SW Tips/Tricks

Volume 2, Issue 3 www.triaxialdesign.com May / June 2000

Breaking In-Context Surfaces

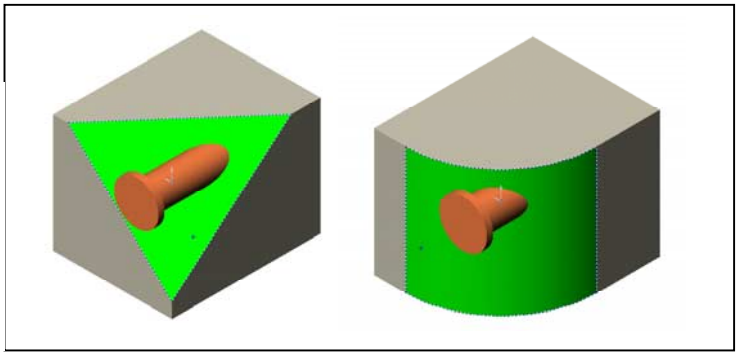
The challenge with this tip is to understand that you have a problem. The figure below shows two different assemblies and in each case the pin part has a boss extrude feature with the end condition is up to surface. The surface is located on the rectangular part. If you choose to keep the in context relationships between the parts in the assembly, there is no problem. If you do choose to break those in context relationships, the problem is how do you break the in context relationships between these two parts.

If you open the pin component in the first example, the pin is being extrude up to a flat surface that is skewed. To remove that in context feature, you need to duplicate this skewed surface in pin part file. Where is it located? It was easy to extrude up to it in the assembly, but now in the individual pin part file, the surface to extrude up to does not exist. The pin part still has the same shape, but you have to realize that the surface is still based on the assembly, so you cannot just create a plane with an offset of zero on top of the existing surface in the part. You would have a circular reference, the plane would be based on the surface and the surface would be extruded up to the plane. Where we need to do is duplicate the skewed surface of the rectangular part with a plane in the pin part file. That is the description of the problem. Now here is the solution. Open the pin part file, and open a new sketch on the flat surface that the feature is extruded up to. If you drew a line normal to the new sketch plane starting at the origin of the new sketch it would pass directly through the origin of the pin part file. So if we duplicate this line, we can construct a plane perpendicular to a curve (the line) passing through an endpoint of the line (the origin of the new sketch).

To do this, place a point entity in the new sketch. Then add a relationship of coincident between the origin of the new sketch and the point entity. Close the sketch, pick the point entity that you just created and go up to Tools,

continued on page 2

Inside This Issue



TriAxial Design and Analysis

SOLUTION PARTNER SolidWorks

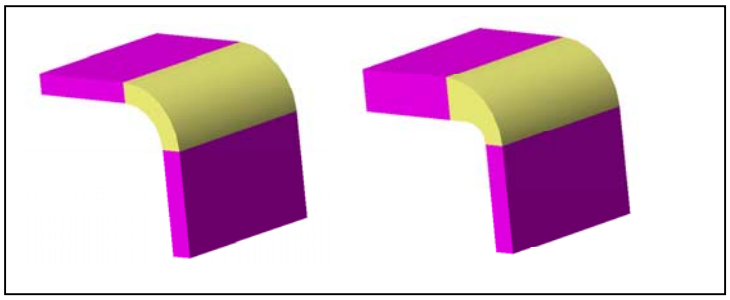
Complimentary copies of SW Tips/Tricks are independently published for SolidWorks users worldwide.

All back issues can be downloaded from www.triaxialdesign.com

…continued from page 1

Measure. The Measure dialog box shows the coordinates of the point. Now go up to the Insert pull down menu, and pick Curve, Curve Through Free Points. This dialog allows you to create a curve passing through a set of points. For the first point use 0,0,0 and for the second point use the coordinates listed in the Measure dialog box for the point entity. Press OK and now you should see a straight line going from the origin of the part to the point entity that you created. Once this line is created, you can delete the sketch containing the point entity. Now create a plane perpendicular to that line passing through the end opposite the origin of the part. You now have a plane in the same location as the original skewed surface. There is a possibility that you will get some rounding off in the Measure dialog box, so adjust the number of decimal places to display on the grid/units tab as required. Now you can go back to the original feature in the pin part, edit the definition of the extruded feature, and change the selected surface to the new plane you just created.

The other example shows the same pin, this time extrude up to a curved surface. To break this in-context relationship we cannot just create a new plane because we are extruding up to a curved surface. We need to export the curved surface and then import it back in as a reference surface. To do this, open the pin part, select the curved surface and then go up to File, Save As, drop down the file type list to IGES file, name the file, and then picked OK. It will ask you if you would like to save just the selected surface, or the entire solid body. In this case save just the selected surface. Now import that surface we just saved back into the pin part file. To do this, go up to Insert, Surface, Imported, and browse for the surface you just saved. Now you have a curved imported surface in the pin part file, that is a duplicate of the curved surface that was extruded up to. Once again you can go back and edit the definition of the original feature in the pin part, and change the selected surface to the new curved surface you just imported.



