



# Threaded Hex Bolt Tutorial

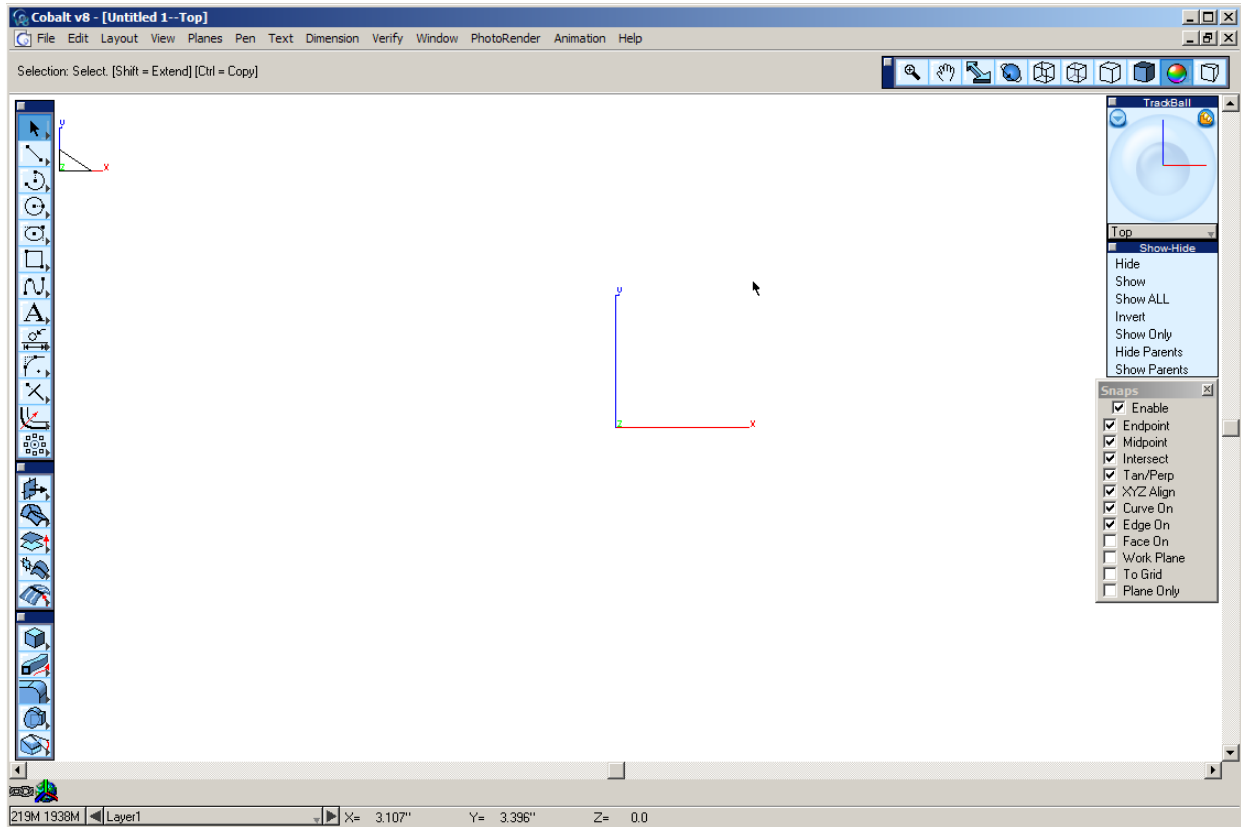
*Using Cobalt<sup>™</sup>, Xenon<sup>™</sup>, Argon<sup>™</sup>*

# Threaded Hex Bolt

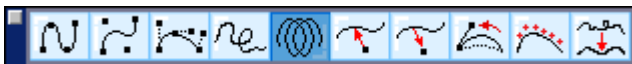


This tutorial describes one method for modeling a threaded hex bolt.

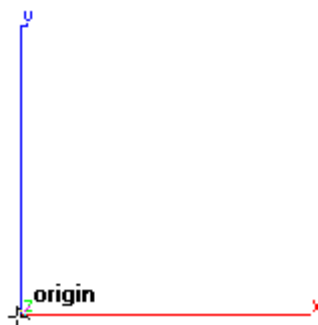
To begin, open Cobalt, Xenon or Argon and organize the screen so that it looks similar to the image below. Open the Show-Hide and the Snaps windows, the rest of the tool palettes on the picture are opened by default when the application is launched. All of these tools are accessible through Window menu. Start in the Top View.



