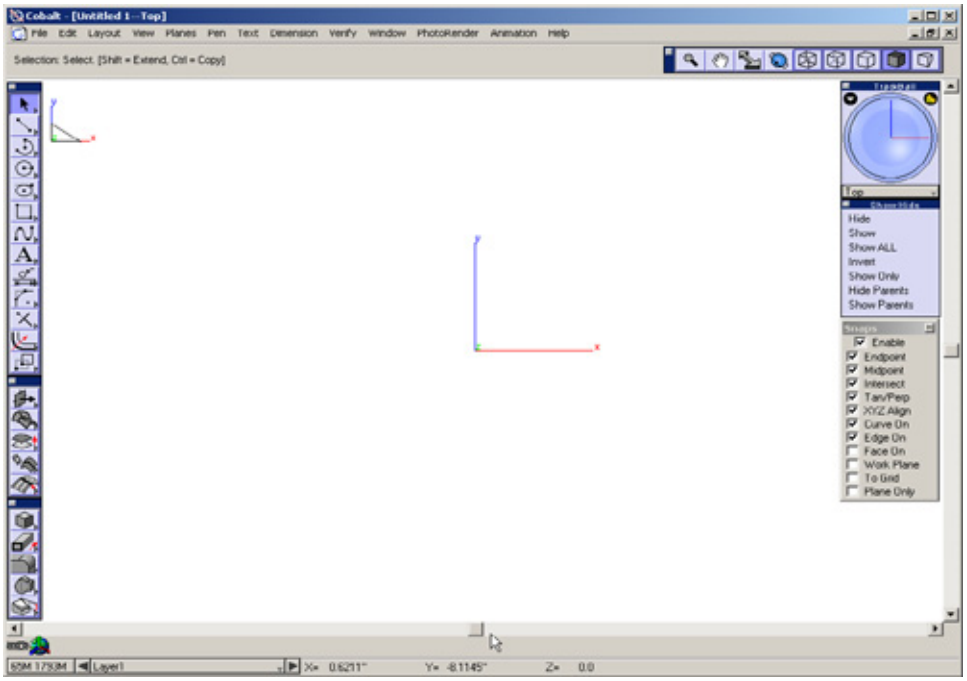
Modeling a Threaded Hex Bolt

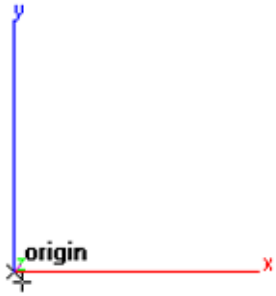
This tutorial describes one method for modeling a threaded hex bolt. To begin open Cobalt and organize the screen so that it looks similar to the image below. I have the opened the **Show-Hide** and the **Snaps** windows, the rest of the tool palettes that you see are opened by default when the application is launched. All of these tools are accessible through ***Window*** menu. We will start in the **Top** View.



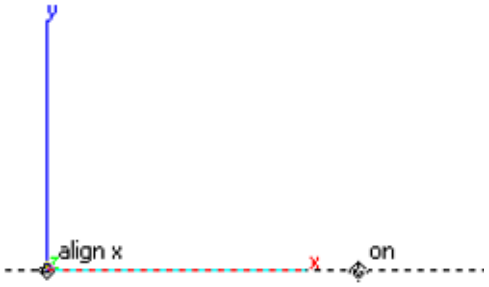
First, we will create the thread of the bolt. This is done by sweeping a triangular profile along a path, a helix.

To create the helix for the profile to be extruded along, select the ***Helix*** tool from the tools palette.

Place the first point at the origin.

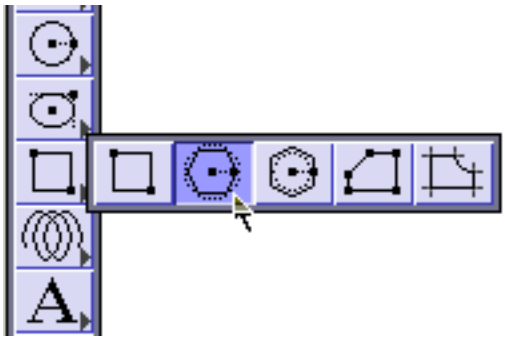


Place the second point to the right along the x-axis. Use the drafting assistant to keep the point aligned.