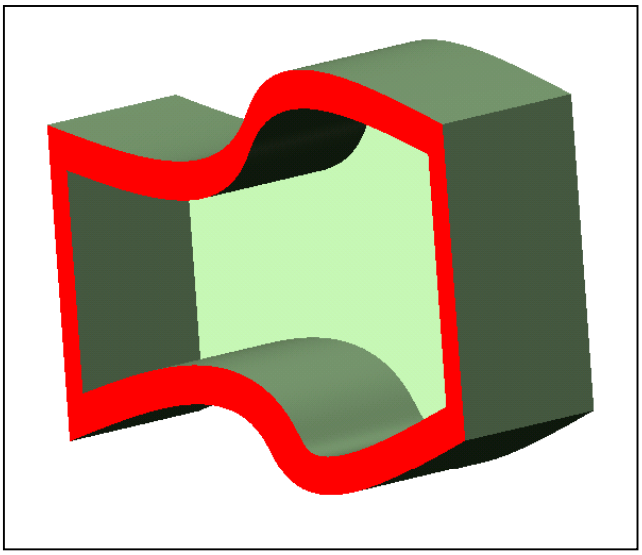
Multiple Thickness Shelling with Radii

Multiple thickness shelling is straightforward if there are no radii between the two surfaces that are to be different in thickness. If the thickness was consistent throughout the part, the radii can be placed on the edges prior to shelling. At the problem arises when you would like to model two surfaces that are to be different thickness with an adjacent radius between the surfaces. When you specify the thickness for each surface SolidWorks expects you to also specify the thickness of the surface for the radius. This means that if surface “A” is to be 2 mm thick and surface “B” is to be 3 mm thick, what thickness should you specify for the surface of the radius. Depending on what thickness you specify for the surface of the radius, the modeling results will range from an odd shape to failed geometry. To solve this all we do is place the radius on the part after the multiple thickness shelling has been done. Now visualize the same part with the 2 mm thick surface intersecting the 3 mm thick surface without a radius. Now add an outside radius and an inside radius to your model. The end result is a nice smooth transition from the 2 mm thick surface to the 3 mm thick surface (since both radii are tangent to the inside and outside surfaces of the shell respectively).

Offset from Surface and Offset Surface

This can be confusing, and may affect wall thickness of a part if you do not realize what the command is doing. The example was created by sketching a rectangle on a plane at the horizontal center of this part. Then a cut-extrude using the “offset from surface” end condition was added, What this did is and the cut to the specified distance from

continued on page 3



…continued from page 2

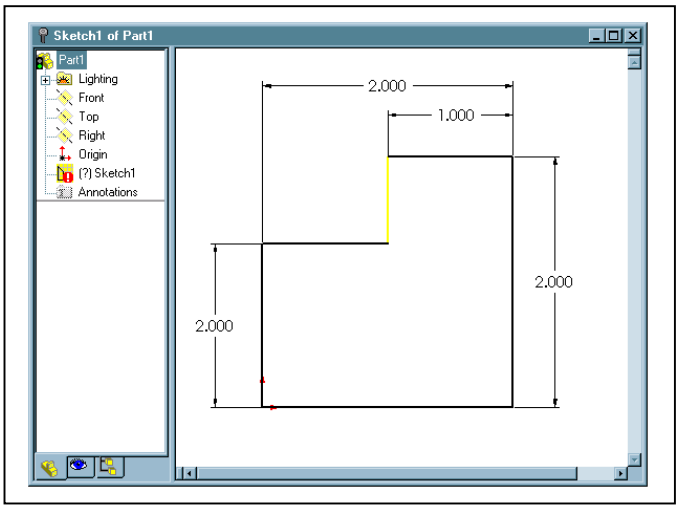
the upper surface, measured perpendicular to the sketch plane. This means that as you go along the curved surface you are extruding up to, the wall thickness of the part measured perpendicular to the surface is not consistent. What is consistent is the wall thickness measured perpendicular to the sketch plane. This may or may not be what you intended. In order to get a consistent wall thickness measured perpendicular to the surface you need to use another end condition, but first you need to create an offset surface. By definition, a surface is reference geometry that you can use in your modeling. The surface is parametric and tied back to the original surface you offset and will change if that original surface changes. Simply select the surface you want to offset and then go up to Insert, Surface, and Offset. Specify the distance and direction you would like to offset. SolidWorks will than create a surface for you to use in your cut extrude. Edit definition for the cut-extrude and change the end conditions specifying “up to surface”. Now pick the surface you just created and press OK. The model will rebuild and now you can see the wall thickness is consistent when measured perpendicular to the curved surface. You can also hide the reference surface you just created by right clicking the surface either in the graphics area or off of the feature manager, and selecting Hide.

Extend with the Trim Tool

If you’ve been around SolidWorks for a while, you probably already know this one. You can actually extend using the trim tool. This really isn’t the trick, it’s just the way you used to have to extend geometry. All you do is pick the trim tool, then select and drag the entity you would like to extend. As you drag with the left button, the first entity that you have selected will highlight in red. Now simply drag the cursor to the second entity you would like to extend to. When the cursor is over the second entity, you should see a preview of the extension. Release the mouse button, and the first entity will extend to the second entity.

Invalid Solution

Dimensions or relationships that cause an entity to become geometrically impossible or zero length will cause a change in the status of your sketch to “invalid solution”. If you look at the figure you can see that the two vertical dimensions are both the same therefore the yellow line would become zero length if the sketch were solved. Instead of solving the sketch, the entity that has effected is highlighted in yellow. As soon as you change one of the dimensions to something different, the sketch will be able to find a valid solution. One other tip worth mentioning here is if you cause multiple entities to become invalid, and you try to erase one of them, SolidWorks will not let you. Control select all of the



If you would like to receive issues of SW Tips/Tricks please provide us the following information by:
Phone (619) 460-0216, Fax (619) 460-0902, or
Email sluder@triaxialdesign.com

Name _____

Address _____

City, State, Zip _____

Email _____

Calendar of Events

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

TriAxial Design and Analysis
4817 Palm Avenue Suite K
La Mesa CA 91941-3840

ADDRESS CORRECTION REQUESTED