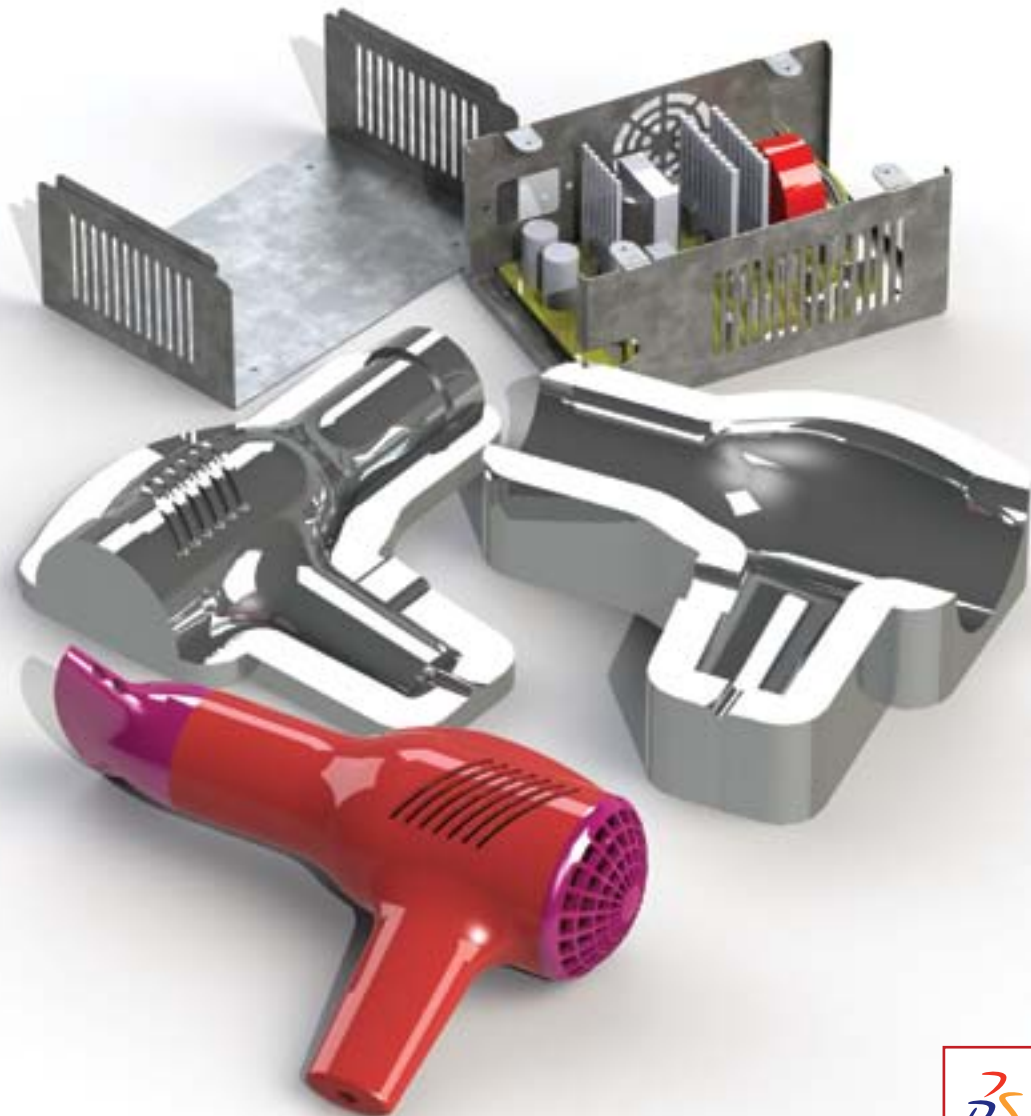


Beginner's Guide to SolidWorks® 2012 – Level II

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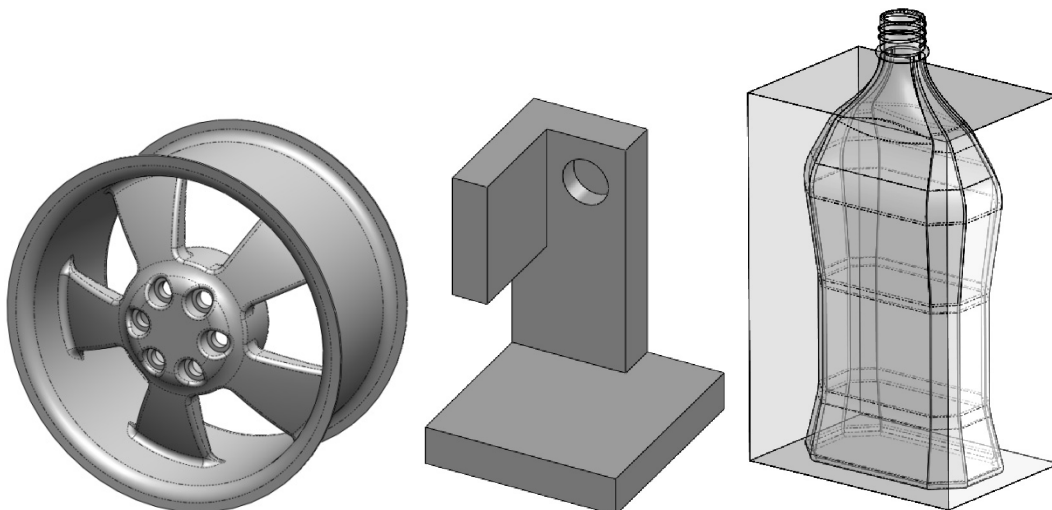
Multi Body Parts, Sketch Editing and Other Tools

SolidWorks offers a number of tools to help us model components easier and faster with powerful options like multi body parts and contour selection. Multi body parts means that we can have more than one “solid” in a part. Up until now we have been modeling parts as single body parts, meaning that each feature either added or removed material to the part, but always resulted in a single solid body. Designing with multi body parts allows us to do things that sometimes are very difficult or even impossible using a single body component like, among other things, combine bodies, remove one body from another or calculate their common volume. Multi body operations are particularly useful when working with mold design and is the essence of the weldments environment (both are covered in this book).

So far we have worked with single and multiple contour open or closed sketches. In this section we'll cover how to use Contour selection to work with sketches with multiple entities sharing an endpoint and/or having self-intersecting lines. We'll learn how to take advantage of these situations and make the best of them.

