SW Tips/Tricks

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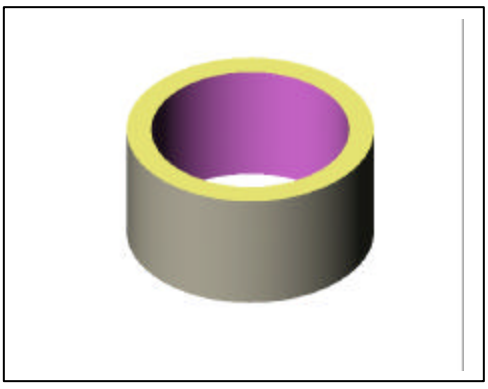
Creating Holes Specifying the Wall Thickness

Capturing design intent within the model and communicating that with others users is one of the most useful aspects of SolidWorks. What follows is a description of four different ways to create wall thickness and a part. It is always better to know more than one way to do the same thing, because if you get stuck with one method, you can revert back to a different method. So with that in mind let's look at for ways to create wall thickness in a part. The first is the most obvious, shelling. Simply select the surface or surfaces you like to remove and use the shell feature to remove material leaving only the wall thickness. As you double click on the feature, SolidWorks will highlight the feature dimensions, in this case, wall thickness. The second method is to use in equation to specify the wall thickness. You simply would sketch the outside of the cylinder, specifying that diameter. Then for your next feature, create a hole through the center of the cylinder. In the sketch for the hole, add a dimension for the diameter of the circle, then add a new equation that makes that diameter equal to the outside diameter minus the quantity to times the wall thickness. Then end result to this would be an outside diameter that controlled the inside diameter, according to the wall thickness value that you included in your equation. The third method is a little trickier. As you draw the inside diameter circle, do not sketch it concentric with the outside. What you need to do is sketch it offset from the center of the outside diameter. Then you can put a dimension from the circumference of the outside diameter to the circumference of the inside diameter. Then right click on the dimension each is created, go to properties, and then pick minimum for both the arc conditions. The last method is probably the one you thought of first. Just open a sketch, and offset either the surface or the outside diameter edge, and pick the offset command from the sketch tool bar.

continued on page 2

INSIDE THIS ISSUE



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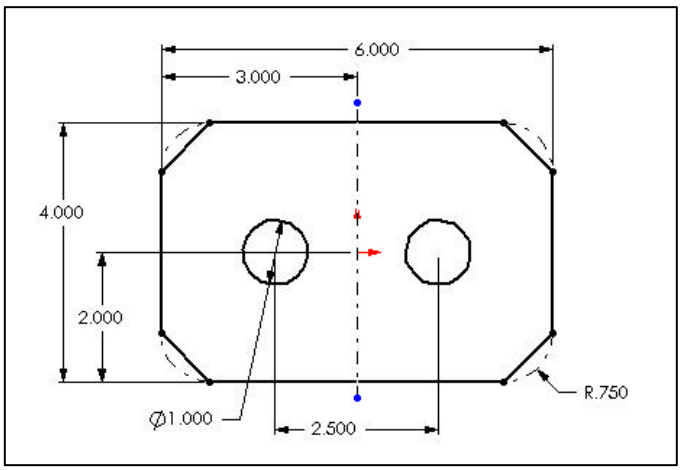
…continued from page 1

Dimension Round Off Error

This is something I see as I'm teaching all the time. For instance, if you have previously said the number of decimal places under the grid/units tab to two decimal places. Sketch a rectangle that just happens to turn out 5.003 inches long, and you dimension the rectangle, the dialog box for confirming the dimension will only display two decimal places. Therefore not knowing the dimension is actually slightly longer than 5 inches (5.003), the dimension is accepted as is. You've actually created a model that is not exactly 5 inches long, even though, the dimension dialog box seems to have specified exactly 5 inches (5.00). One way to avoid this problem is to always type in an exact number in the dimension dialog box. It is not necessary to type in the entire 5.000 each time. In this case, you would only have to type in a five and then press enter. SolidWorks would take this as 5.000 exactly.

Parametric Sketch Chamfers

SolidWorks does not create parametric sketch chamfers the way it does with sketch fillets. If you have the need for 45-degree chamfer and you would like to put them in the sketch, go ahead and put radii in your sketch at each corner you would like the chamfer. Then select all the arcs and change their properties to construction arcs. For each chamfer, sketch a line between each arc endpoint. This will create a 45 degree chamfer at each fillet. Now you can not change the value of the radii, and the chamfers will follow the arc.

