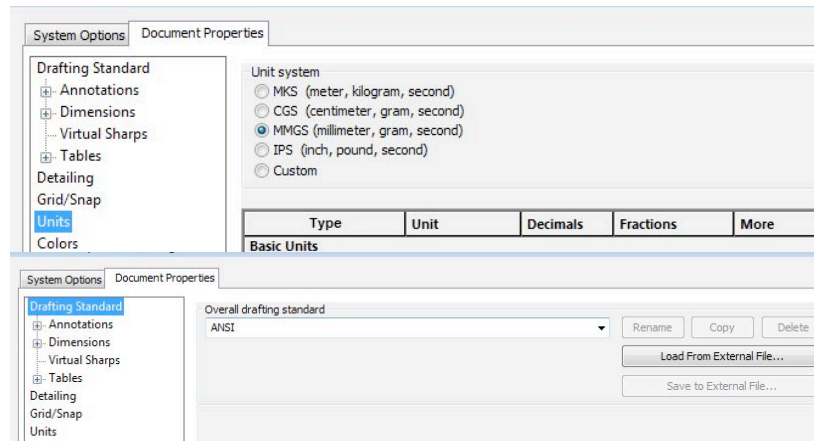


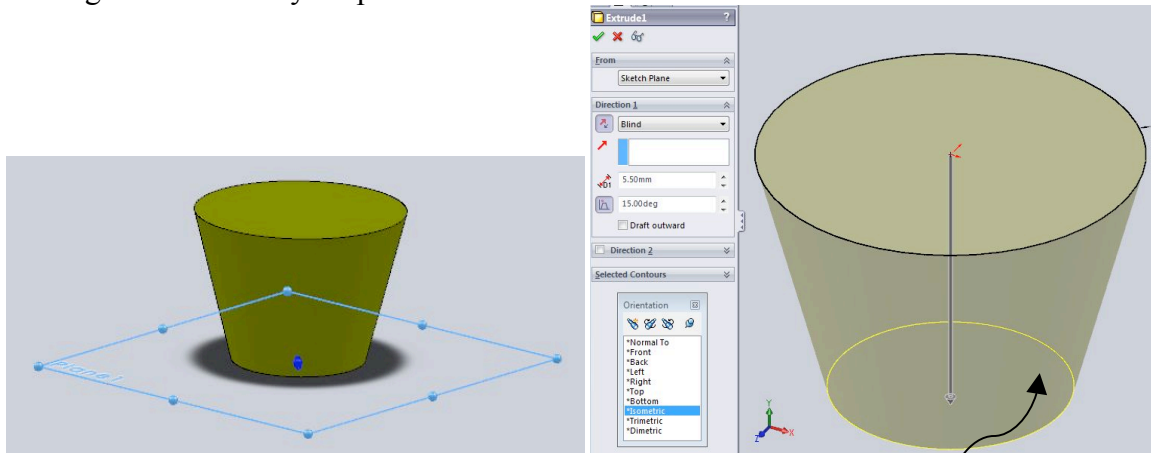
Solidworks 2009 Tutorial: Creating Pieces of a Craft Knife

Part I: Rendering the Slotted Collet

1. After opening Solidworks, open a new part file and immediately make sure your document's settings are set to overall ANSI standards and metric units (mm, g, s). I have found that checking these settings at THIS point is much less troublesome than doing so later.

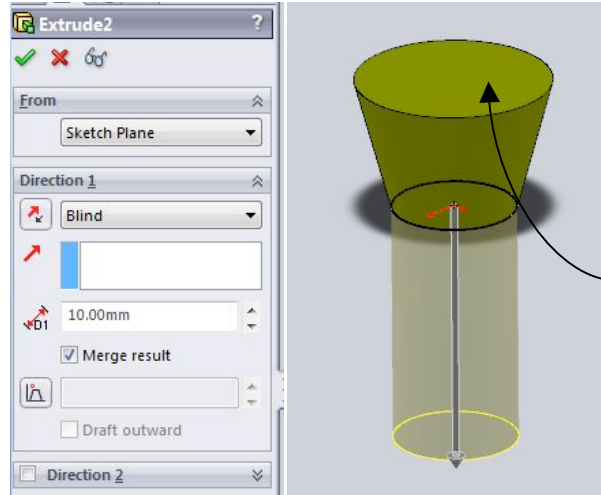


2. Before you do anything else, you need to give your part some character—make it stand out from the monotony that many call gray. Edit its appearance to change its sheen, transparency, and most importantly its color.
3. Open a sketch in the top plane and sketch a circle from the origin. Smart dimension this circle to 8mm. Use the extrude command to extrude this circle downward 5.50mm at a 15° angle. Make sure your preview looks like this.