Another topic that is very important to modeling in SolidWorks is editing parts to fix errors or change components. After all, when we are designing, most likely we'll have to make changes along the way, and sometimes those changes can cause trouble down the road. In the Editing section we'll cover different options available when editing parts, as well as fixing and recovering from errors.

One last topic we'll cover in this section deals with maintaining design intent; we'll learn how to add, edit and delete equations in a part, giving us powerful tools to make our designs more flexible, robust and predictable when making design changes.

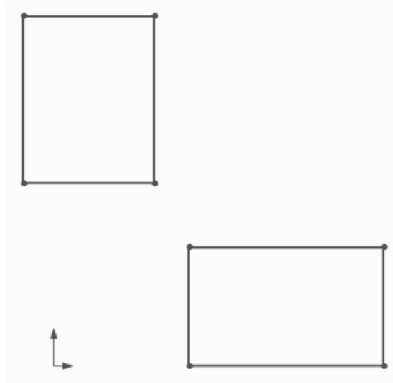


Notes:

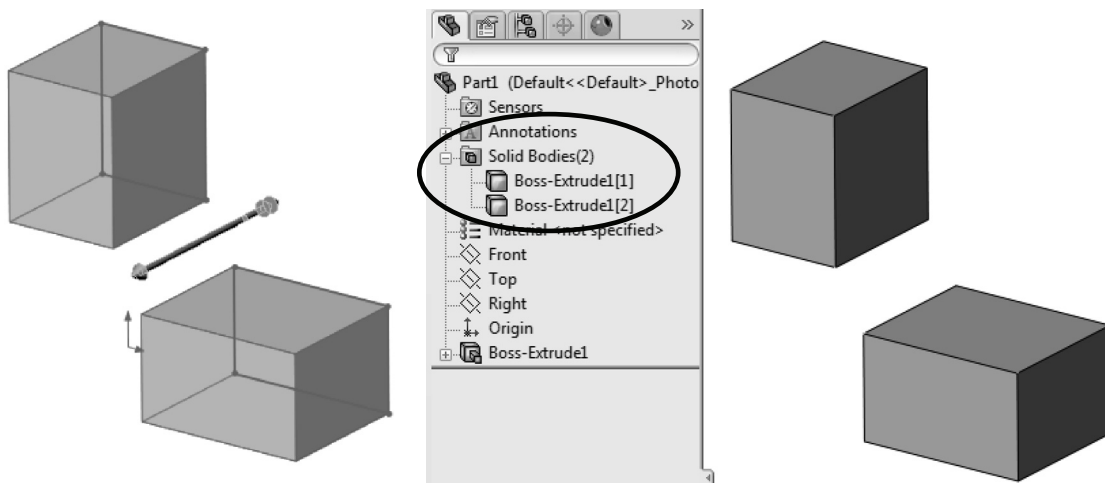
Multi Body Parts

As we stated earlier, multi body parts allow us to do certain operations that would be difficult to accomplish with a single body part. First, we'll talk about how to make multi body parts, and after that how to use them.

1. – Multi body parts are made by adding material that is not connected to the current solid body, in other words, that is purposely not merged to it, or by splitting an existing body. To show how it works, make a new part, and add a sketch to the Front plane as shown. Do not worry about dimensions here as we are just illustrating the concept of how multi bodies work.

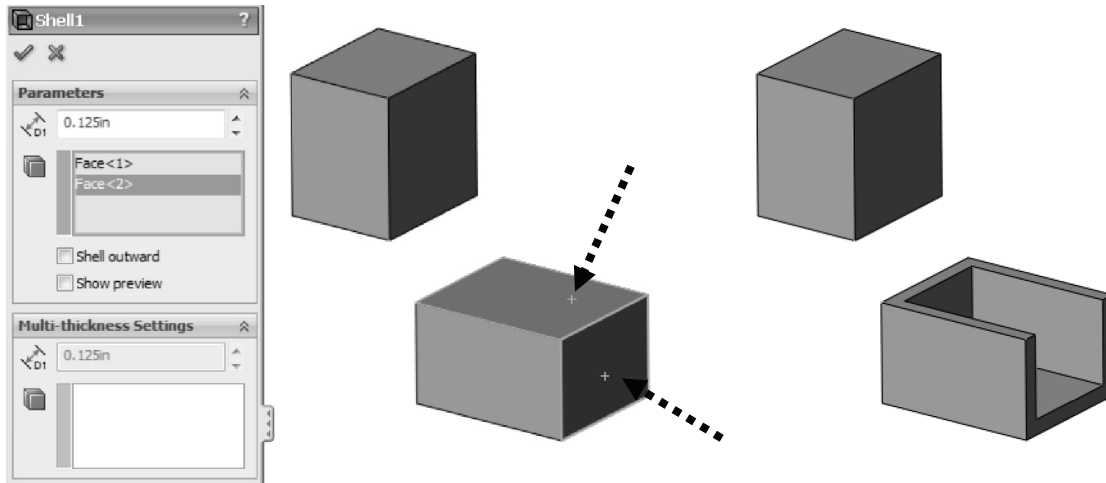


2. – Select the “Boss Extrude” command. Notice that we are not given a warning about having two separate bodies; it just works. Extrude any size that looks similar to the following image (dimensions are not important at this time) and click **OK** to finish. (Making two or more extruded/revolved/swept/lofted features would work just the same.)

