CENTRAL WASHINGTON UNIVERSITY
IET 265.001
ENGINEERING DESIGN USING SOLIDWORKS

INSTRUCTIONS FOR MODELING TYPICAL HEXHEAD UNIFIED FINE AND UNIFIED COURSE THREADED FASTENERS USING SOLIDWORKS 2009.

THIS EXERCISE UTILIZES THREADED FASTENER NOMENCLATURE TO DESIGN BOTH ENGLISH AND METRIC HEX HEAD BOLTS.

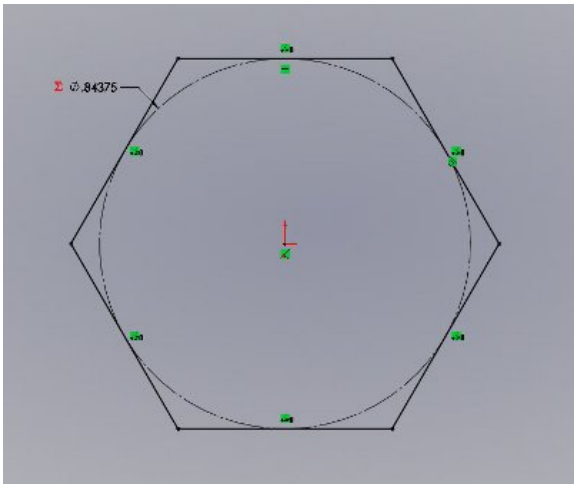
Requirements:
Precision to 3 significant digits
Material: AISI 1045 Steel
Read and understand the information provided through the links on the Student Resources page of the Class Website

Table of formulas needed for this design:

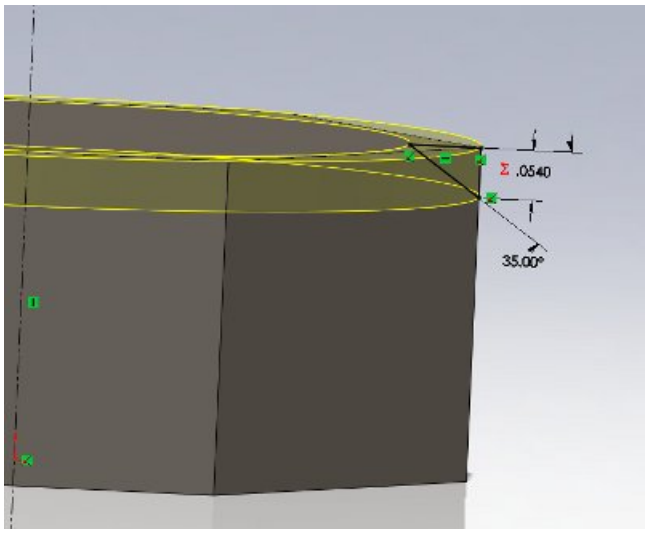
Item Name	Letter	Formula	Derived	Location
Bolt Diameter	Db		Given	Bolt Nomenclature
Bolt Length	Lb		Given	Bolt Nomenclature
Threads per Inch	Ti		Given	Bolt Nomenclature (inches only)
Head Diameter	Dh		Look up	Website
Head Height	Hh	Hh=Db*0.64	Determine	
Head Chamfer Cut	Ch	Ch=Hh*0.15	Determine	
Hd Chamfer Angle		35 degrees		
Ring Shank Diam.	Ds	Ds=Dh*0.90	Determine	
Ring Shank Height	Hs	Hs=Hh*0.05	Determine	
Ring Shank Fillet	Rs	Rr=Hs	Determine	
Bottom Chamfer	Cb	Cb= Dh *0.10	Determine	
Bt. Chamfer Angle		45 degrees		
Thread Frequency	Tf	Tf=1/Ti	Determine	Inches only
Thread Frequency	Tf		Given	Bolt Nomenclature (metric only)
Thread Cut Pitch	Pt	Pt=Tf	Determine	

Procedure:

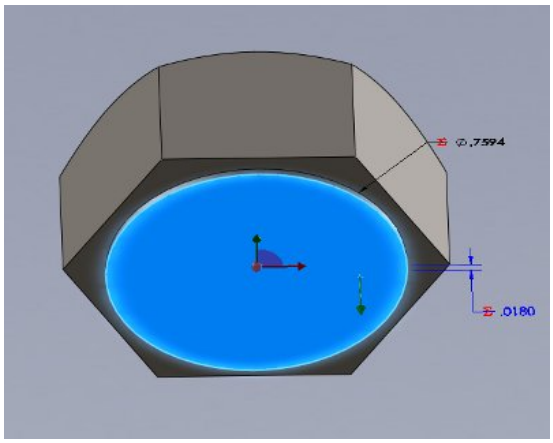
1. Using the information on the hex bolt provided to you, begin to build your model. Think about the orientation of your model and start a sketch of a hexagon on the appropriate plane with the appropriate dimensions given to you for your design. Select the Inscribed circle option of this hexagon and fully define this sketch. One of the elements of definition is to make one of the sides of the hexagon either horizontal or vertical. The diameter of the inscribed circle of the hexagon, typical of many of the features in this and other similar designs will be determined using the formulas above. Extrude this sketch the appropriate distance as your Base Extrude. The following image is representative of a 9/16 bolt with 12 threads per inch which will be used throughout this example.



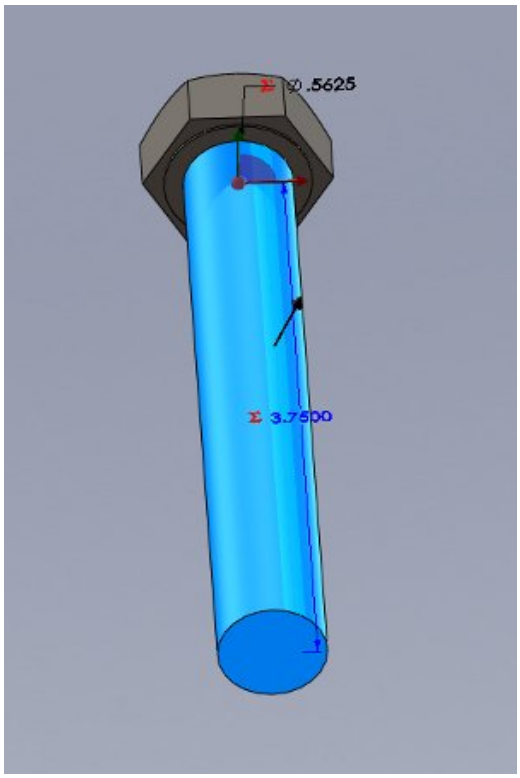
2. Create the Top Taper Revolved Cut feature guided by the information provided in the table above. Notice where the sketch plane is, in reference to the side of the Bolt Head.



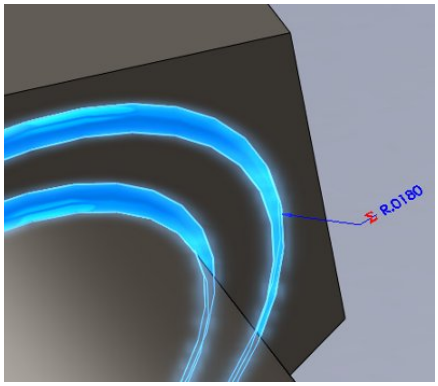
3. Sketch the diameter of the Ring Shank feature in the center of the bottom of the Bolt Head. Extrude to the derived length using values given to you for your design.



4. Sketch the diameter of the Bolt Shank in the center of the bottom of the Ring Shank Feature and extrude to the length given to you for your design.

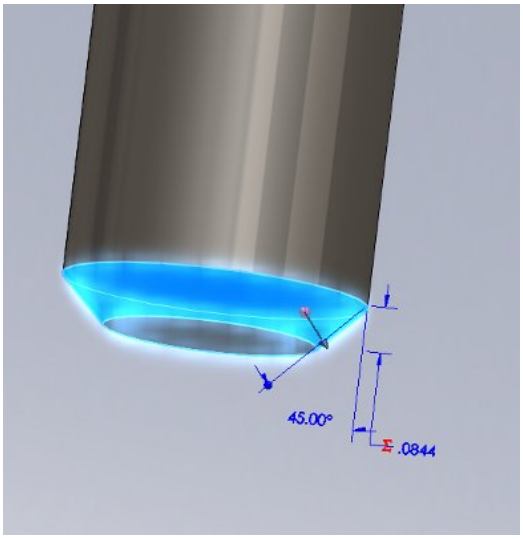


5. Apply the Ring Shank fillet feature between the Ring Shank and the bottom of the Bolt Head and between the Ring Shank and the Bolt Shank using the derived values given to you for your design.