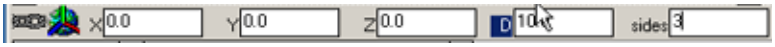


Once you have entered those two points. Type ***.5*** into the diameter field and ***2*** into the length field and hit ***Enter***. This will give us the thread we need for a 1/2" diameter, 2" long bolt.

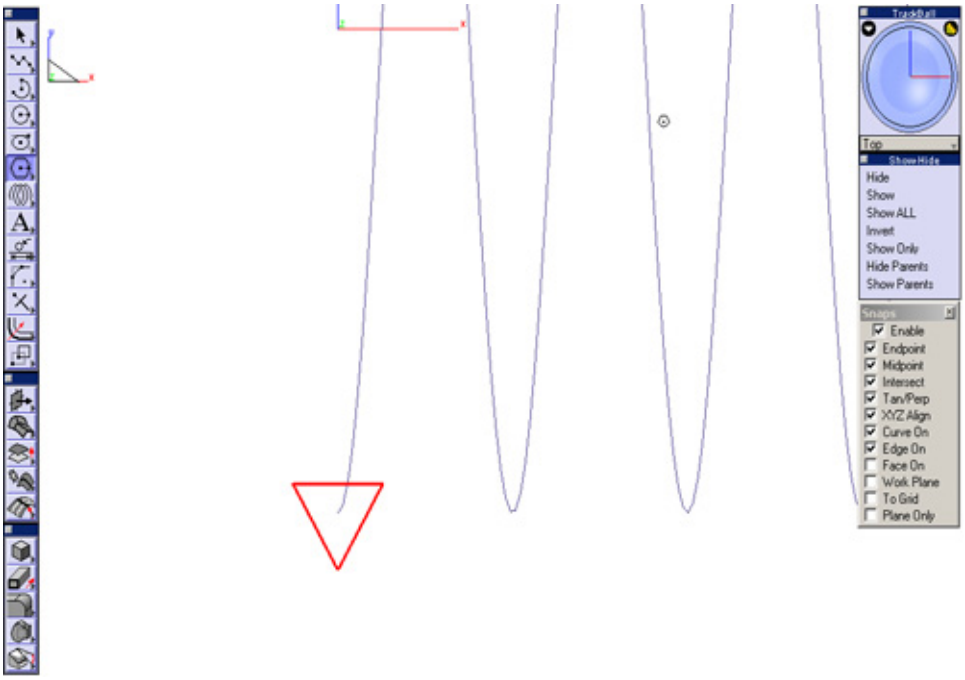
Next we will create the profile of the thread. The profile is just a triangle that can be created using the ***Inscribed Polygon*** tool.



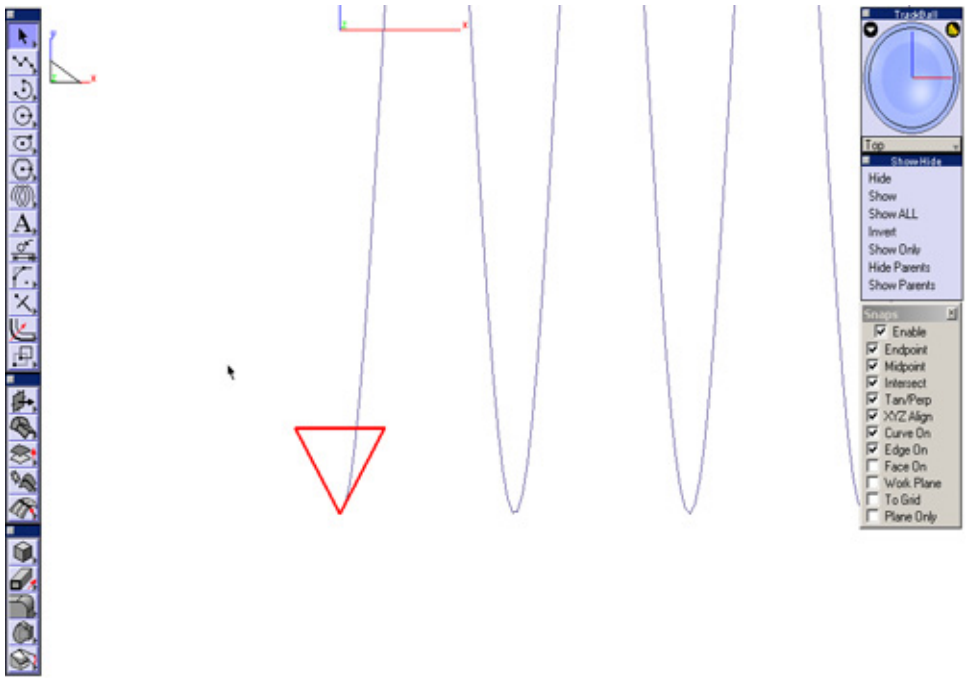
In the **Status Line**, set the number of sides to three.