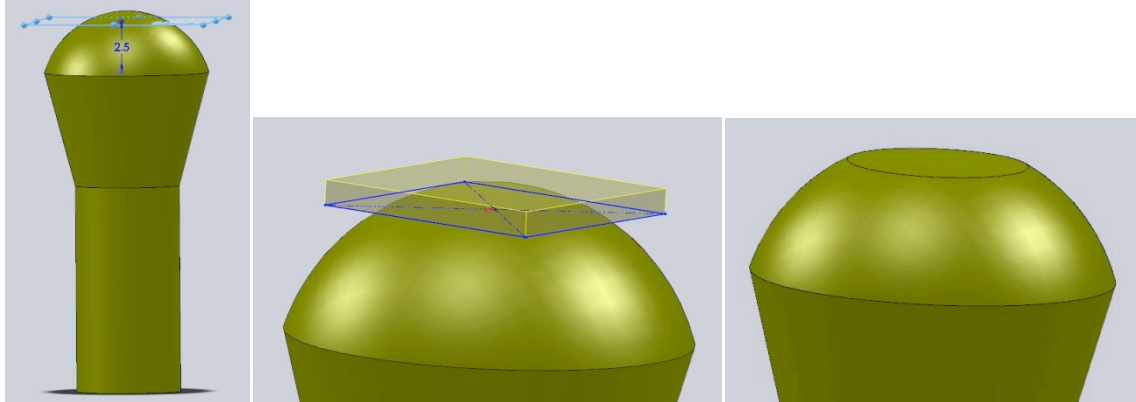


4. Next, you will need to insert a plane parallel to and coincident with the face bounded by the yellow line in the previous picture. To do this, select Insert/ Reference Geometry/ Plane, and select the face I just described. Make sure the option "Offset distance" has the value zero, or else you will make a very non-functional collet. Click the check mark.
5. After you insert the plane, insert a sketch on the plane. Use the "Convert Entities" tool to convert the circular face (in the previous step) to the current plane. Now you will extrude

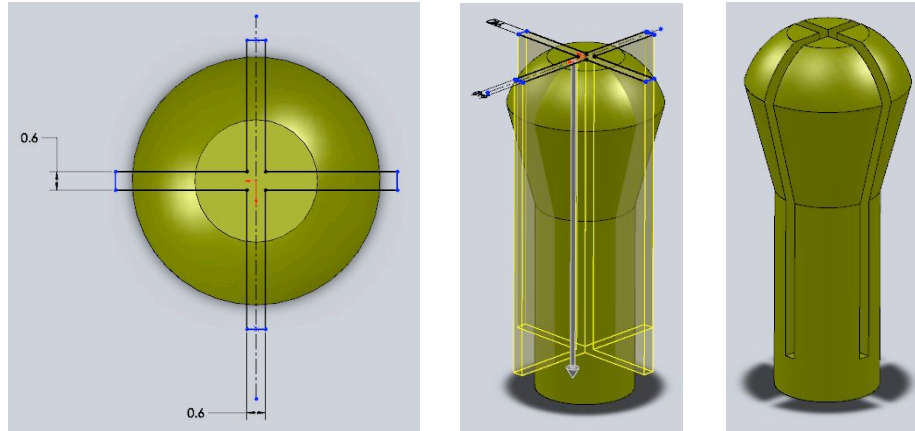
your new circle downward 10mm. Do not draft the extrusion this time. Ignore the physically impossible shadow.



