

# SW Tips/Tricks

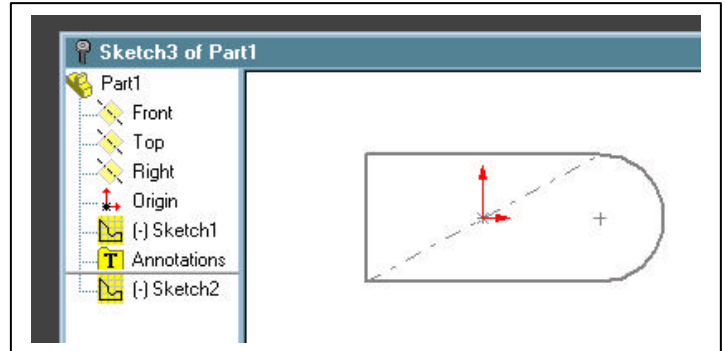
Volume 2, Issue 5

www.triaxialdesign.com

September / October 2000

## Sketch Closed by Rebuild

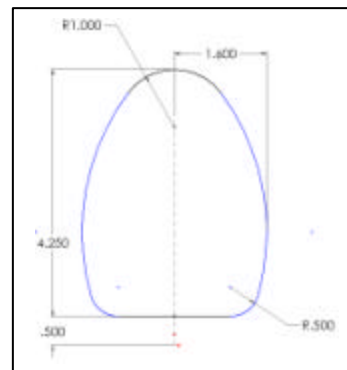
This is a common mistake for a new user. Once the sketch has been opened, and a few entities are created, the user picks on the rebuild icon to clean up the screen. After realizing the sketch has been closed (noticing the pencil icon is not depressed), the user mistakenly picks the same plane as before and the open sketch icon again. Thinking the same sketch is now open, the user will try to add to the previous sketch, not realizing a second sketch is now open. Dimensions from the first sketch are not visible at this time, nor do the sketched lines appear in the usual blue and black colors. A glance



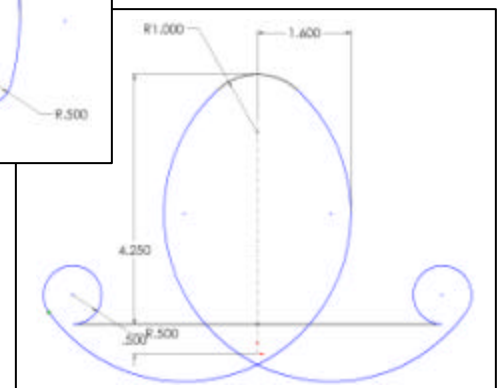
at the Feature Manager design tree and the Title bar reveals the second sketch has been started. To correct the situation, exit the second sketch. Single pick on the sketch icon for the second sketch in the Feature Manager design tree, hit the delete key and confirm the deletion. Right click on the first sketch in the Feature Manager design tree, and pick "Edit Sketch" from the menu. The user is then back in the first sketch and can continue to edit.

## Nonsense Sketches

Nonsense sketches occur when you drag or change a dimension in a sketch and even though the relationships are maintained, they are not what you had in mind for the



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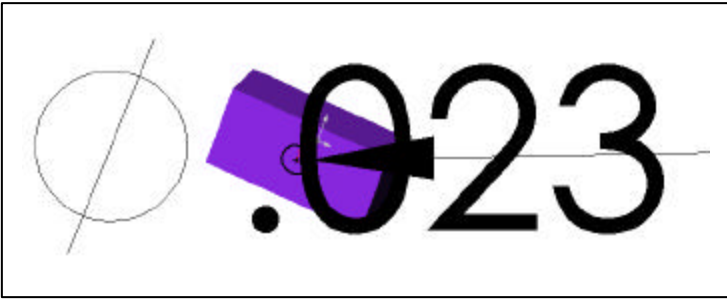
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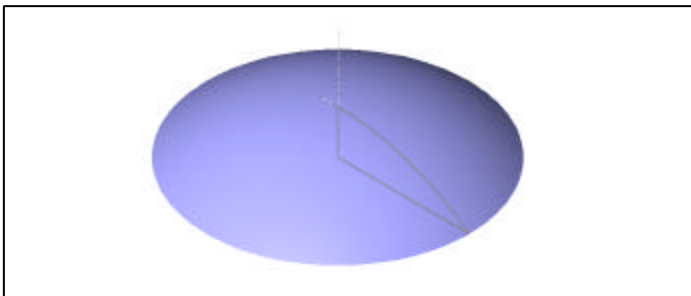