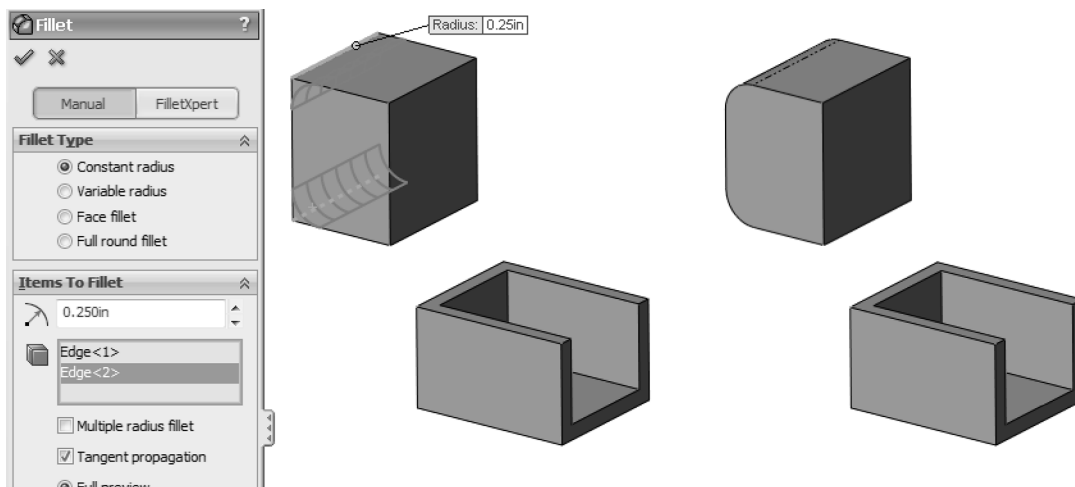


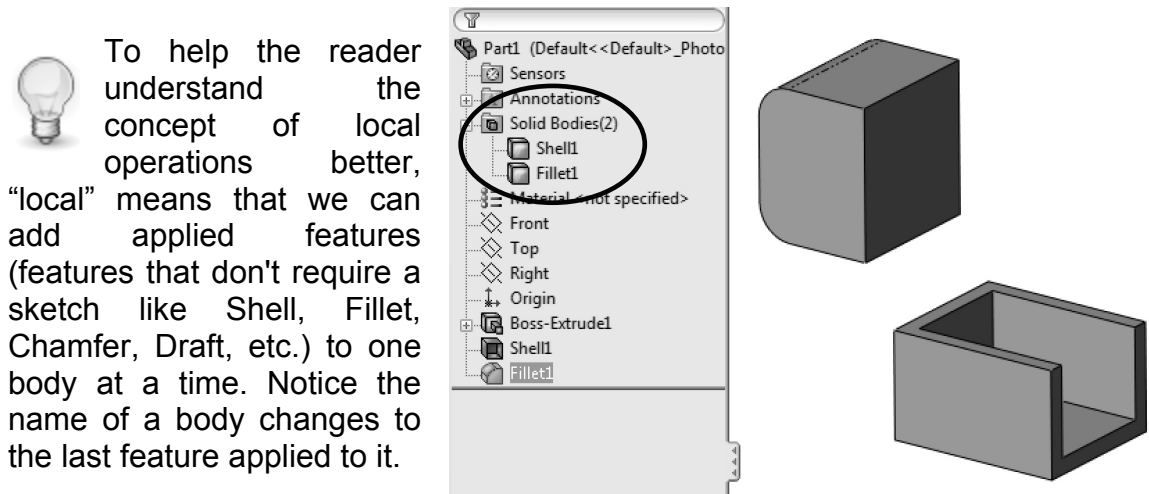
The first thing we notice is a new folder in the FeatureManager called “**Solid Bodies(2).**” This folder is automatically added when SolidWorks detects multiple disjointed bodies in a part and lists the number of bodies found in the part (in this case, 2). If we expand the folder, we can see the two bodies in our part listed under it. The important thing to know and remember is that **multi body parts are not to be confused or used in place of an assembly**; parts and assemblies have significant differences and each serves a different purpose. A multi body part is used mostly as a means to an end.

3. – The next step is to add a new feature. When working in a multi body part we can make ‘local’ operations, for example, a shell feature affecting only one body. Applied features like fillets, shells, chamfers, etc. can only be applied to a single body at a time. Select the “Shell” command from the Features toolbar, and shell the bottom body as indicated.

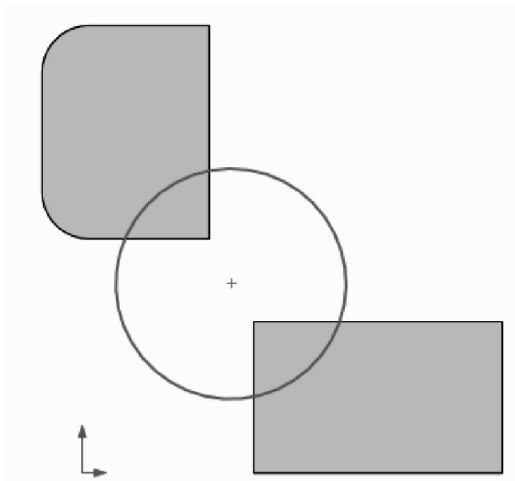


4. – Select the **Fillet** command and round two corners of the top body as shown.

