

# SW Tips/Tricks

Volume 2, Issue 6

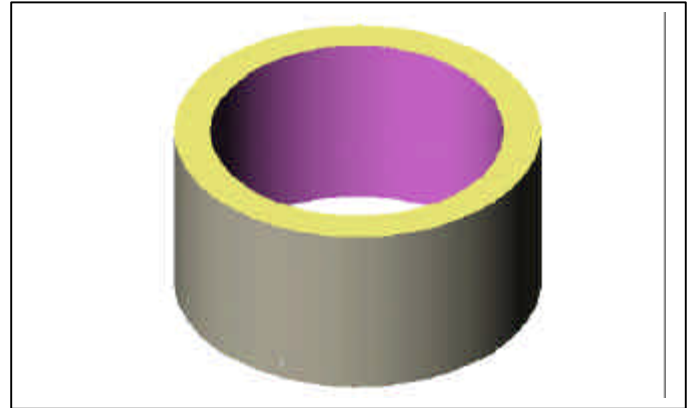
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November / December 2000

## Four Ways to Specifying Wall Thickness

Capturing design intent within the model and communicating that with others users is one of the most useful aspects of SolidWorks. 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 four ways to specify the wall thickness of 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 Shell 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 two times the wall thickness. Then 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.

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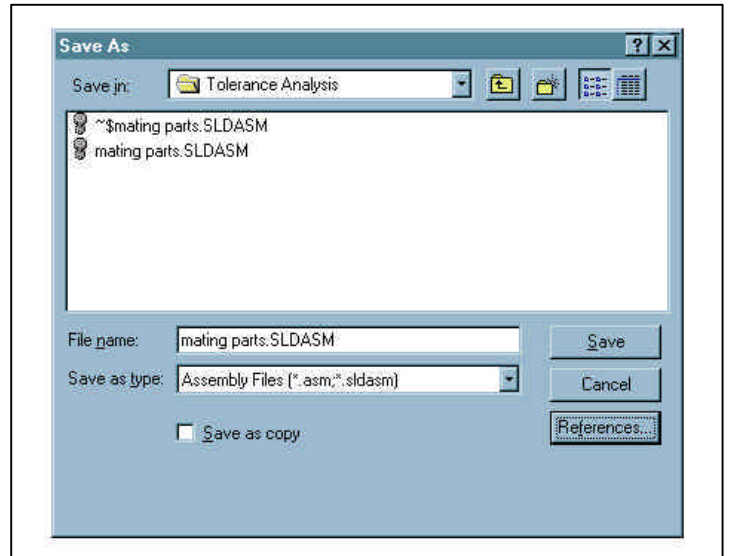
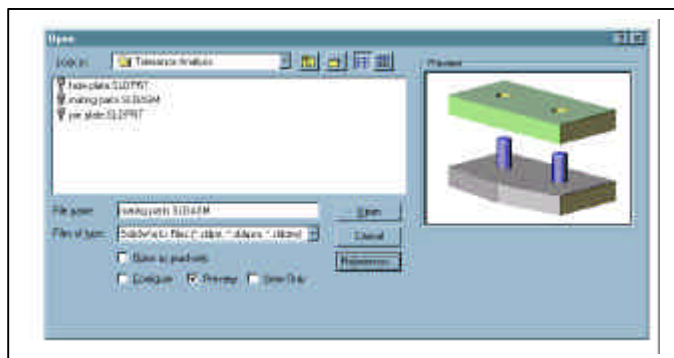
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The third method is a little trickier. As you draw the inside diameter circle, do not sketch it concentric with the outside. What you need to do is sketch it offset from the center of the outside diameter. Then you can put a dimension from the circumference 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. After adding the dimension, then go back and add the geometric relations needed to get the circle concentric to the outside.

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. By now, you probably have thought of a few more ways to specify the wall thickness.

## Two Uses for the References Button

The first is in the Open dialog (see figure). As you open an assembly or drawing, you can change the part and assembly files that are



used to create the assembly or drawing you are about to open. SolidWorks will only allow you to change the components at the top level of an assembly, so if you are opening an assembly that contains subassemblies, the parts in those subassemblies will not be listed in the Edit referenced file locations dialog. To change the assembly or part files used in a subassembly, you need to open the subassembly file.