1. First, 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.



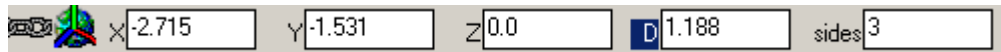
- Once those two points are entered, type **.5** into the Diameter field and **2** into the Length field and press ENTER. This gives the thread for a 1/2" diameter, 2" long bolt.



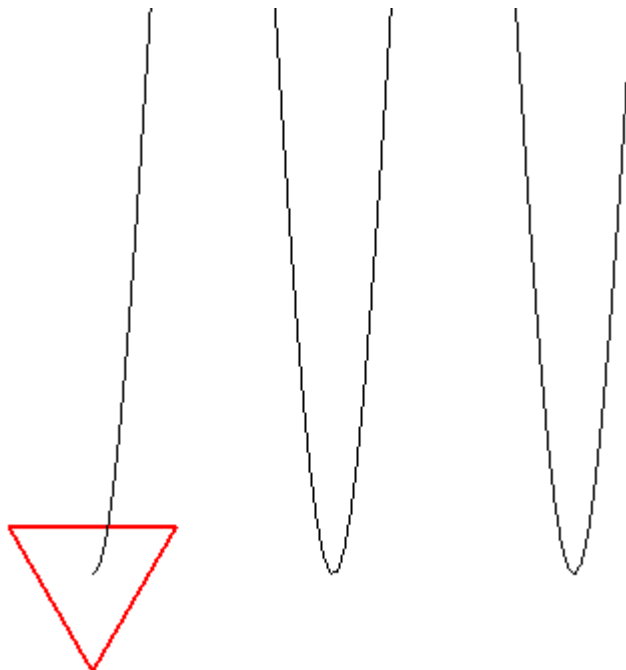
- Next create the profile of the thread. The profile is just a triangle that can be created using the **Inscribed Polygon** tool.



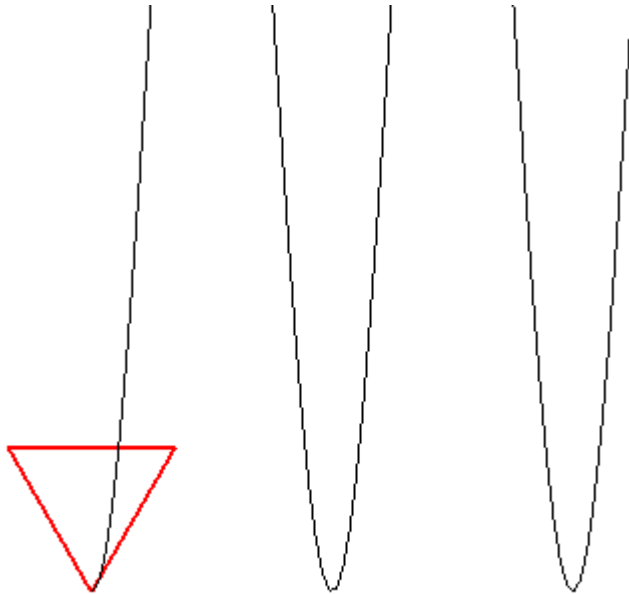
- In the Status Line, set the number of sides to three.



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



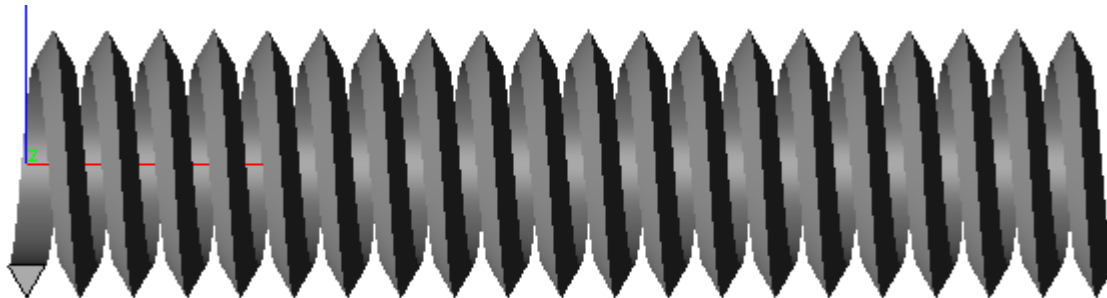
- Next, move the bottom endpoint of the triangle polygon to the endpoint of the helix.