Create a polygon as shown below, using the endpoint of the helix as the center of the polygon.



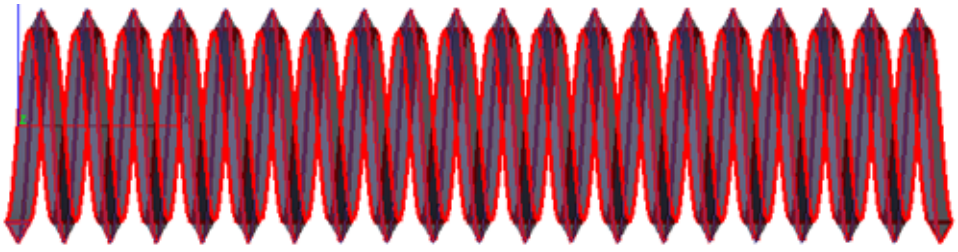
Next, move the bottom endpoint of the triangle polygon to the endpoint of the helix.



Now that we have the path and the profile, all that's left is to sweep it. Select the ***Swept 1 Rail Solid*** *tool*.

Select the triangle polygon for the path to profile. You can either hold the ***Shift*** key while clicking on each line or you can **Fence** select all three line at once.

Next, select the helix for the sweep path.



Save your file.

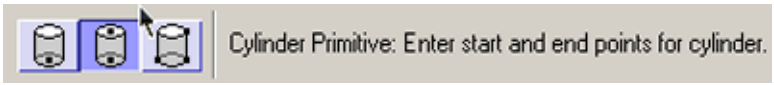
Creating the Body

To make things easier to see in the drawing area, we will temporarily hide the thread we just made.

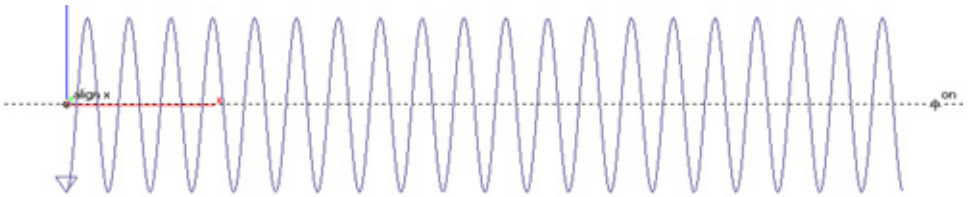
Select ***Hide*** from the Show-Hide palette and select the sweep that we just created or **Right Mouse** click on the sweep and choose **Hide** from the menu.

To create the body of the bolt, select the **Cylinder** tool from the solids tool palette.

In the tool options in the message line, make sure that create cylinder by two points option is selected as shown below.

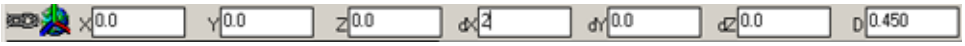


Set the first point to be at the origin of the drawing.



Click the second point somewhere to the right of the origin. You can see that the drafting assistant ensures that the second point is aligned with the first point along the X-axis.

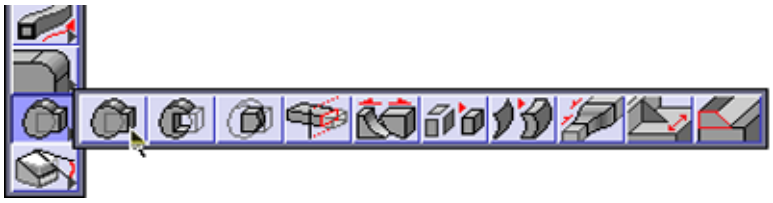
The body of the bolt needs to be a little smaller than the helix so the we can see the thread poking out. Therefore, make the D field (Diameter) .45. Also change the **dx** field to 2, as this is the length of the cylinder.