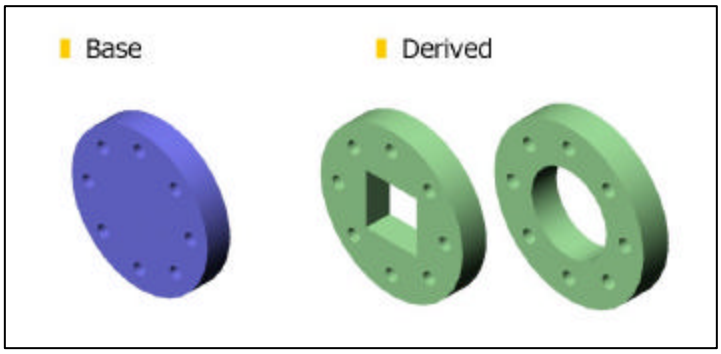


By the way, when you are adding the sketch fillets and you miss one, you re-invoke the Sketch fillet command, and add the one you missed, SolidWorks adds another dimension for the new radius. To get all the fillets under one dimension, go into the sketch, delete the extra fillet dimension, and add an equal relationship between the previous arcs and the last arc. This is what SolidWorks is doing when it created the fillets in a group. You are just going back and adding the relationship to the last arc. Now when you change the dimension, all the arcs change together, or for this tip, all the chamfers.

Base Part - Derived Part

The base part - derived part relationship can be utilized in many different ways. One way use this technique is when you have a family of parts that have several common features, but have some portions of their geometry that are different. This would be the case of a flange with several different center options. You could create the base part with all the common features. Next create a new part. This new part will be considered the derived part. For the first operation, go to the Insert pull down menu, then Base Part (not Insert, Base, Part). You can then open the part you previously created as a base part. Once you rebuild the new part, look at the Feature Manager. You notice that there is on feature that represents the entire base part, but you cannot access the individual features. They are accessible only in the base part. If you go back to the base part, modify a feature, rebuild, then go back to the derived part, the modifications are incorporated. If you consider the flange example, once you insert a base part into several derived parts, you can continue to add any features to the derived flanges. You will not have access to the original base part features from inside the derived parts, but let's say

continued on page 3



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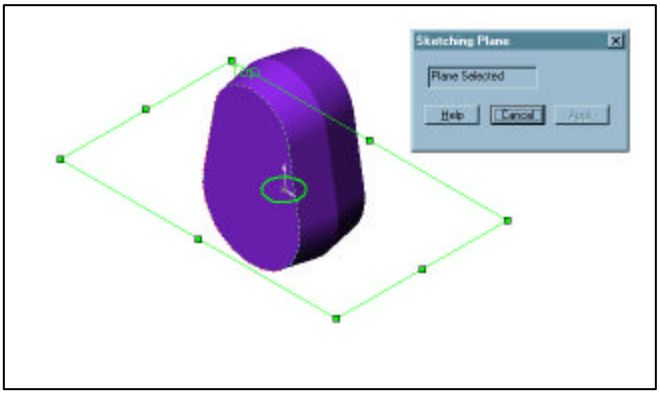
you want to change the number of holes in the bolt pattern. Just go back to the base part, change the circular pattern definition, then go back to the derived components and they will all have changed. Another example of when to use a base part - derived part relationship is in an enclosure that will be designed as one piece, then later split into several pieces. The single piece design would be considered the base part, and each of the pieces would be made as derived parts. Then go to each of the derived parts, cut the enclosure and add joggles or lips that will be used to mate the individual pieces. Then in an assembly, you can mate the individual derived parts together to document the enclosure. If you want to change the overall size of the enclosure, open the base part, modify the features, and the derived parts will update accordingly. You may think this is the same as taking the base part and performing Save As several times to get the pieces, but the difference is with the base part - derived part relationship, the pieces are always related to the base part, and will update automatically.

Mirror Construction Geometry

There are two ways to mirror construction geometry. The first is to draw the mirror line, then place the sketch and mirror mode (this is where you will see the to hash marks at each end of the center line), then proceed to draw the construction geometry. As you draw the first construction geometry, the mirrored geometry will automatically be created on the opposite side of the mirror line. The second method is to draw the entities on the first side as non-construction geometry, mirror the geometry using any method you like. Control select all of the geometry that you would like to be construction, then right click and pick properties, and construction geometry.

Editing A Sketch Plane

Editing a sketch plane is a method for moving the sketch to a different flat surface or plane. This new surface or plane need not be parallel to the original so proceed with caution. In this non-parallel example, the sketch will relocate but the relations or dimensions in the sketch may dangle if the entities can no longer be found. To invoke this command, it is first necessary to find the sketch you want to relocate. Right click on the sketch (not the feature) and select the second choice, Edit Sketch Plane. Next the sketch and it currently related surface or plane is highlighted in the graphics area. This highlighting is also a quick way to check which surface or plane the sketch is located on. Simply cancel the dialog box at this point if you do not wish to relocate the sketch. If you do want to relocate the sketch, pick a new flat surface or plane on the model. After picking the new surface or part, the Apply button will be available. Pick the Apply button to complete the command. -(O||||O)-



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Calendar of Events

SolidWorks World 2001
Orlando, Florida
Week of February 11, 2001

San Diego SolidWorks User Group
Digital Dimensions, Inc.
3934 Murphy Canyon Road Suite B-100
2nd Wednesday of the Month at 7:00pm

Group discussions, tips, and ideas. Various beginning and advanced topics presented each month. Arrive early for pizza/soda. For info call Phil Sluder at (619) 460-0216

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