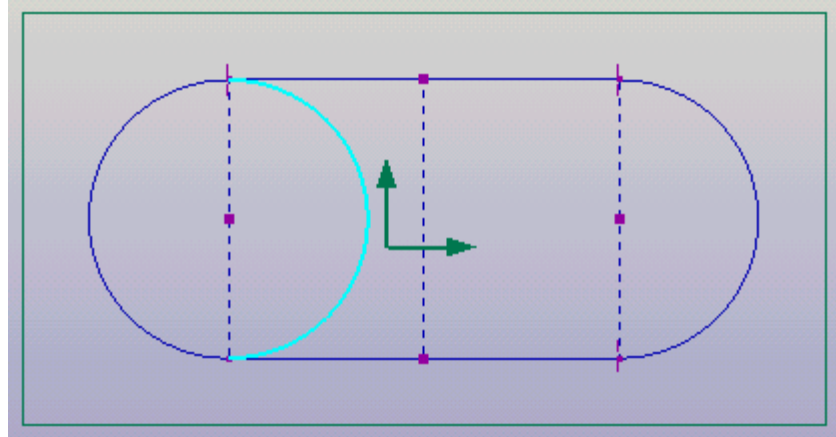





Using the  **Select Lines** tool, click one of the three vertical lines. While holding Shift, select the other two vertical lines. Change the lines into construction lines by using **Toggle Construction** from the **Lines** menu. The shortcut key for **Toggle Construction** is **Ctrl + G**. A construction line can be used for creating relations with geometry that are not visible on the actual part when it is completed. For our part, we will be using the three vertical lines to help us find the locations of the holes.




You can make circles into construction circles using the same method. The resulting circle can be used for geometry that can be helpful with bolt hole circles and other, more complex geometry.

Now click  **Delete Line Segments** from the **Design** toolbar. This tool is used to trim parts of arcs and lines from the sketch. Click on the two inner semicircles, but not the vertical lines. The resulting sketch is beginning to look like our part.



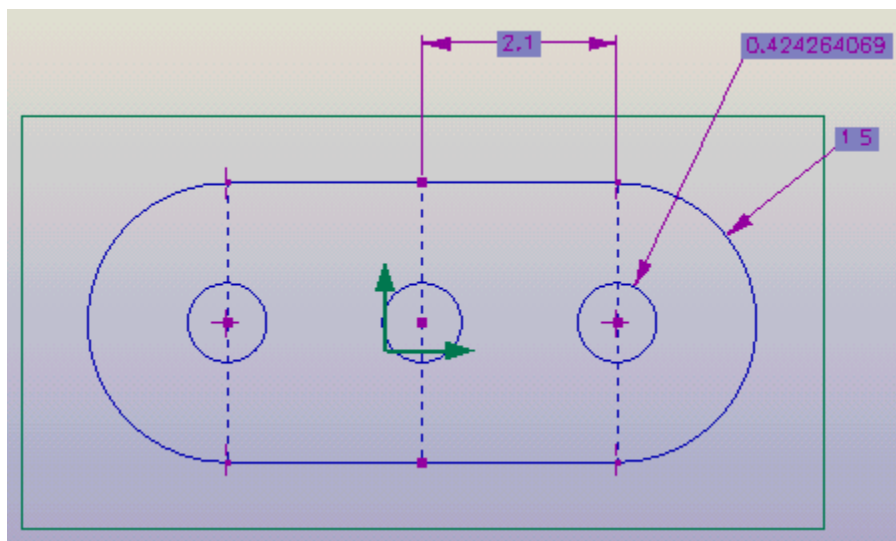
Now we will add the 3 holes to the part. Using the  **Circle** tool, draw one small circle from the midpoints of each of the three vertical construction lines. Use the  **Select Lines** tool and the Shift key to select only the three small circles. Choose  **Equal Radius** from the **Constraints** toolbar, or from the **Constraints** Menu. All three circles should change to be the same size.



## Dimensioning

The sketch is now complete, as far as geometry goes, but it is probably the wrong size. Now we will add dimensions to the sketch to complete the constraints. Click  **Sketch Dimension** on the design toolbar. Click on one of the outside semi-circles and drag the dimension to a location outside the sketch. Also dimension one of the holes, and the hole spacing. We will change the dimensions in the next step.

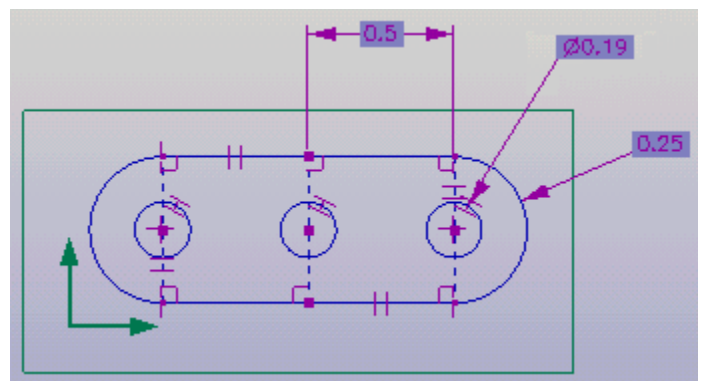


To dimension between two objects, click one object, then click and drag the dimension from the second object. Dimension the hole spacing by choosing one construction line, and then the other. Dimension a hole by clicking the circle once, then clicking again and dragging the dimension out. This creates a dimension of the diameter, instead of the radius