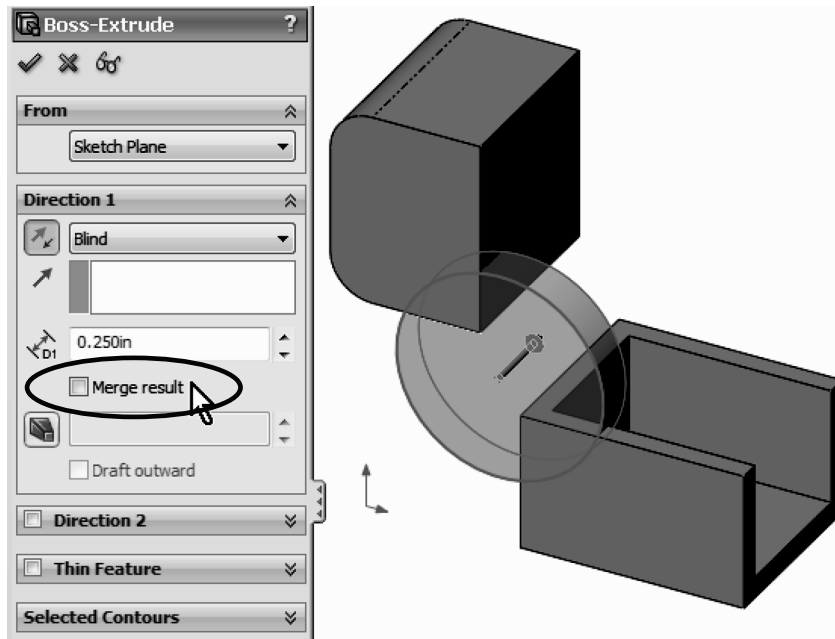




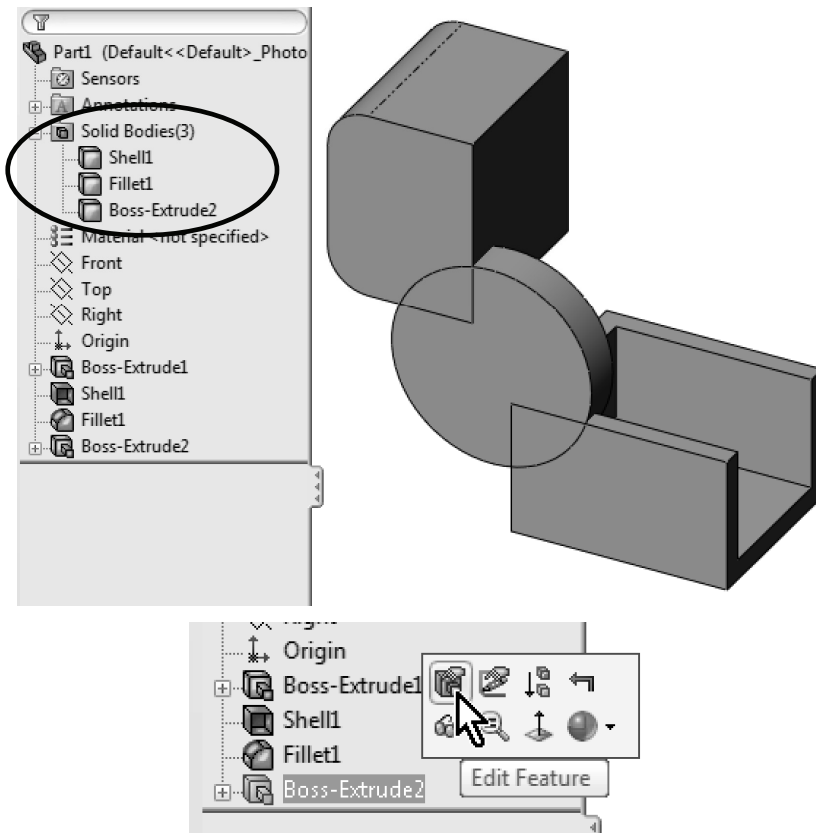
5. – When working with multi bodies, adding more features like a Boss-Extrude automatically selects existing bodies to modify; this is the default behavior. Optionally, we can select which bodies to “merge” (or fuse with) to make a single body, or selectively pick which bodies to merge with. We'll make a new Boss-Extrude and explore the option to merge with existing bodies. Select the front face of a body and make the following sketch in it.

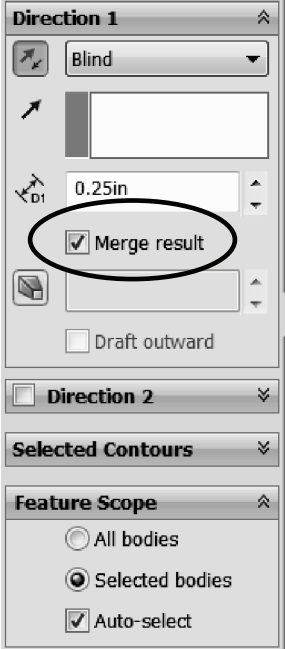
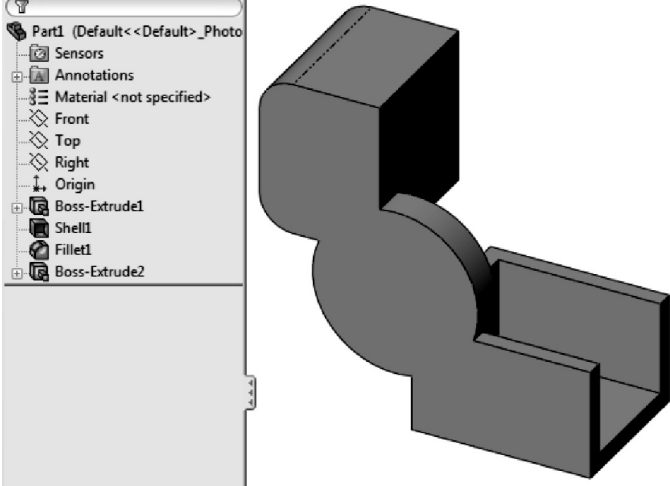

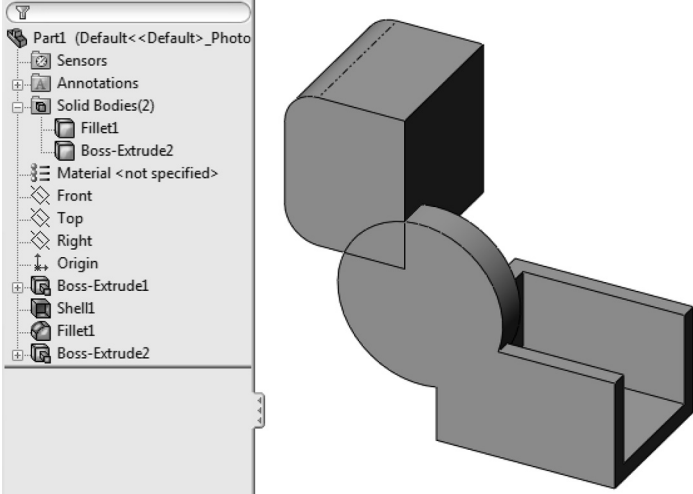


6. – Extrude the sketch *into* the existing bodies. Notice the “**Merge result**” option in the Extrude command. It's always been there (except when there are no solid bodies), and by default is always checked. A new option at the bottom of the Extrude command when we have multiple bodies is “**Feature Scope**.” This is where we can select which bodies to affect. We can choose to affect “All bodies” or “Selected bodies” either automatically or manually selected. By default the Feature Scope option is set to “**Selected bodies**” and “**Auto-select**.” These two options mean that by default the new feature will merge with any body it intersects with. When the “**Merge result**” option is unchecked, “*Feature Scope*” is automatically removed and the new Boss-Extrude will not merge with the existing bodies. Uncheck “Merge Result” and click **OK** to finish.