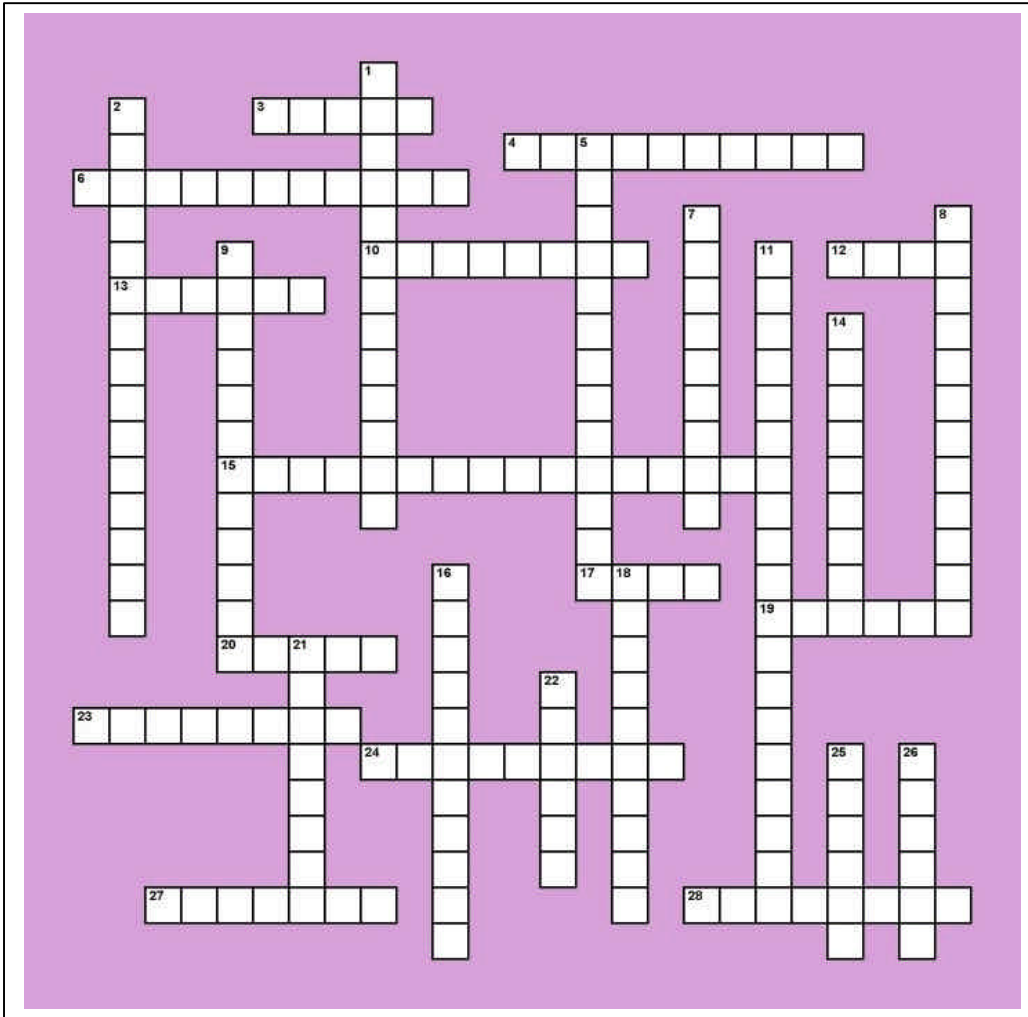
The second is in the Save As dialog (see figure). This is used when you want to save the assembly or drawing as a different name, and would like to save the subassemblies and parts used in the assembly or drawing as a different location and/or filename. This time the Edit referenced file location dialog will list all the assembly, subassembly, and part paths and filenames.

## Revolved Dome

In sketch geometry you cannot make an arc

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## Not Enough Solid Modeling Challenges?



### Across

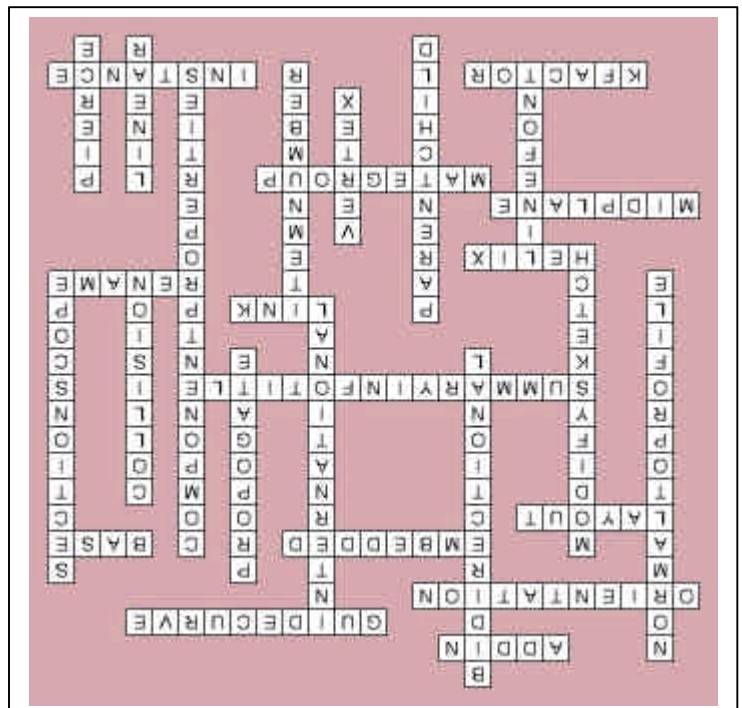
3. 3rd party (3,2)
4. Loft element (5,5)
6. Trimetric
10. Design table
12. First
13. Assembly sketch
15. Part number column (7,4,5)
17. Same value
19. Slow double click
20. Spiral
23. Two directions
24. Can have multiple (4,5)
27. Stretch (1,6)
28. Copy

### Down

1. Associativity
2. End condition (6,2,7)
5. "I" in IGES
7. Follow edge
8. Specify components (7,5)
9. Move origin (6,6)
11. Swap assemblies (9,10)
14. Stop or highlight
16. Dependency (6,5)
18. Balloon (4,6)
21. Phantom (4,4)
22. Point
25. Circular / \_\_\_\_\_
26. Poke

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perpendicular to a line. You can however, making an arc tangent to a line. So in order to create a dome that has a partial spherical surface on one side, you can create an arc that is tangent to a horizontal construction line. Then you can create another vertical centerline for the purpose of revolving the sketch you just created. Since you have more than one center line in the sketch, SolidWorks does not know which center line you would like to use for the revolve. So just make sure you highlight the vertical centerline before you try to revolve the shape. -(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**

**2<sup>nd</sup> 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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