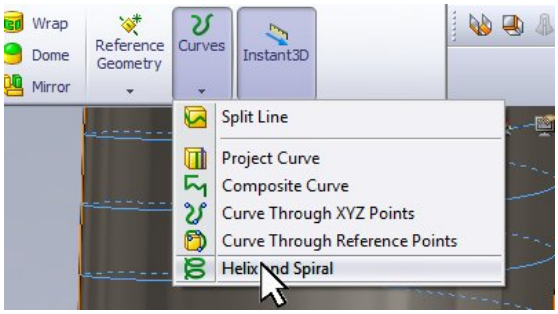


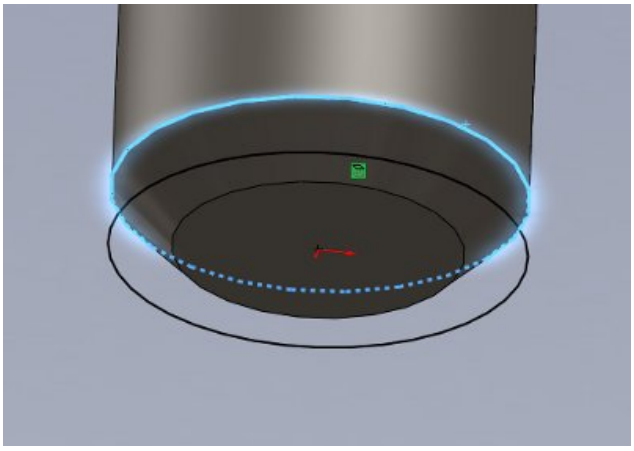
6. Apply the Bottom Chamfer feature using the derived values given to you for your design.

7. The final steps involve a Swept Cut feature involving a helix path and a specifically designed sketch for the cut profile. For the helix you need to sketch a circle as a parent for the Helix Feature. Click on the bottom face of the Bolt Shank, start a sketch, select the outside circular edge of the shank and apply the Convert Entities sketch entity. This will project the circular edge of the Bolt Shank onto your sketch.