3. Now there is the path and the profile, all that's left is to sweep it. Select the **Swept Solid** tool.



- Select the triangle polygon for the path to profile. Either hold the SHIFT key while clicking on each line or fence-select all three lines at once.
- Select the helix for the sweep path.

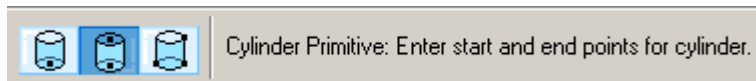


## Creating the Body

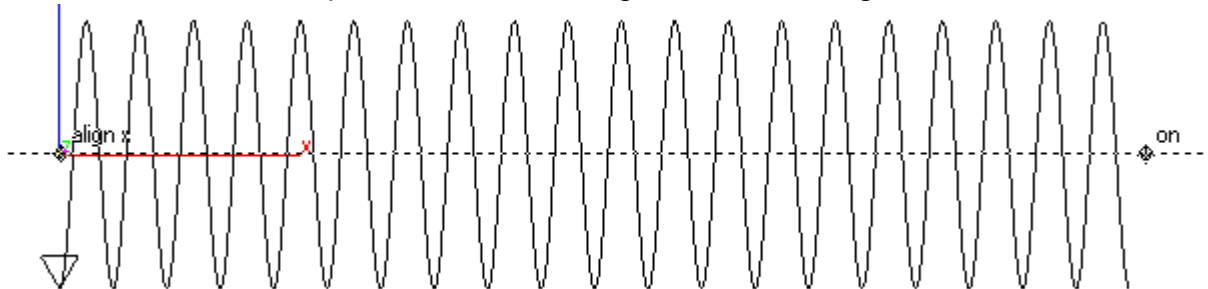
- To make things easier to see in the drawing area, temporarily hide the thread just made. Select Hide from the Show-Hide palette and select the sweep that we just created or Right Mouse click on the sweep and choose Hide from the menu.
- To create the body of the bolt, select the **Cylinder** tool from the solids tool palette.



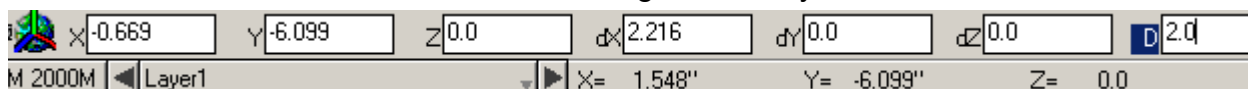
- In the tool options in the Message Line, make sure that Create Cylinder by Two Points option is selected as shown below.



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



- Click the second point somewhere to the right of the origin. 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 to 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. Press ENTER.



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

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

- Select the **Union Solid** tool from the solids tool palette.

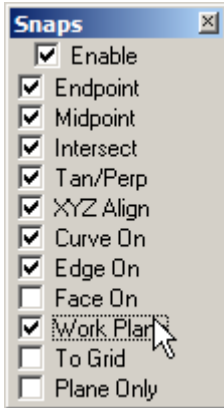


- Select one of the solids and then the other solid. This will join the two together into one solid.
- Save the model before going to the next step.

## Tapering the Edge

To give a chamfered edge to the end of the bolt, create a revolved surface and use it to trim off the end of the bolt.

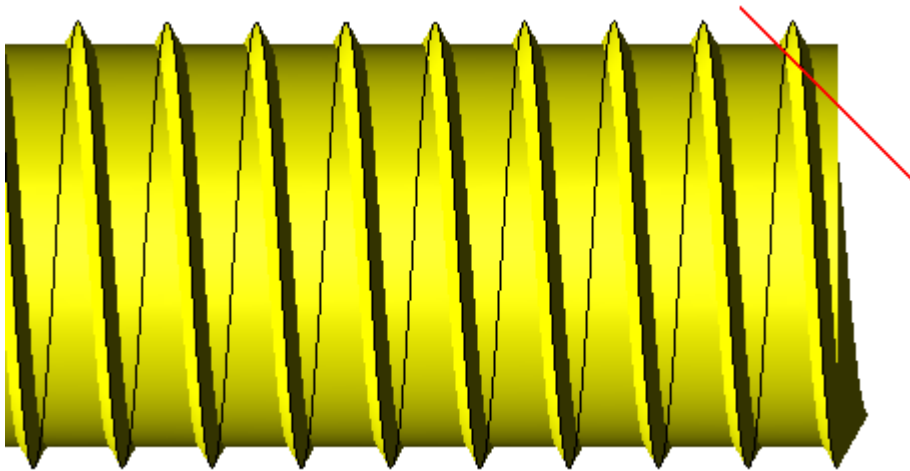
9. Check Work Plane option in the Snaps window.