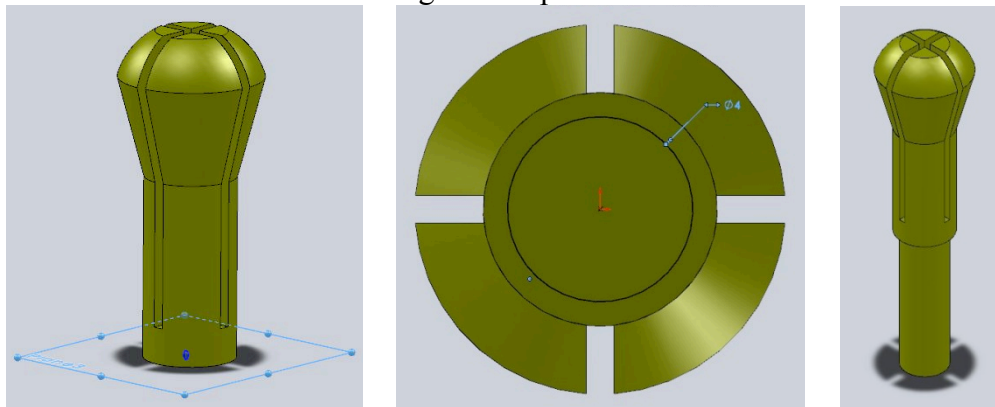
6. Now, use the dome command to create a dome on the uppermost circular face. Set your spherical radius to be 3mm.
7. Next, create another plane 2.5mm upward from the face you just domed. Insert a sketch on this plane, and draw any shape your heart desires, just as long as it is sufficiently large. To be safe, just draw a shape larger than any face on your part (Mine is the blue rectangle). Use the extruded cut tool on this shape to remove the portion of the dome above the plane you just created.



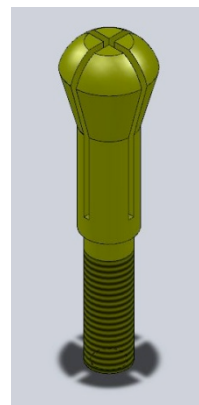
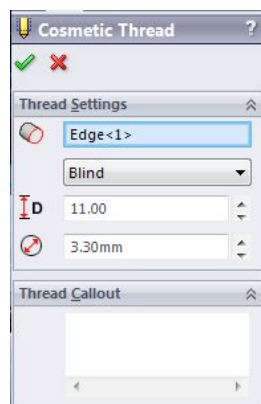
8. Now, insert a new sketch on your most recent plane. Looking from the top view, draw a vertical centerline through the part that intersects the origin. Mirror about this line using the dynamic mirror tool (it is faster to draw if you mirror), and draw/dimension a sketch exactly as you see in the following picture. The thickness of the cross on both sides is 0.6mm. Use the extruded cut command on this sketch and extrude your cut 16mm deep.



9. Now that you are well versed in the art of creating planes, create another plane resting on the bottom of your model (where the shadow is). On this plane, insert a sketch of a circle, concentric with the axis of the entire model (in other words, make the circle centered). Smart dimension the circle to 4mm. Extrude this circle downward 13mm. Your model should look like the right-most picture.

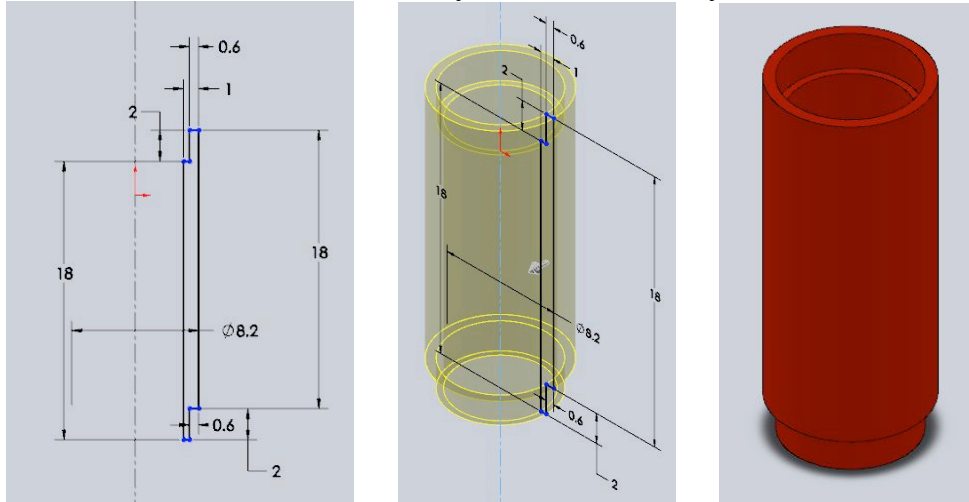


10. The last step is to add a cosmetic thread to part of the last cylinder you created. Insert a cosmetic thread (Insert/ Annotations/ Cosmetic Thread) and select the bottom-most *edge* of your model as the reference edge. Set the depth to 11mm and the minor diameter to 3.3mm. After you click the check mark, make sure the option to view cosmetic threads is enabled (right-click Annotations/ Details). Your complete slotted collet should now look like the last picture.