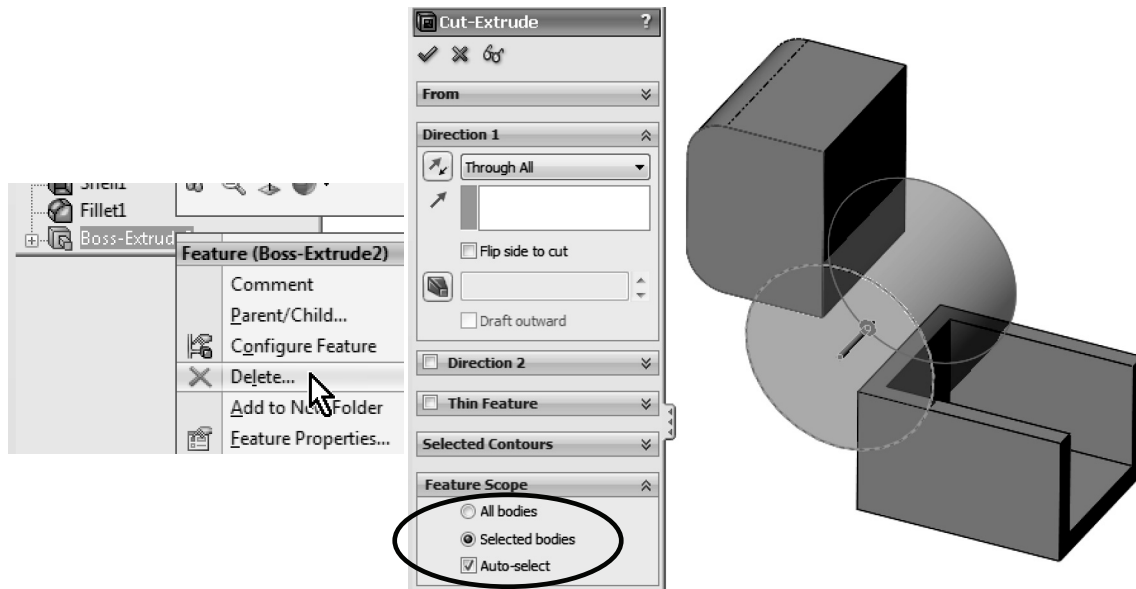


7. – The result of this operation is three bodies in our part. See how the different bodies' edges intersect each other. If we had left the “**Merge result**” option checked, we would not see these edges overlapping as they would have merged into a single body. Edit the definition of the last extrusion to explore the effect of different “**Merge Result**” and “**Feature Scope**” combinations.

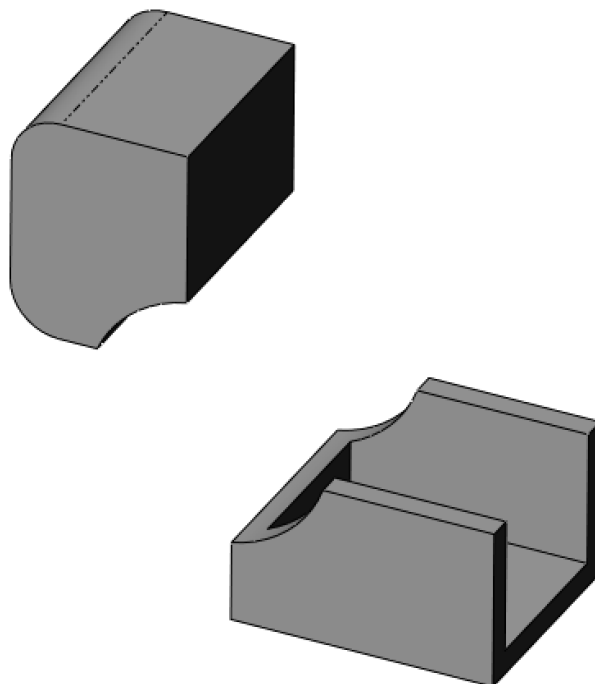


Combination:	Result:
<p>Merge Result: Checked Feature Scope: Auto-Select</p> 	<p>Result: Single body. By touching both existing bodies “Auto-select” merges with them, fusing them into a single solid body. The “Solid Bodies” folder is no longer visible.</p> 
<p>Merge result: Checked Auto-select: Unchecked Add shell body to selection box</p> 	<p>Two bodies. We are telling the Boss-Extrude to merge only to the Shell body (or whichever we pick). Notice the edges of the upper body and the Boss-Extrude overlap, as they are different bodies.</p> 