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



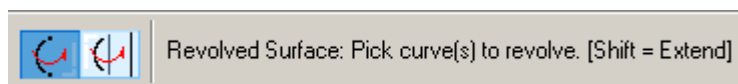
- Draw a line similar to the one shown below.



11. Select the **Revolved Surface** tool from the surface tool palette.

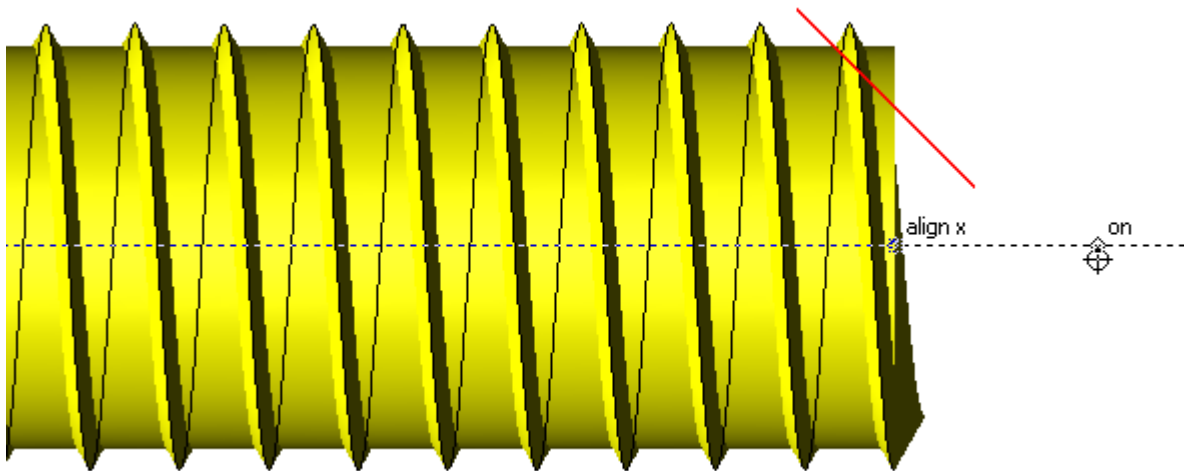


- Make sure that the option shown below in the message line is selected. The one selected is revolve around two points. It is not necessary to draw another line to use as a reference for it to revolve around.



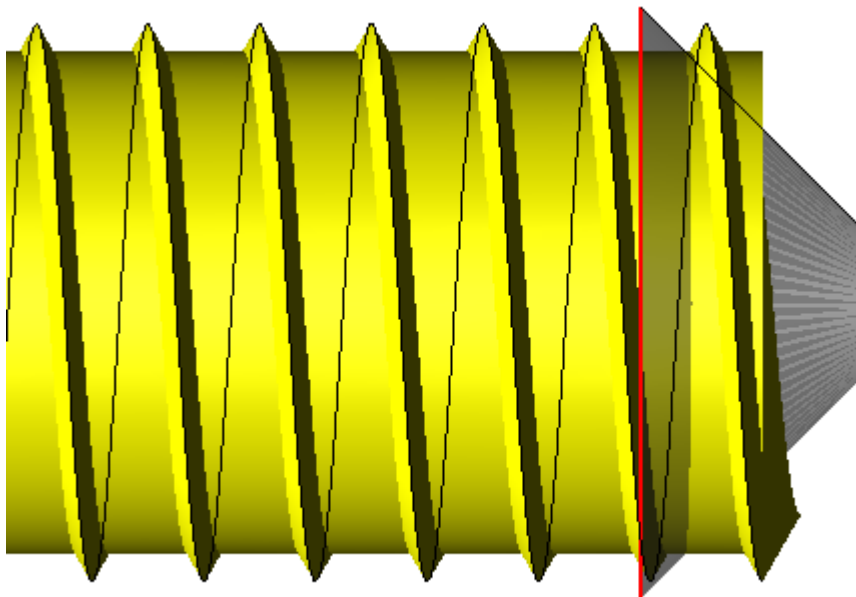
- Select the line for the curve to revolve.