Now we need to bring back the swept thread that we hid earlier. To do this, select **Show All** from the Show-Hide palette. If you have hidden other objects besides the solid swept thread you should select **Show** from the palette and choose the thread as the object you want to un-hide.

Right now there are two solid objects in this file, the cylinder and the thread. We need to combine them into one solid object to perform the next step.

Select the **Boolean Add** also known as the**Union** tool from the solids tool palette.



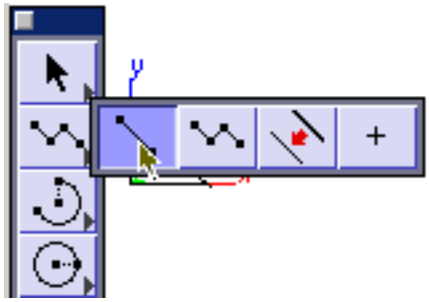
Select one of the solids and then the other solid. This will join the two together into one solid.

Save your model before going to the next step.

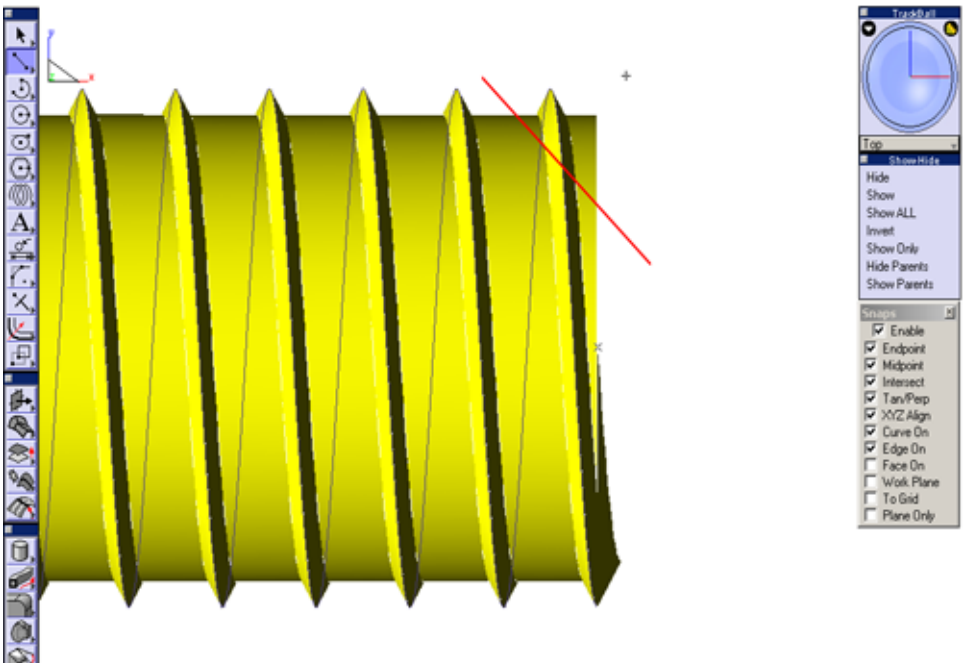
Tapering the Edge

To give a chamfered edge to the end of the bolt, we will create a Revolved Surface and use it to trim off the end of the bolt.

Select the ***Line*** tool from the tool palette.

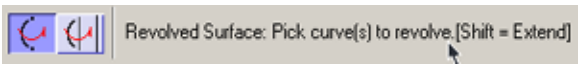


Draw a line similar to the one shown below.



