

SW Tips/Tricks

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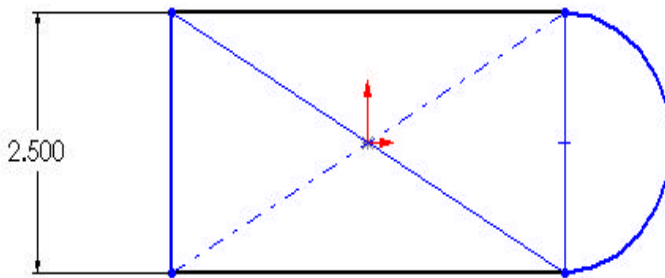
More! More! More!

Phil Sluder

TriAxial Design and Analysis

Everything is going great, and this issue of SW Tips/Tricks includes several more useful ideas for improving your solid modeling techniques. I hope you find each issue useful and learn something new. Thank you for all of the pleasant comments about this newsletter.

Please drop me a note at sluder@triaxialdesign.com if you have any specific topics you would like to see presented. -(O|||O)-



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Three Entities with a Common Endpoint

You have probably got clues to this problem before, but never realized the clues were visible. This example shows several lines in the sketch. The clue is the blue diagonal line which is thinner than the others. You may have thought this was just a hiccup of your graphics card, but instead, SolidWorks is pointing to the problem of three entities sharing a common endpoint. If you continue and try to extrude this sketch, you will get the error message, with the thin blue diagonal line highlighted in green. SolidWorks is guessing which line is the third entity, and therefore not allowed as part of the closed shape. It may not be the one you want to eliminate. Use your judgement for your specific part, and either delete the highlighted line, or pick the line that is the problem. The third and extra line should be one of the other lines that share the common endpoint.

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 its 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.

Tolerances in the Sketch

Sketches can convey the design intent by the way they are dimensioned. For example, all dimensions form one side of the part to prevent tolerances from adding together, or dimension between two hole centers to

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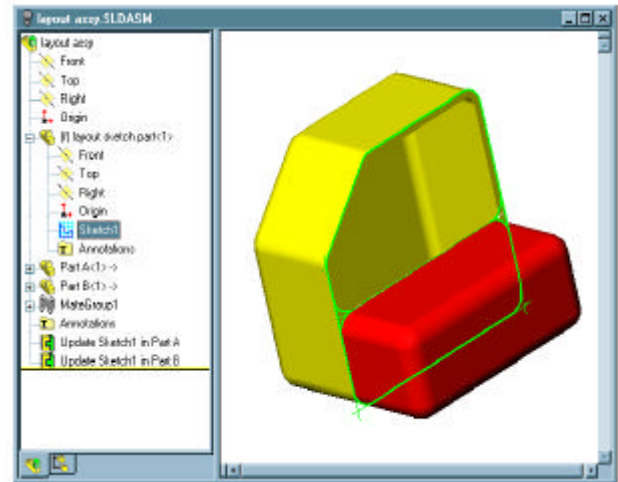
assure the components fit together. Another thing you can do in the sketch is to actually specify the final tolerances of a dimension, in the sketch. The nominal dimension is still the one SolidWorks will use to build the model, but the tolerance will be there when it is time to view the feature dimensions on the model, or import the item to the 2D drawing. Either way the tolerances that were intended by the modeler are conveyed downstream. Right select the dimension you wish to tolerance, and pick Properties. Then find the tolerance button, and proceed to add the appropriate tolerance. After selecting OK, the tolerance will then be added to the dimension.

Create Silhouettes

Create Silhouettes is related to a setting in the Tools, Options dialog box under the General Tab. This setting is Enable Silhouettes and can be changed only when a sketch is not open. First a little background on what a silhouette is. If you look a cylinder from the side with the axis up and down, or more specifically open a sketch on a plane that is parallel to the side of a cylindrical feature on a part, there are two types of edges on the cylinder. If you look normal to the sketch, the top and bottom of the cylinder are real

Since most of the time we are not concerned about these silhouette edges anyway, we can uncheck the Enable silhouettes option...

edges. There is actually a corner where the flat top meets the circular portion of the cylinder. The other two edges are known as silhouette edges. There is not a corner associated with the edge, but what you are looking at is just the surface where it goes from visible to hidden, and seems to be the edge of the part. If you have the Enable Silhouette option checked, each time you open a sketch, SolidWorks will search all features for silhouette edges associated with the current sketch. Think about it, if you have 300 holes in you sketch that are parallel with the sketch plane, then it will find 600 silhouette edges. This has a tendency to slow down the modeling process. Since most of the time we are not concerned about these silhouette edges anyway, we can uncheck the Enable silhouettes option and SolidWorks will not automatically search for all silhouettes edges. Then when you are sketching



and you need to find an edge that you know is a silhouette edge, simply right select in white space, pick Create Silhouettes and the edge will be selectable. You should see cursor feedback of a cylinder with a dark edge indicating you are about to select a silhouette edge. This will save a lot of time when opening sketches with numerous silhouettes edges in the part model.