- Select the center of the solid object as the first point of the rotation axis. The Drafting Assistant helps to 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 the points are lined up.

*This is the surface that is created. Trim the part of the bolt off the end using this surface.*



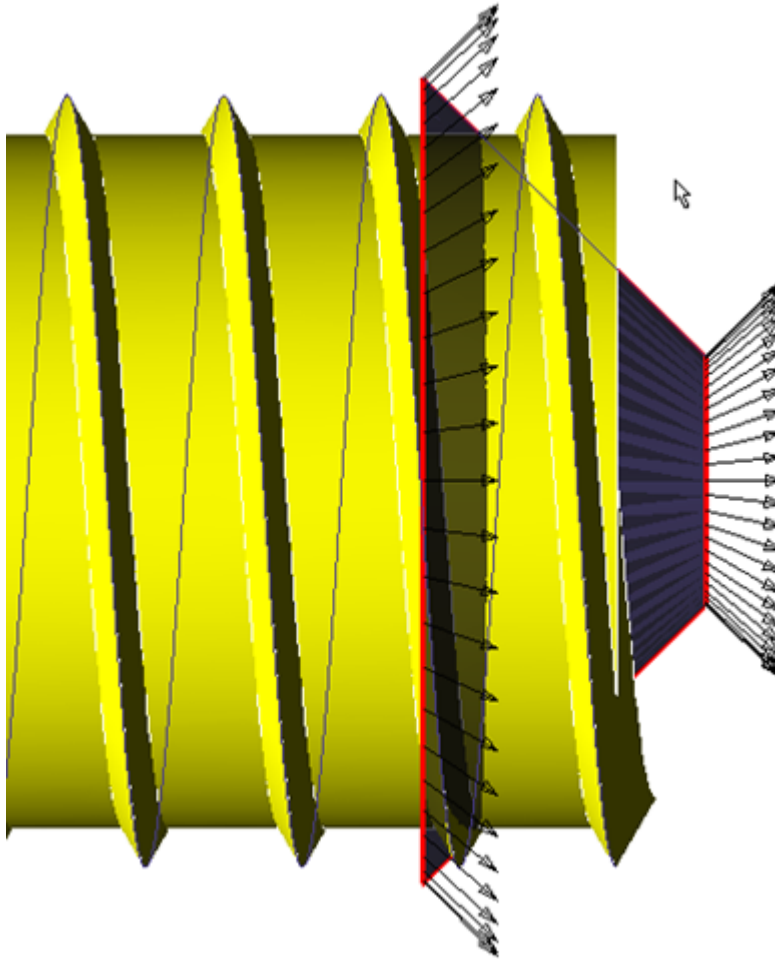
12. Select the **Trim Solid** tool.



- Make sure that the option in the Message Line is set to **Perpendicular to View**. (It is the default setting for the tool.)



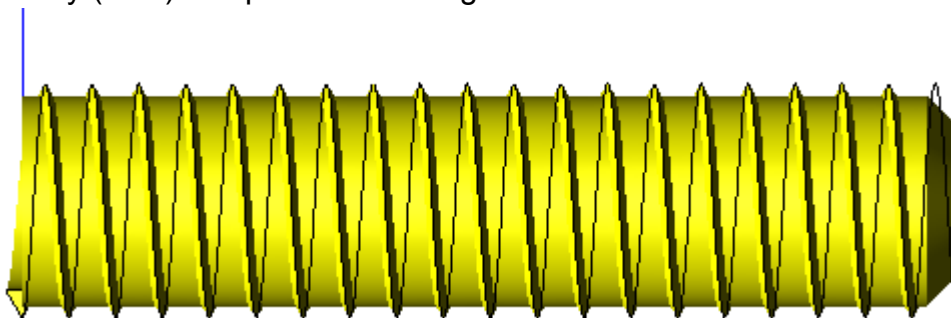
- Select the surface first. Then select the solid.



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

*The model should now look like this.*

- If the wrong part got trimmed, tap the CTRL key (Windows) or the OPTION key (Mac) to flip the side that gets trimmed.