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sketch. The tangency you intended between a line and an arc now has jumped the line to the other side of the arc. It is still tangent, but not what you originally intended. The solutions to this situation are as varied as the problem, but one helpful hint is to use the Undo command, and try to figure out additional parameters to add to prevent the ambiguous situation. The nonsense sketch usually occurs at a specific step in the sketch creation process, and this is one place (in a sketch) where the undo command works well.

### Always Display Text at the Same Size

This frustrating situation is corrected with an option in the Tools, Options dialog box. Go to the Document Properties tab, and then pick the Annotation Display branch under Detailing. If you check the “Always display text at the same size” option, dimension text in the sketches will always appear the same relative size to the screen. It will always be the same number of pixels high, regardless of how far you are zoomed in or out. This will avoid the dimension text covering the entire display preventing you from picking the geometry. Alternately, right select the Annotations icon in the Feature Manager design tree, select Details, and check the option.



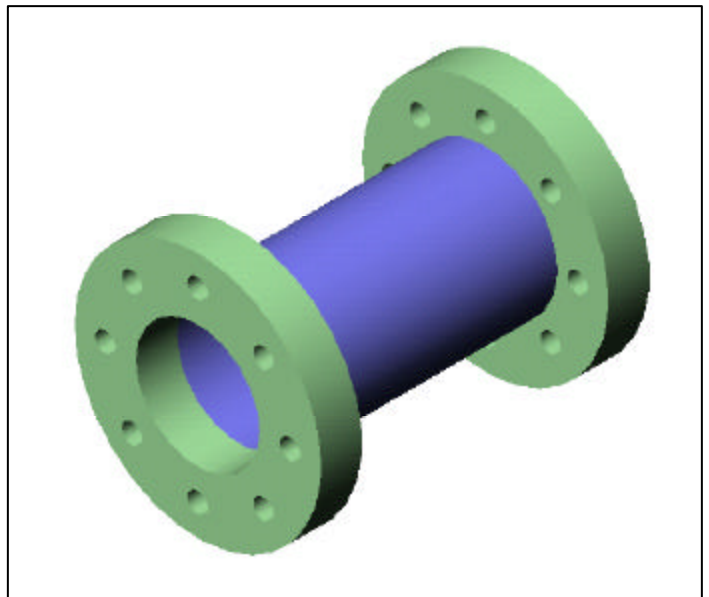
### Revolved Feature vs. Circular Pattern

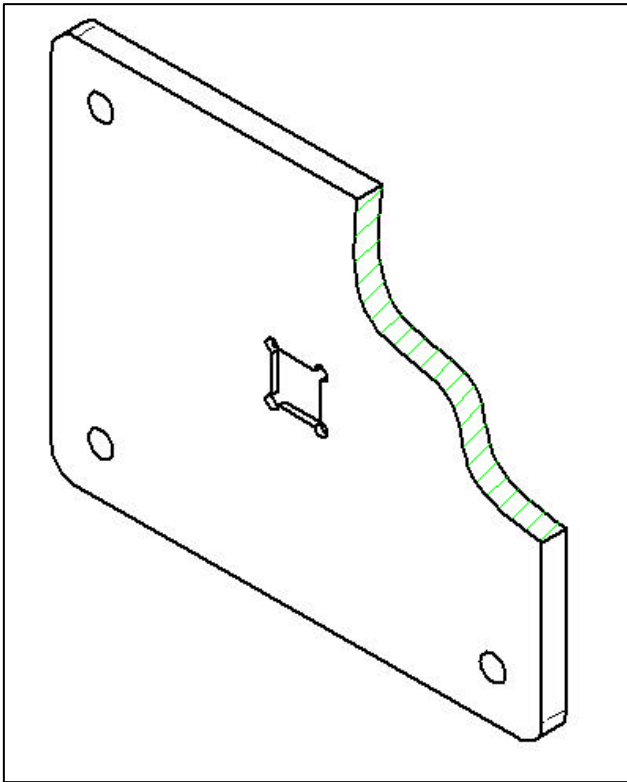
The items required to perform these two commands are often confused. Revolving a boss or a cut feature requires two entities in the same sketch, a centerline and a shape. The centerline in the sketch is needed to define the center of the rotation of the feature (remember a centerline and construction lines are the same types of entities). The shape you want to revolve is also required to be in the same sketch as the centerline. A circular pattern copies one or more features around an axis such as a bolt circle of holes or spokes on a wheel. In contrast, a circular pattern needs an Axis feature (reference geometry), temporary axis, linear edge, or angle dimension to define the axis of the pattern. What is often confused is that the centerline entity is not the same as the axis feature. The axis feature is not even contained in a sketch.