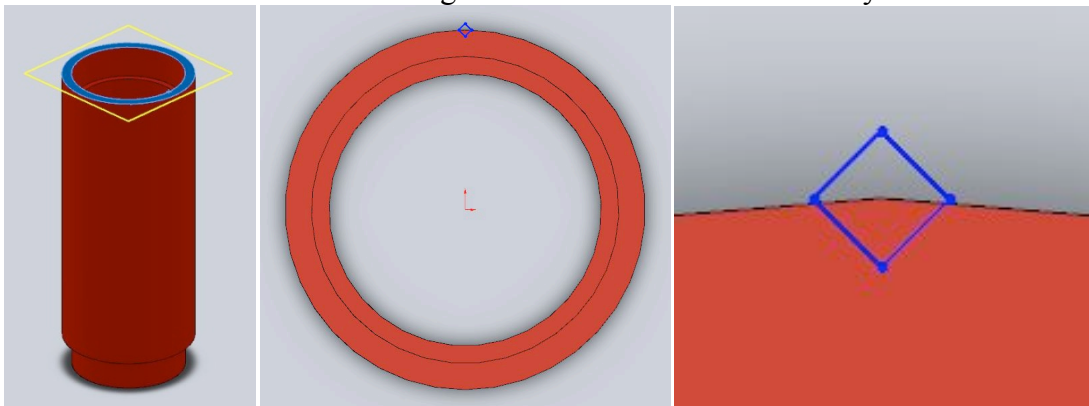


Part II: Rendering the Collet Jacket

1. Open a new part file, and again make sure your overall document standards are set to ANSI and your units are set to metric (mm, g, s).
2. Again, give your part some character. Set its appearance to a different color than your slotted collet so that you will be able to distinguish between them in an assembly.
3. The easiest way to start is to insert a sketch into the front plane, then draw a vertical centerline that intersects the origin. Draw and dimension a sketch that looks exactly like the left-most image below. Once your sketch is drawn and dimensioned correctly, use the revolved boss/base tool to revolve your sketch around your centerline.

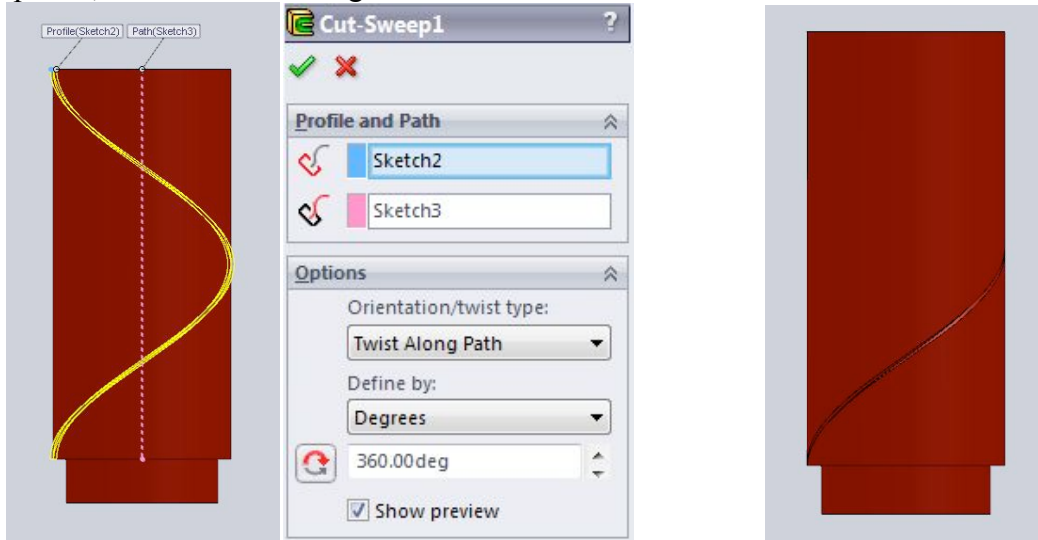


4. Now you will create true knurling on this object. The process is somewhat difficult to comprehend, so follow the instructions very closely. First, create a plane touching the upper-most face of your newly revolved object. Then, change your view orientation so that you are “normal to” this plane, and insert a sketch into the plane. Draw a small diamond like the one in the picture below. Make sure the left and right points of the diamond intersect the circular edge of the model. Save and close your sketch.