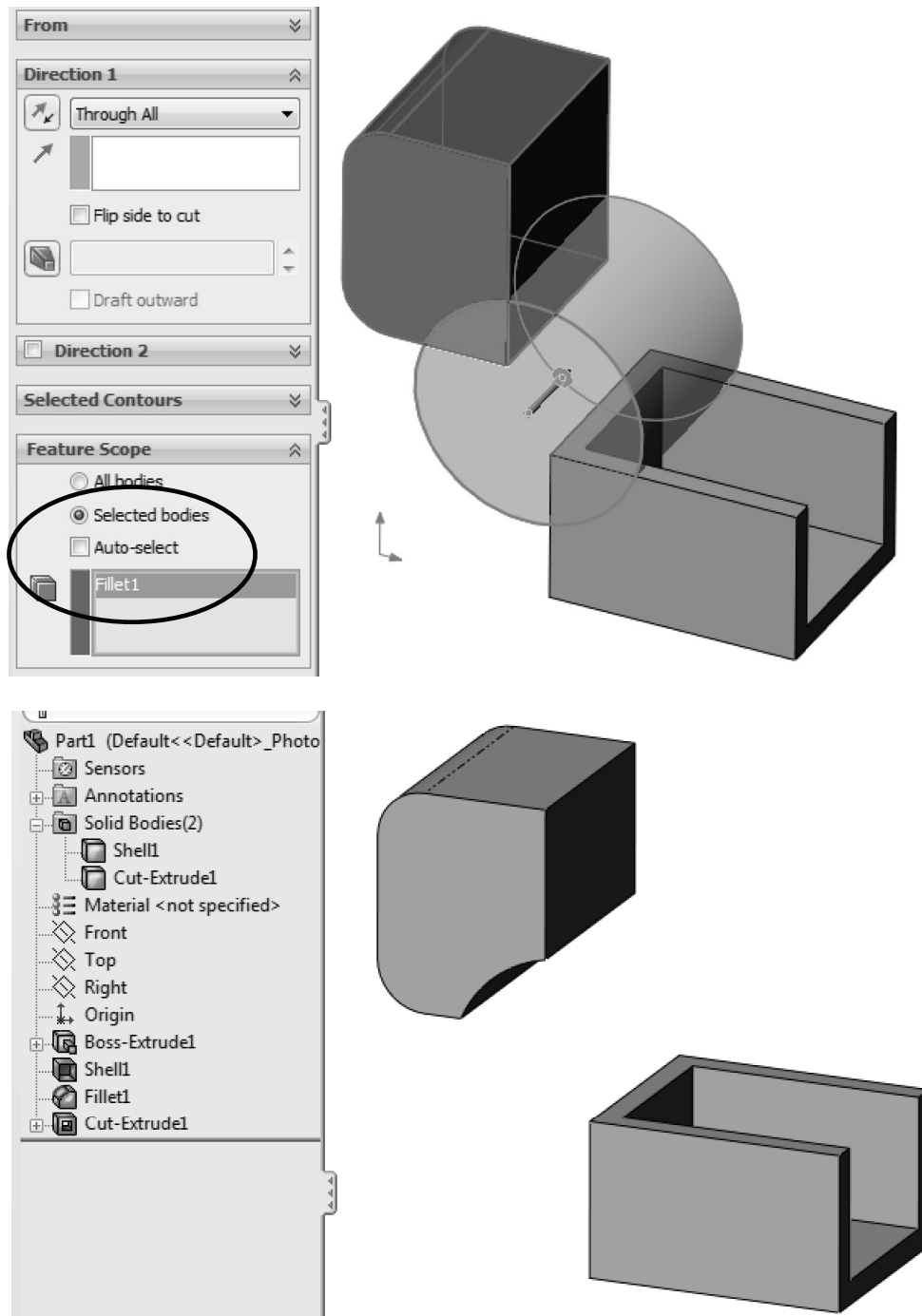
8. –The “**Merge result**” option works the same way with any feature that adds material to the part, including revolved boss, sweep, loft, etc. Now we’ll see how it works when we remove material. Delete the Boss-Extrude feature and keep the sketch. Select the sketch in the FeatureManager and make a Cut-Extrude “Through All.” In this case the “Merge Result” is gone and we only have the “**Feature Scope**” option at the bottom with the same selection options: “All bodies” or “Selected Bodies” with/without “Auto-select.” Leave the “Auto-select” option on and click **OK** to finish.



9. – What we end up with is the same two bodies we had before, but now they have a cut through them.

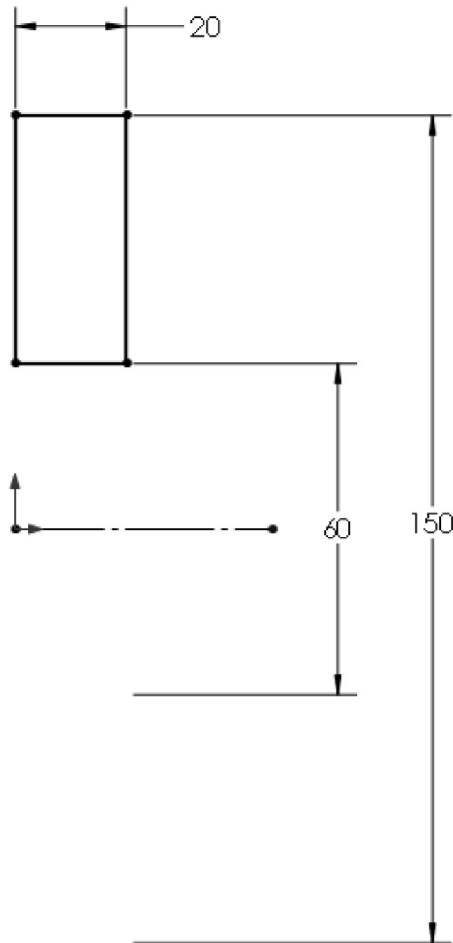


10. – Edit the Cut-Extrude definition, turn off the “Auto-select” option in the “Feature Scope,” and select only the top body. Click **OK** to finish. Now we are only modifying the top solid, even if the Cut-Extrude overlaps the lower body.

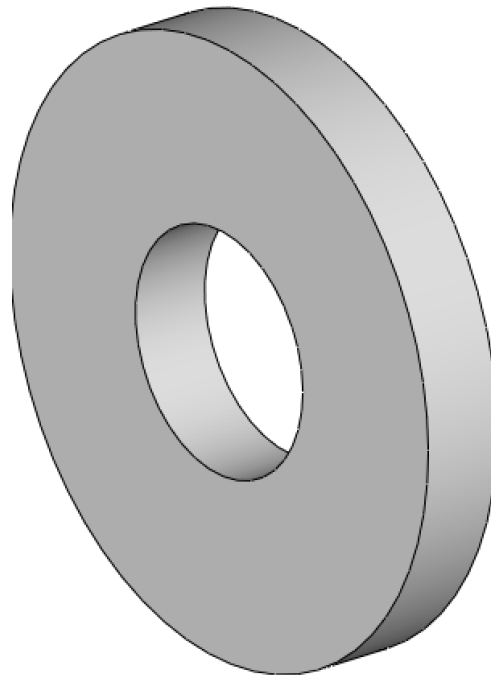


And just as with the features that add material, this technique works the same way with all the features that remove material including revolved cut, sweep cut, loft cut, etc.