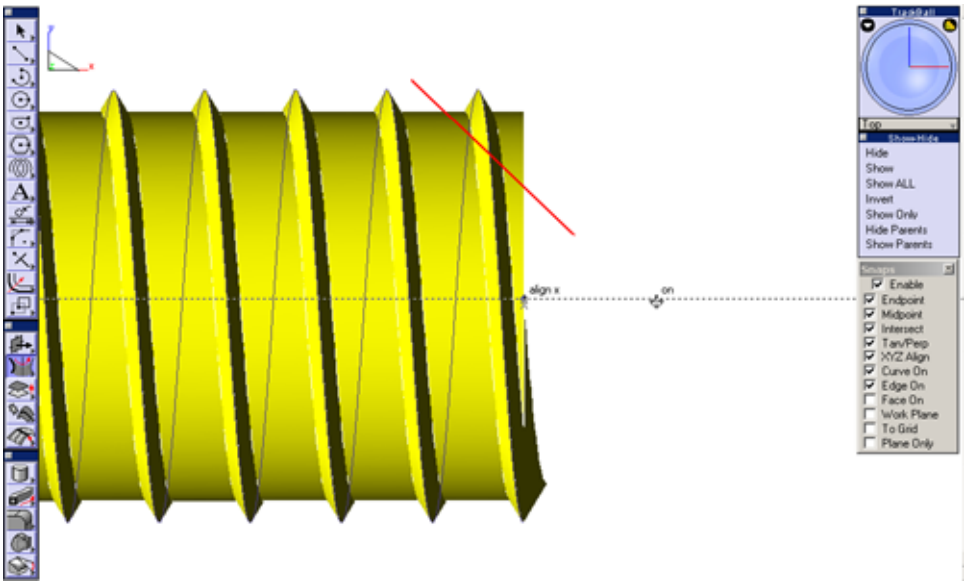
Select the create a ***Surface of Revolution*** tool from the surface tool palette.

Make sure that the option shown below in the message line is selected. The one selected is revolve around two points. This makes it so that we do not have to draw another line to use as a reference for it to revolve around.



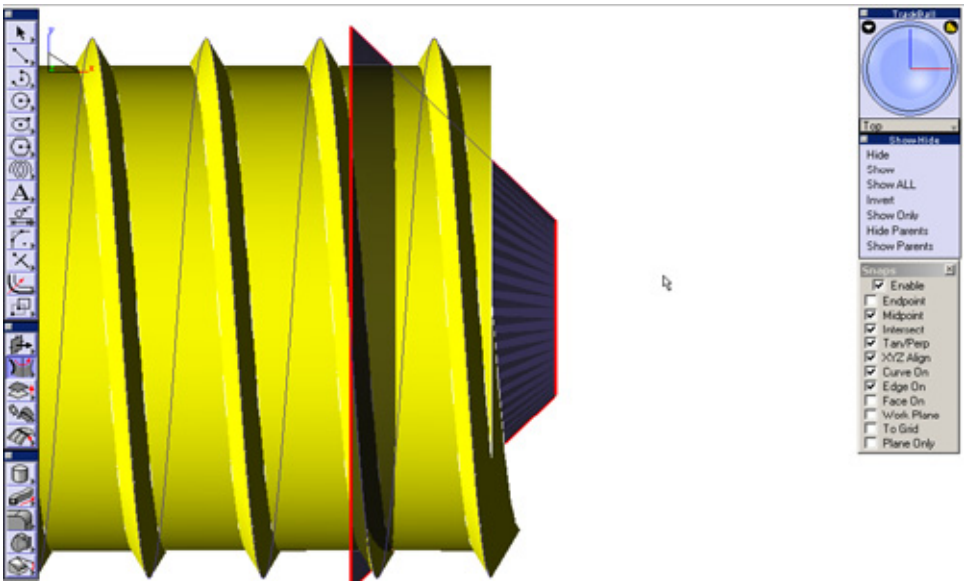
Select the line for the curve to revolve.

Select the center of the solid object as the first point of the rotation axis. Let the drafting assistant help you select the point shown below.



The second point is to the right and along the X-axis. Once again, the drafting assistant is very helpful in making sure that your points are lined up.