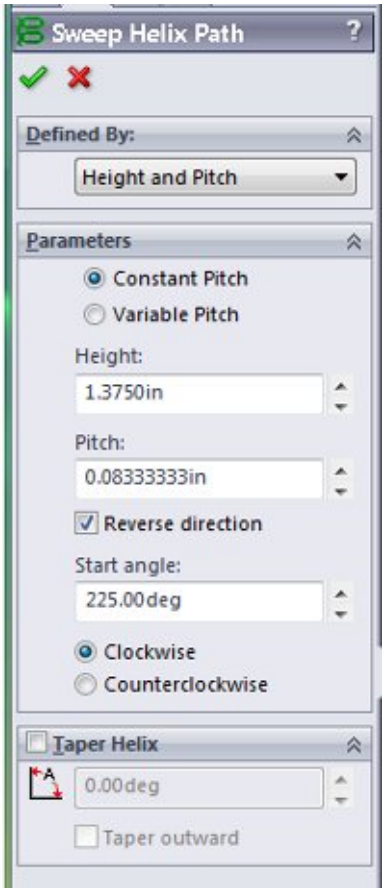


8. Select the Helix and Spiral from the Features tab of the Command Manager.

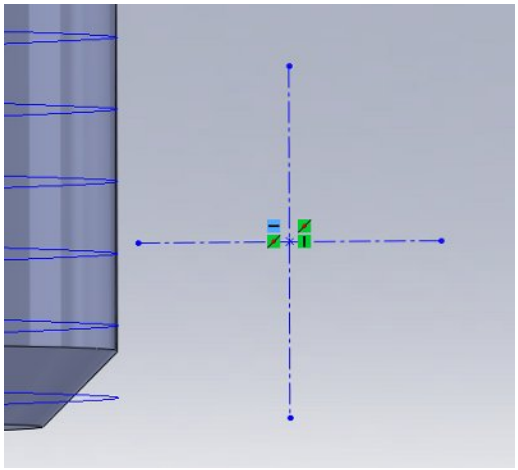




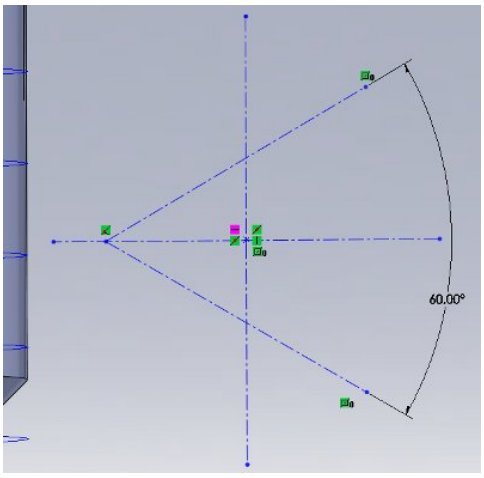
9. There are three selections to choose from when modeling a helix, we will choose the Height and Pitch selection since the thread length and Thread Cut Pitch will be given or derived. Enter the Height and Pitch of the helix in the places shown, check the Reverse Direction box, choose or enter the default start angle of 225 degrees and check the Clockwise radio button.



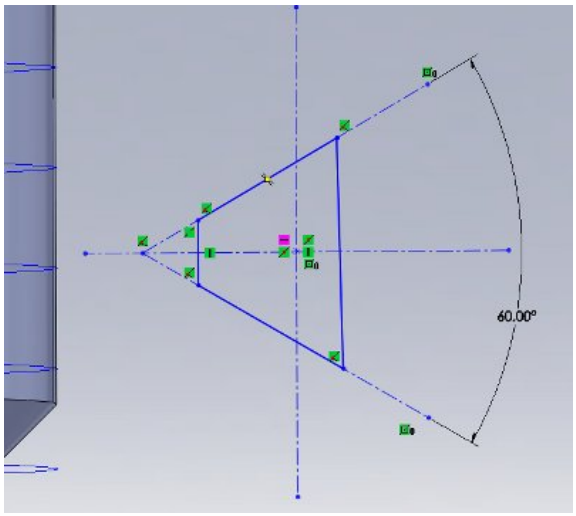
10. Now we need to sketch our cut profile. Select a primary plane that is perpendicular to the center axis of the bolt. I would suggest the following procedure. Sketch a horizontal and vertical centerline that have their midpoints intersect with each other. At this intersection place a Point like you would do for a Hole Wizard.



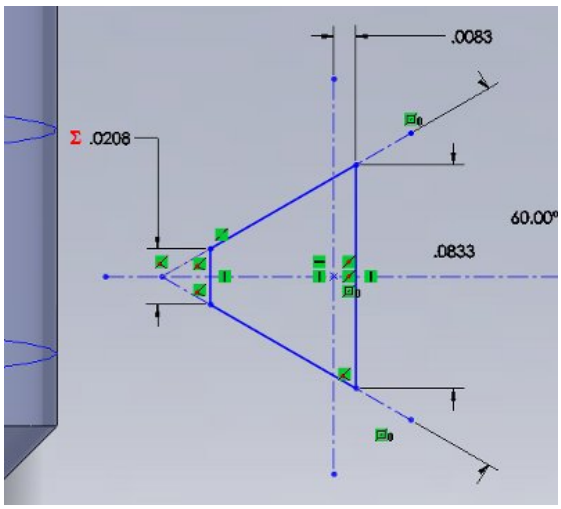
11. Sketch a centerline at roughly a 30 degree angle and mirror it to the bottom using the horizontal center line as a mirror line. This establishes a symmetric relationship. Put a 60 degree angular dimension between these two lines.



12. Roughly sketch a trapezoid on the centerlines draw as shown. Make the lines collinear with the center lines. The vertical lines should have a vertical Sketch Relation.



13. You will be placing three related dimensions on the following sides of your trapezoid. The longer vertical line will be the Pitch. This dimension represents one full cycle of cutting and is equal to the Pitch for the helix. The shorter vertical line is 0.25 times the Pitch. And the distance between the vertical centerline and the longer vertical line is 0.108 times the pitch (I did the trigonometry for you on this one.



14. The sketch is almost fully defined; one last step will be to fully define it. Take the point that you drew at the intersection of the two centerlines and select it along with the helix. Select the Pierce Sketch Relation option. Rebuild or Exit Sketch.

15. Rename this sketch to Cut Profile Sketch and apply the Swept Cut Feature using this sketch and the helix as a path.