Multiple Assembly Layout Part

In Volume 2 of the SolidWorks Training courseware there is a lesson on Layout Assemblies. Briefly this is the practice of creating one or more sketches in the assembly file that will be used for locating and mating the parts of the assembly. These sketches are not used to create parts or features, just a kind of wire frame to locate key points in the assembly. Combined with the use of axes and planes, this is one method for modeling your assemblies. This works well in a single assembly, but if you want to model a family of similar assemblies, when you perform a save as on the first assembly to create the second, now you have two separate assembly sketches that are parametric, but are no longer the same. If you change one, the other will not change. You will have to update each one separately. One solution to this is to begin the first assembly, and instead of creating the first layout sketch on an assembly plane, insert a new component and create the sketches on the new component's

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planes. Again, these sketches will not be used for creating features, they will be used to locate other components on the assembly. The advantage of this separate part is that it can then be inserted into other assemblies and you will have this one part, controlling the positions of the components in several assemblies. Therefore when you change the one component layout, all of the assemblies will update. A good example of when you could take advantage of this is an assembly of a row of airline seats. The armrest subassemblies are all similar, but depending on the position of the armrest (isle, center, or window) the components used will be different. The armrest subassembly on the isle may have an additional bumper or trim panel, where the center armrest subassemblies do not require these components. The positions and main locating points for each subassembly are similar enough to control the layout with a single layout part. If you modify the layout part sketch, all the armrest subassemblies will also update.

When you need to change the basic shape of the armrest, you can modify the original layout part, and all the subassemblies and components will update.

Another procedure that can be used with layouts in an assembly is to create the outlines of all of the individual components in a single layout sketch. Going back to the example of the armrest, the thermoformed plastic panels are all similar, and have to fit on the same frame, but are slightly different depending on the location of the armrest in the row of seats. The layout part would contain the overall shape of the armrest and frame. When you are ready to create the individual plastic part, create a new part in the armrest subassembly. Open the first sketch (while editing the new component), and convert the entities from the original layout sketches that will be useful in creating the individual plastic part. This sketch can be used to extrude the shape of the plastic part. Then add fillets, shell, and cut as required to get desired shape of the thermoformed part. When you need to change the basic shape of the armrest, you can modify the original layout part, and all the subassemblies and components will update. This also assures you that the plastic parts will fit together, because they are all based on the

same layout part.

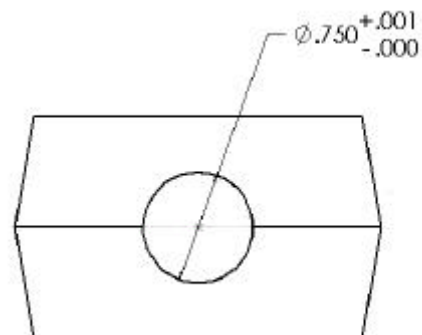
Add Center Planes on Parts for Mating

Open a sketch on a flat surface that represents the front of the part. Sketched two centerline segments at right angles with a common end. Then relate the ends of the segments to external midpoints. Now you have the two centerlines with the end points attached to the midpoint of one vertical side, and one horizontal (top or bottom) side. The shared endpoint is in the geometric center of the part, and will stay there regardless of dimensional changes. Then close the sketch, but leave it visible. I then select the horizontal centerline and ctrl select its center endpoint, click on the Insert, Reference geometry, Planes, then notice Perpendicular to Curve option will have been automatically depressed. This creates a plane perpendicular to the centerline at the shared endpoint. Do this a second time for the second centerline and you have two non-volatile center planes.

Use center mating and axis mating for virtually every part, with the notable exception of fasteners, since it permits easy mating to corresponding reference planes in the assembly. This way, a number of simplified versions of the assembly can be created by suppressing components, and mates don't blow up on you (remember when you suppress the parts you also suppress the mates for those parts).

This last tip courtesy of Ed Salter, General Manager of San Diego Technology Group, Inc.

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If you would like to receive issues of SW Tips/Tricks please provide us the following information by:
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Email sluder@triaxialdesign.com

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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 Joe Nebolon at (760) 931-7975

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TriAxial Design and Analysis

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Let us help you with:

- New designs from your ideas and concepts
- Modeling of your existing parts and assemblies
- Current design projects to free up your personnel
- Software development similar components
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