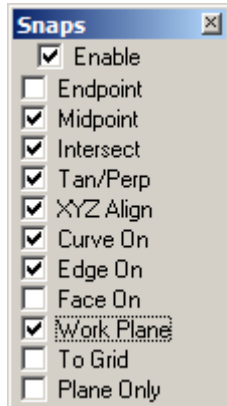
13. Hide the surface that was used to trim the bolt to see the tapered edge.

14. Save the file.

*The last step is to put the hex head on the bolt.*

## Creating the Bolt Head

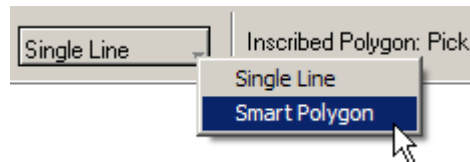
15. From the Show-Hide window, select the Show Only option and the main part of the bolt, or Right Mouse click on the solid part and choose Show Only.
16. Change the view to the Right Side. Select this from the drop down list at the bottom of the Trackball.
17. Turn ON the Work Plane snap in the Snap dialog. This will ensure that anything created will be placed on the default work plane.



18. Select the **Inscribed Polygon** tool from the tool palette.



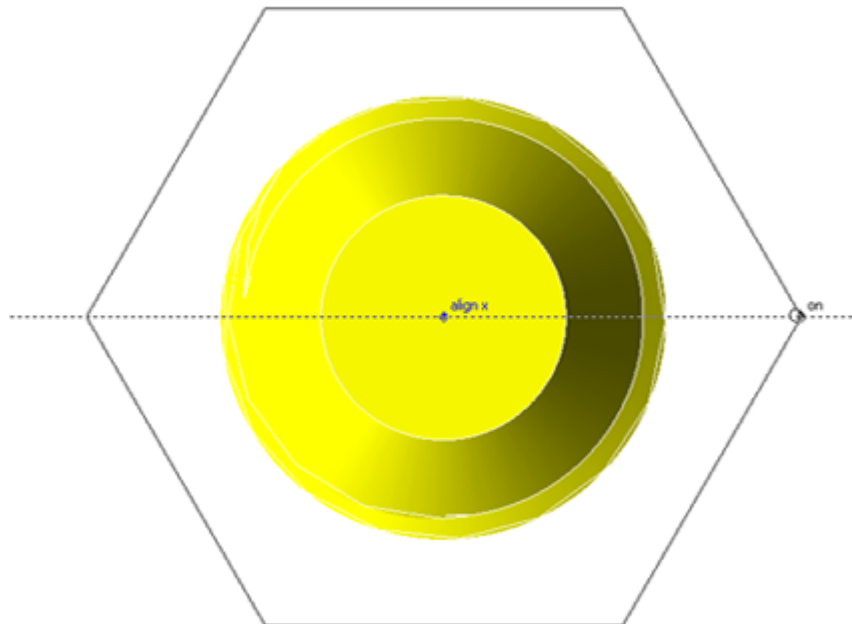
- Set the option in the Message Line to **Smart Polygon**. This creates one six-sided piece of geometry instead of six pieces of geometry. (Change the number of sides to six as it still reads three from the triangle created earlier in the tutorial.)



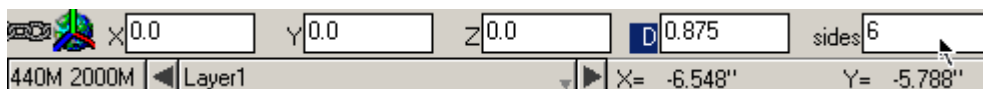
- Select the first point of the polygon (in our case a hexagon) to be in the center as shown in the image below.