### Assembly Features

Assembly features are a very powerful tool when dealing with secondary machining processes to weldments or assemblies. Insert, Assembly Feature, Cut can be used on a completed weldment for such operations as match drilling, machining multiple pieces to a specific length, or boring inside a weldment to achieve a precise internal

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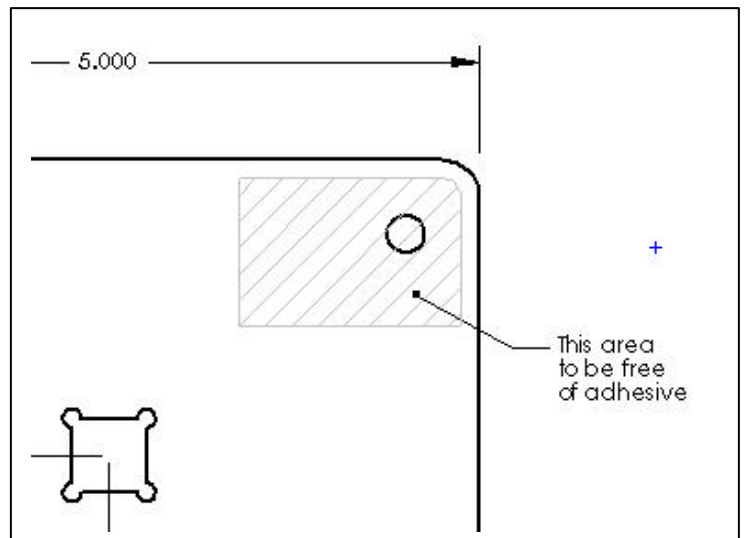


## Area Hatch

Area hatch can be used in a drawing to hatch either individual surfaces of your model represented in the drawing, or hatch an area defined by sketched entities. For example, you can create a section cut through your model and show the isometric view of the model in a drawing with hatching on the walls or surfaces that have been cut. After inserting the view of the model, pick the surface (or control select multiple surfaces) and pick the Insert pull down menu, then Area Hatch. You can change the type of hatch by right selecting the hatch itself, and then properties. You can also hatch a 2D area of your drawing to define an area where a specific criterion applies. Use the sketch tools to create the area. The offset and convert entities can also be used. Any dimensions you do not want to appear in the drawing can be hidden by right selecting the text of the dimension and then pick Hide dimension. After creating the entities that you would like to contain the hatch, select all the entities and pick the Insert pull down menu, then Area Hatch. -

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diameter. The features are added to the assembly of the parts, not in the individual part files, which is what you want. If you specify the Feature Scope, you can specify which component instances the Assembly Feature will effect. To proceed, in your assembly make sure you are in Edit Assembly mode, and from the Edit pull down pick Feature Scope. Specify which components and instances will be effected (remember you can right select in white space and Clear Selections to clear the dialog window). Then open a sketch in the assembly and create either extrude or revolve geometry. When you are ready, go to the Insert pull down, then Assembly Feature, Cut, and either Extrude or Revolve. After the feature is created you can locate it near the bottom of the assembly Feature Manager design tree. If you right select it, you can edit this feature just like any other feature. There is also a Feature Scope choice on the right select menu, to re-specify the components or instances the scope of the feature. This assembly feature can then be patterned and/or other components mated to it.



If you would like to receive issues of SW Tips/Tricks please provide us the following information by:  
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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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