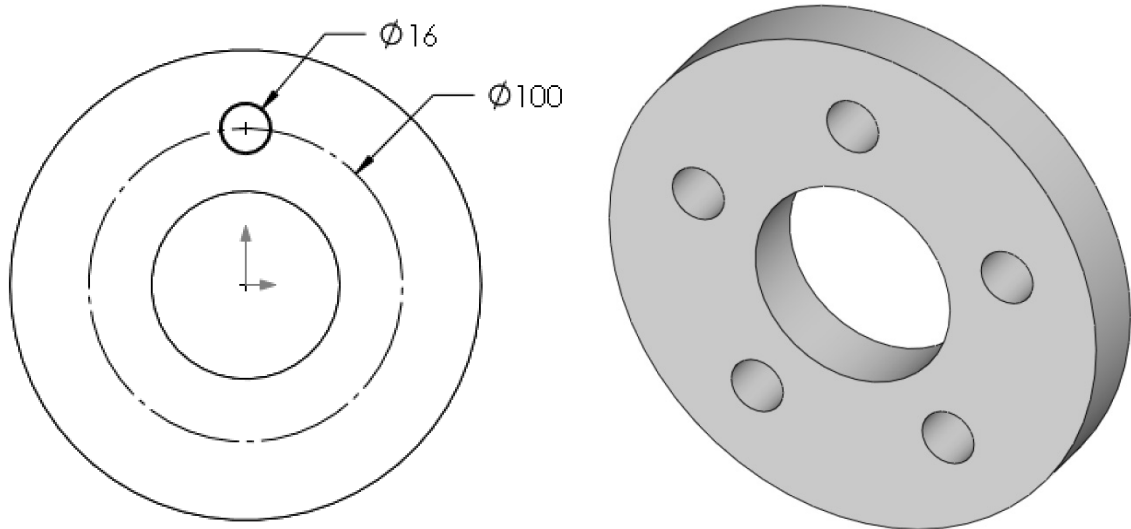
11. – Modeling with multi bodies is a powerful technique to model parts that would otherwise be difficult to complete. One frequently used technique is called “bridging”; this means to connect two or more bodies by adding material between them to merge them into a single solid body. Reasons to use this technique may include a model where we know what opposite sides/ends of a part look like, but we may not know what the middle (the “bridge”) should be like. For our example we’ll assume that we need to design a car’s wheel. We know what the actual tire and hub dimensions should be, but we don’t know yet what the spokes will look like; we just know it has to look great 😊.



We’ll assume the dimensions for the wheel are as shown in the following sketches. The first part of the wheel to design will be the hub or mounting pad. Draw the following sketch in the Right plane and make a 360° Boss-Revolve. Our wheel’s dimensions will be in millimeters, so be sure to change your model’s dimensions accordingly (**Tools, Options, Document Options, Units**, or in the status bar’s **Unit System**). Pay attention to the diameter dimensions (doubled about the horizontal centerline).

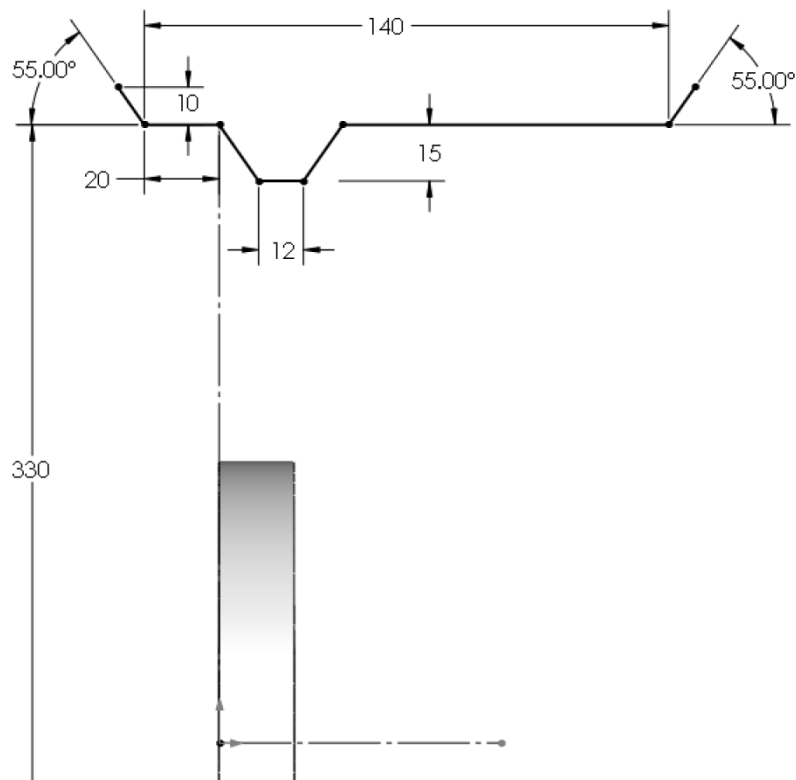


12. – Make a pattern of 5 holes to mount the wheel to the car.

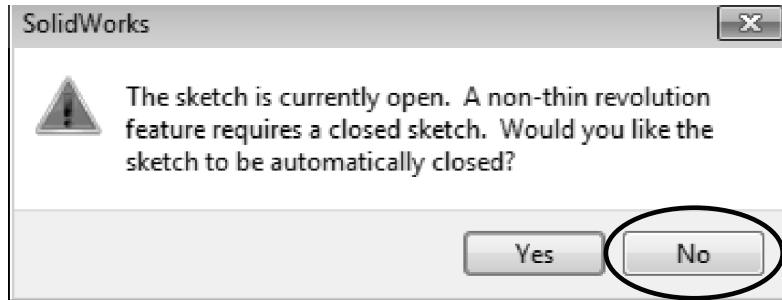


13. – Now we'll make the wheel's rim that matches the tire. Draw the following sketch also in the Right plane. (The 330 mm diameter dimension is doubled about the horizontal centerline as diameter.)