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



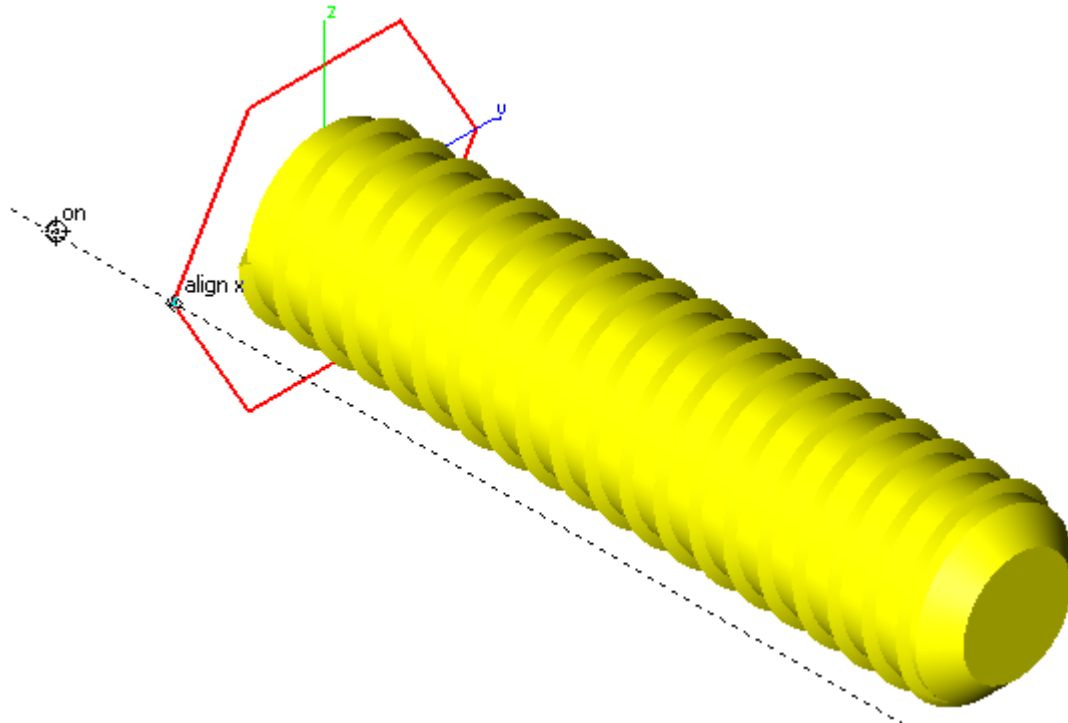
- Type in .875 into the **D** (Diameter) field and pressENTER.



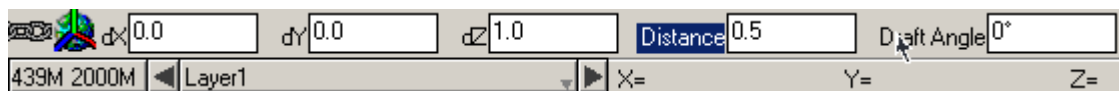
19. Switch to the Isometric view by selecting it from the drop down menu at the bottom of the Trackball or by tapping the F key.
20. Zoom all by pressing CTRL+F (Windows) or COMMAND+F (Macintosh). It is possible to zoom out one time to have a little more area around the model. Do this by pressing CTRL+[ (Windows) or COMMAND+[ (Macintosh) or scroll the middle mouse button.
21. Turn OFF the Work Plane snap that was turned on before. The Work Plane snap can be troublesome to work with when in a non-orthographic view.
22. Select the **Extrude Solid** tool from the solids palette.



- Select the hexagon that was just created.
- Select the two points shown below for the Extrusion vector.



- Type in **.5** for the distance and hit ENTER.



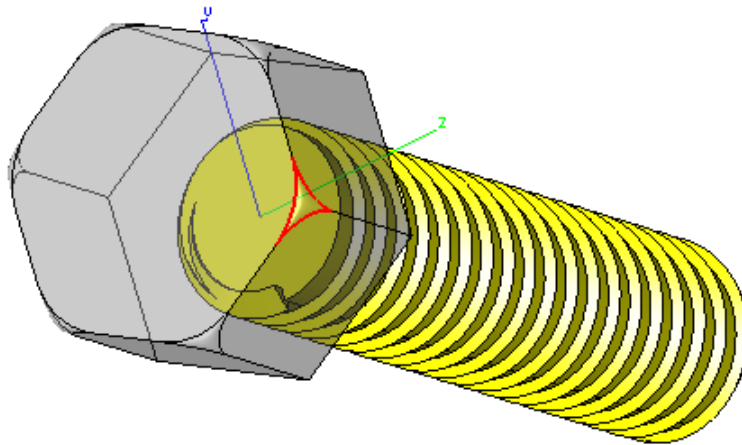
23. Next put blends on the vertices of the head of the bolt.
  - Select the **Blend** tool from the **Solids** tool palette.



- 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. Make the way around and select all of the corners. Rotate the view to see some of the corners better.



24. Once that is complete, all that is left is to union the bolt head and the bolt body together. Do this the same way as the thread was joined to the body.

- Select the **Union Solid** tool from the **Solids** tool palette.



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

The finished threaded hex bolt should look similar to the image below.