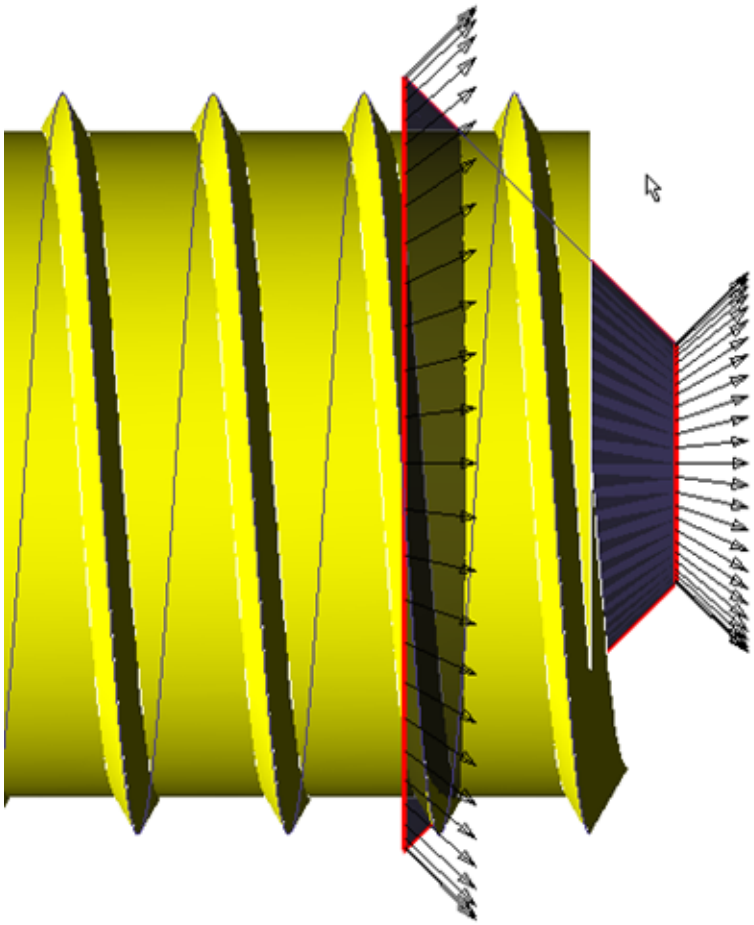
This is the surface that is created. All we need to do now is trim the part of the bolt off the end using this surface.



Select the ***Trim Solid*** tool.

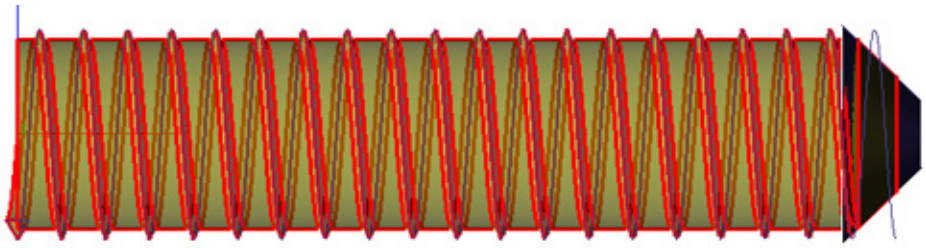
Make sure that the option in the message line is set to Perpendicular to View. (It is the default setting for the tool.)

Select the surface first. Then select the solid.



The arrows on the surface indicate the direction the solid will be trimmed.

Your model should now look like this. If the wrong part got trimmed, you can tap the ***CTRL*** key on Windows or the ***Option*** key on a Mac to flip the side that gets trimmed.



Hide the surface that we used to trim the bolt to see the tapered edge.

Save your file.

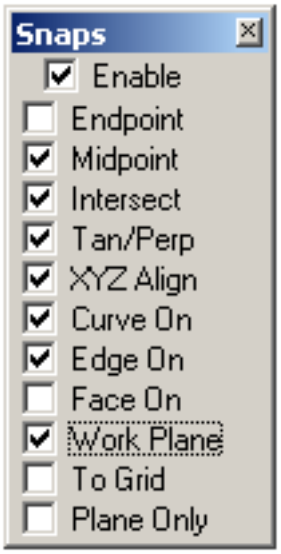
The last step is to put the Hex head on the bolt.

Creating the Bolt Head

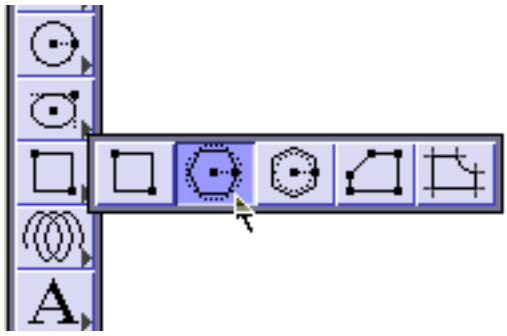
From the Show-Hide Window, select the ***Show Only*** tool and the main part of the bolt. You can also **Right Mouse** click on the solid part and choose **Show Only**.

Change your view to the Right Side. You can select this from the drop down list at the bottom of the ***Trackball***.