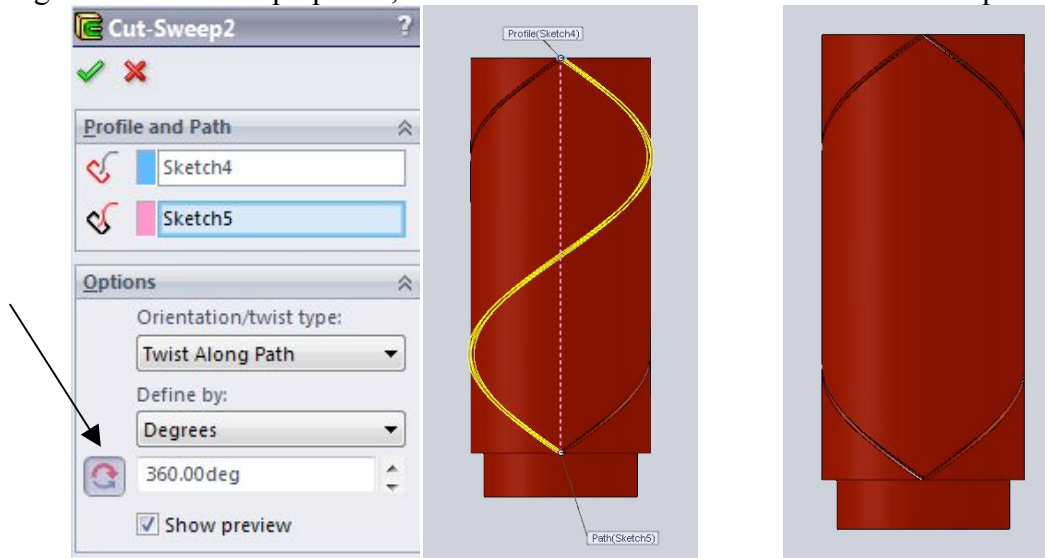


5. Insert another sketch into the right plane and orient your view accordingly. This time draw a vertical line (NOT a centerline) that extends from the top of the model down to the next visible break, through the origin. Save/close the sketch, open the swept cut tool,

and select your diamond sketch as the profile and your centerline as the path. Also, under options, select “Twist Along Path” and define the twist amount to 360°.

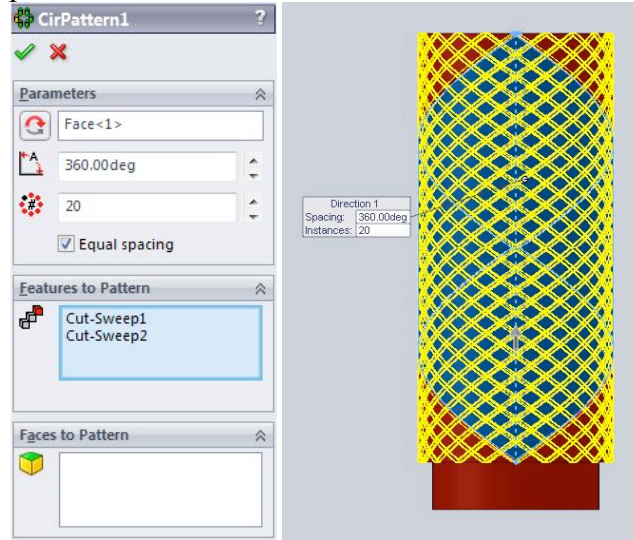


- Now we will create another cut identical in every respect to the one you just made, except it will twist in the opposite direction. For some reason, Solidworks enjoys the feeling of making your life miserable though, so the mirror entities tool does not work in this case. You must repeat steps 4 and 5 with two minor modifications. Instead of drawing the diamond again, use the convert entities tool to copy it to the plane you create. When you get to the cut sweep options, make sure to *reverse the direction* of the swept cut.



- This is the step that would make your computer cry if it had tear ducts. In fact, you should save your part right now because your computer may decide to have a nervous breakdown and freeze up on you in response to the command you are about to give it. Open the circular pattern tool to pattern the swept cuts you just made. Modify your options to look like the image provided, then click the green check mark. Upon clicking the check mark, you may be inclined to indulge in the act of making a sandwich. If you

have no bread, worry not! You will have plenty of time to run to the store to buy a loaf before your computer is done with its calculations.



8. Assuming nothing goes wrong during the seemingly endless span of time it takes your computer to render your model, your finished collet jacket should look like this. Marvel at its beauty, and thank your computer for not dying on you.