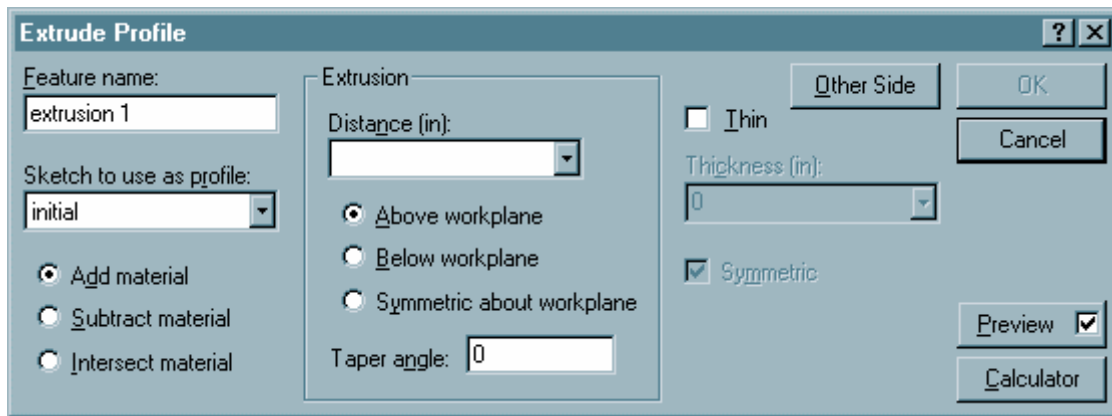
Click  **Select Constraints** from the Design toolbar. Double-click each of the dimensions and change them to the following values in the length field. Clicking once on the dimensions then clicking again and dragging will allow you to move the dimensions closer to the (now smaller) sketch. The  **Autoscale** tool will zoom the sketch out so that you can see all of the geometry and dimensions.




Part of Sketch	Enter this value
Small circles	.19" (Diameter)
Large circles	.25" (Radius)
Hole spacing	.5"

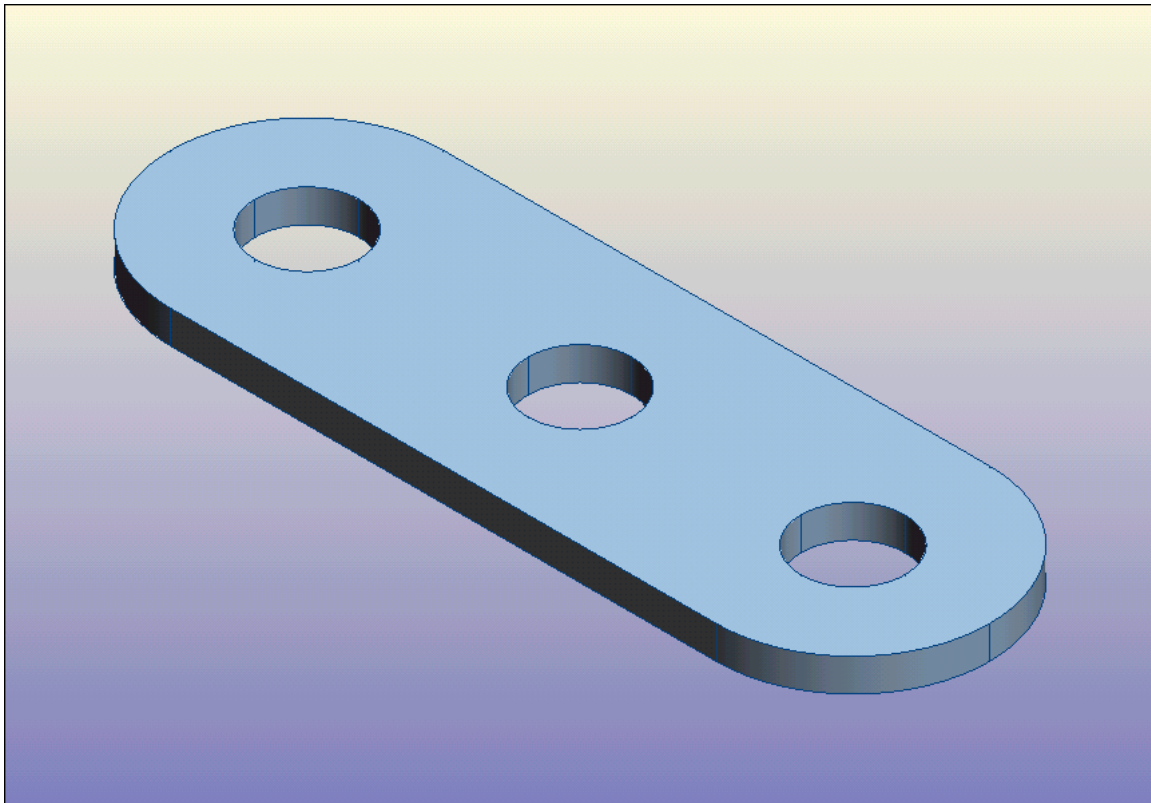


## Creating the solid part

On the **Features** toolbar (the toolbar you repositioned earlier), click  **Extrude Profile**. This opens the Extrude Profile dialog box, shown below.



You can change the name of the feature (useful for complex parts) and the distance of extrusion, which in our case will be the thickness of the part (0.06"). Click OK. Finally, click  **Select Faces**,  **View Isometric**, and  **Autoscale** on the Views toolbar to look at your finished part.



## Save your part

Pro-D uses the ".des" file format for parts.  **Save** your file.