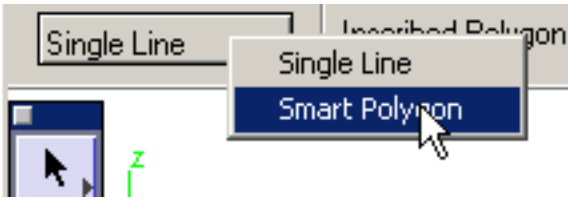
Turn ***ON*** the ***Work Plane*** Snap in the Snap Options Dialog. This will ensure that anything we create will be placed on the default work plane



Select the ***Inscribed Polygon Tool*** from the Tool Palette.

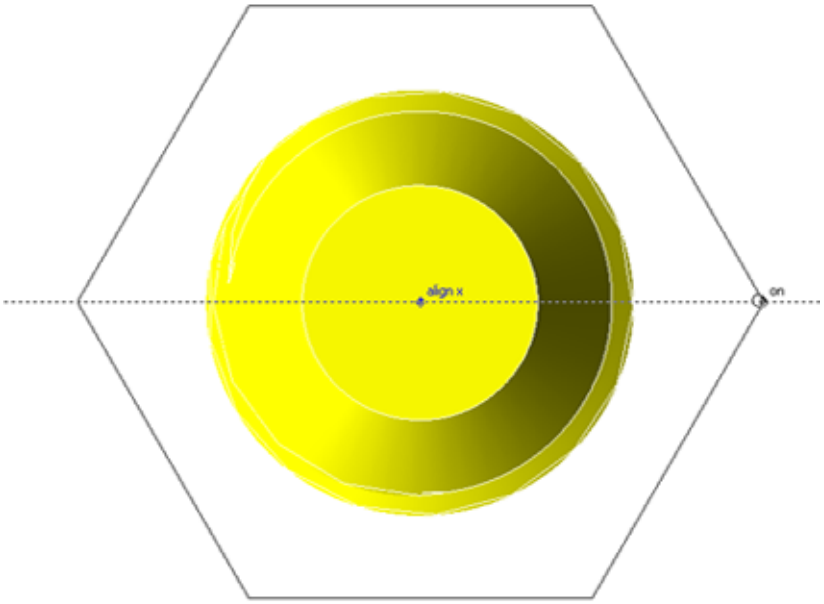


Set the option in the Message Line to ***Smart Polygon***. This will make it so that you create one, six sided piece of geometry instead of six pieces of geometry. (You will need to change the number of sides to six as it still reads three from the triangle created earlier in the tutorial.)

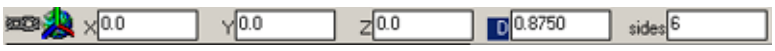


Select the first point of the Polygon (in our case a hexagon) to be in the center as shown in the image below.

Select the second point to the right of the first along the x-axis.



Type in ***.875*** into the D (Diameter) field and hit ***Enter***.



Switch to the ***Isometric*** view by selecting it from the drop down menu at the bottom of the ***Trackball*** or by tapping the “f” key.

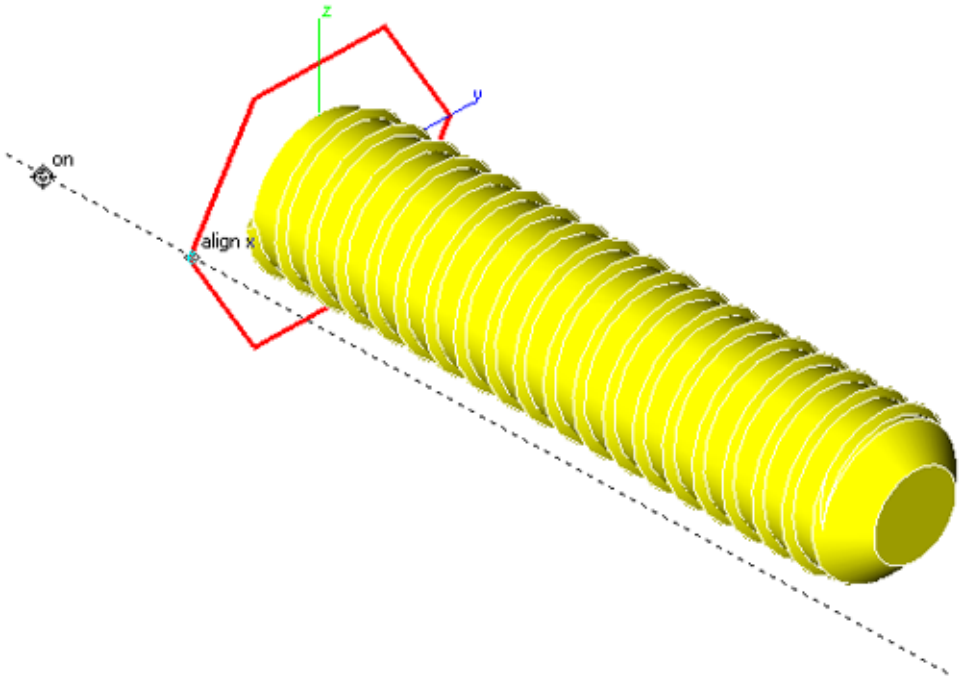
Zoom all by pressing ***CTRL+F*** on Windows or ***Command+F*** on Macintosh. You may want to zoom out one time to have a little more area around the model. Do this by pressing ***CTRL+[*** on Windows or ***Command+[*** on Macintosh.

Turn **OFF** the ***Work Plane*** Snap that we turned on before. The Work Plane snap can be troublesome to work with when you are in a non-orthographic view.

Select the ***Extrude Solid*** tool from the solids palette.

Select the hexagon that we just created.

Select the two points shown below for the Extrusion vector.

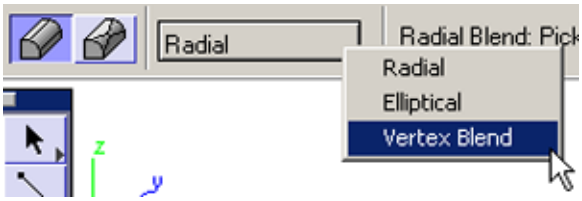


Type in ***.5*** for the distance and hit ***Enter***.

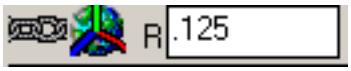
Next we will put blends on the vertices of the head of the bolt.

Select the ***Blend*** tool from the Solids tool Palette.

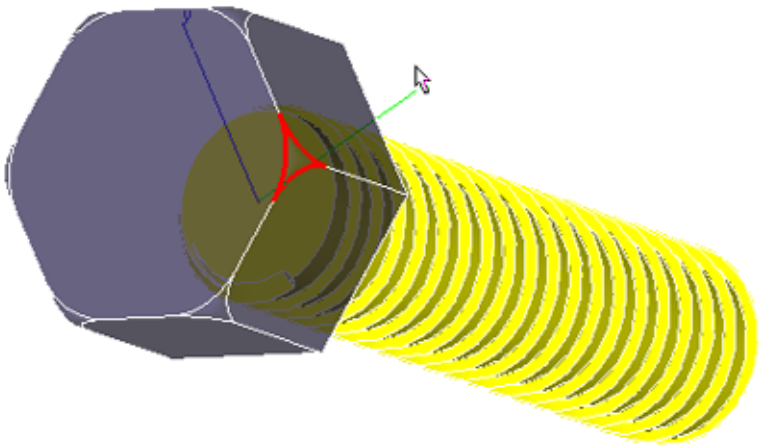
Select the ***Vertex Blend*** Option from the Message Line.



Change the radius of the Blend to ***.125*** inches.



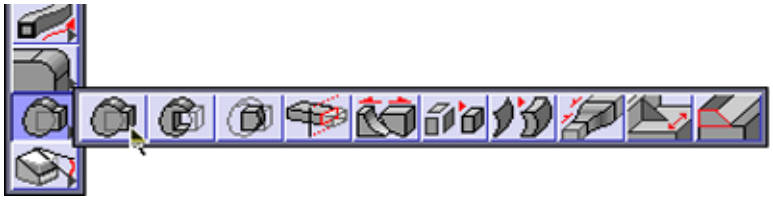
Select one of the corners to blend it as shown below. You will have to make you way around and select all of the corners. You may want to rotate the view to see some of the corners better as you go.



Once that is complete, all that is left is to union the bolt body and the bolt head together.

We will do this the exact same way we joined the thread and the body.

Select the **Boolean Add** aka the ***Union*** tool from the Solids tool palette.



Select the head of the bolt and the body of the bolt.

Your finished threaded hex bolt should like similar to the image below.