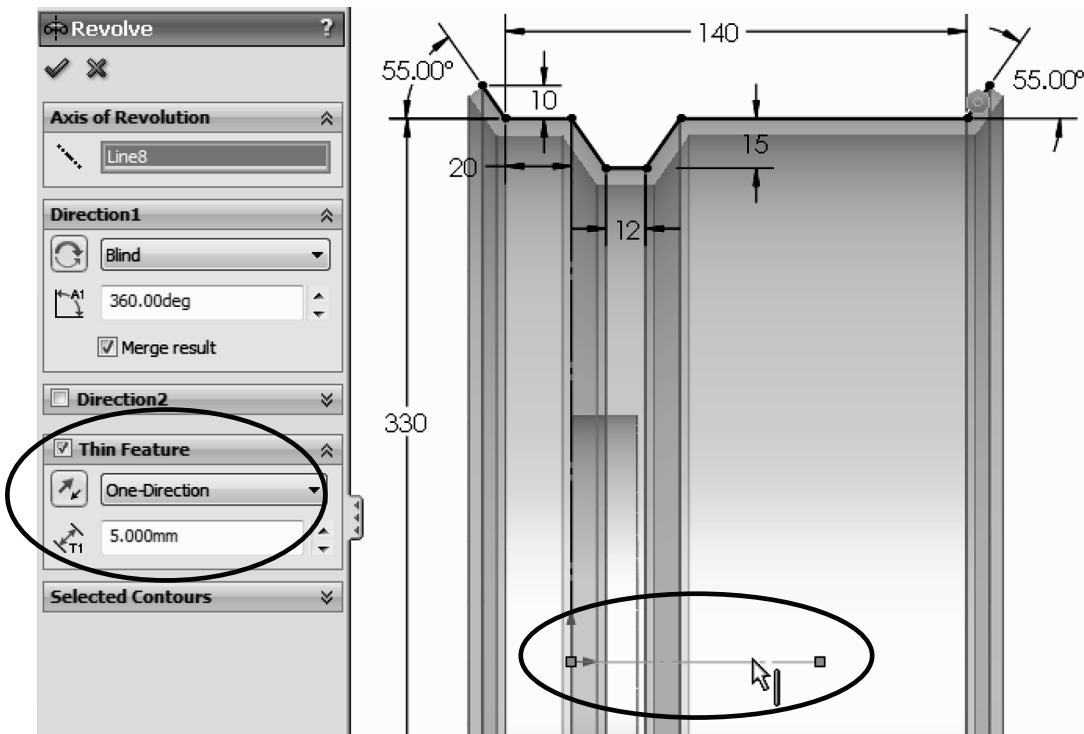
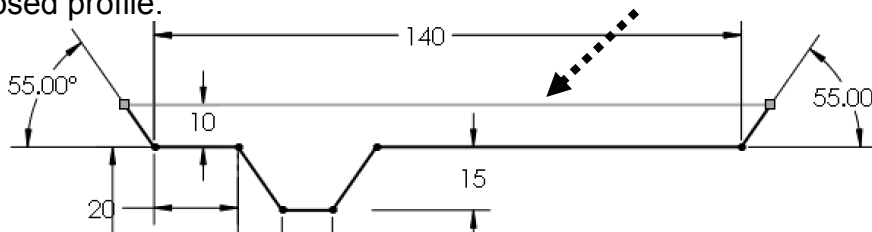
TIP: Make the two short lines on the sides equal, and the two lines connected to the 12mm horizontal line also equal.



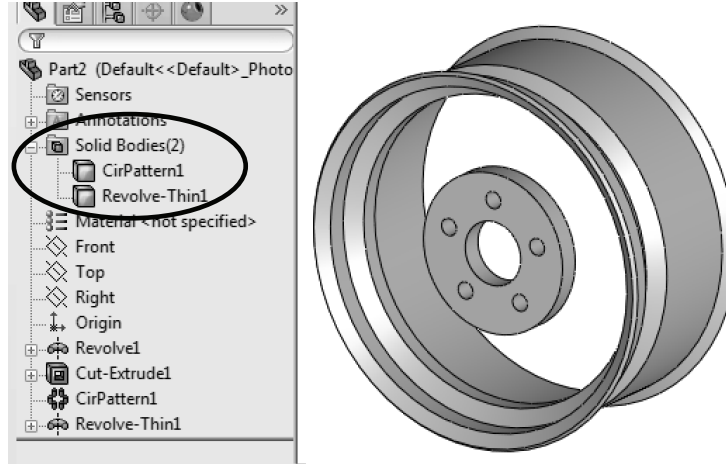
14. – After finishing the sketch, make a Revolved-Boss. Since this is an open sketch we will be warned about closing it if we don't want a thin feature. Select NO when asked. Make a thin revolved feature using the horizontal centerline for axis of revolution, and 5mm thick going inside.



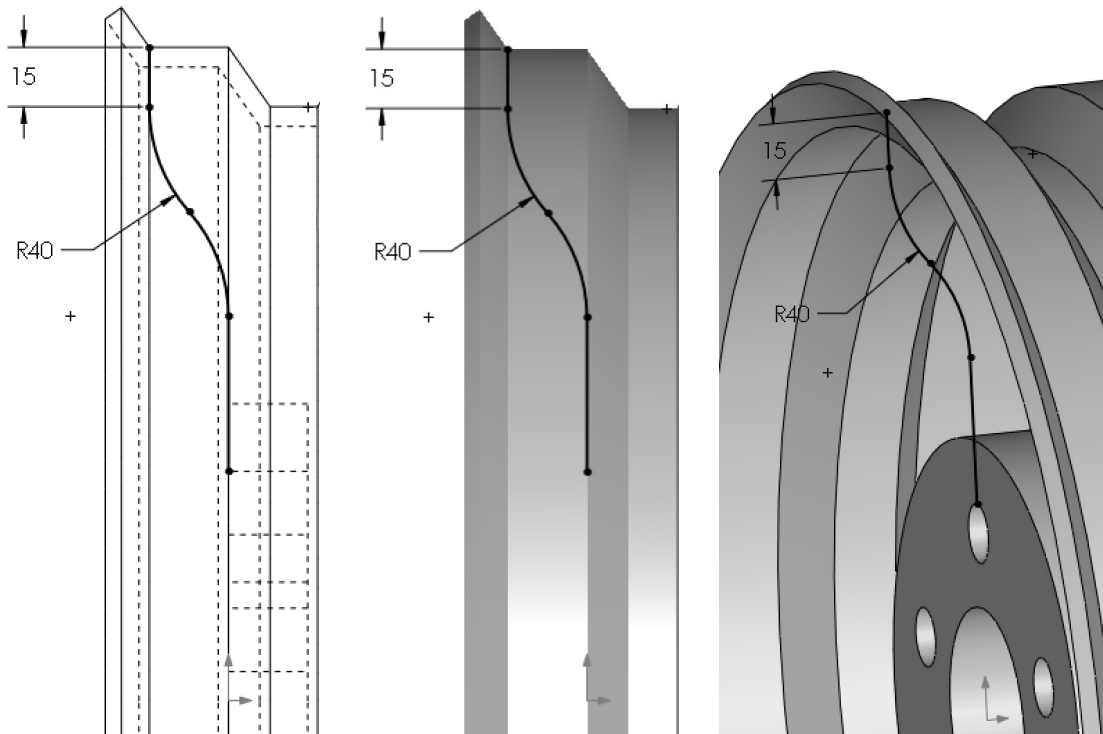
If we answer YES to close the sketch, a new line will be added going from the first line's open endpoint, to the last line's open endpoint making a closed profile.



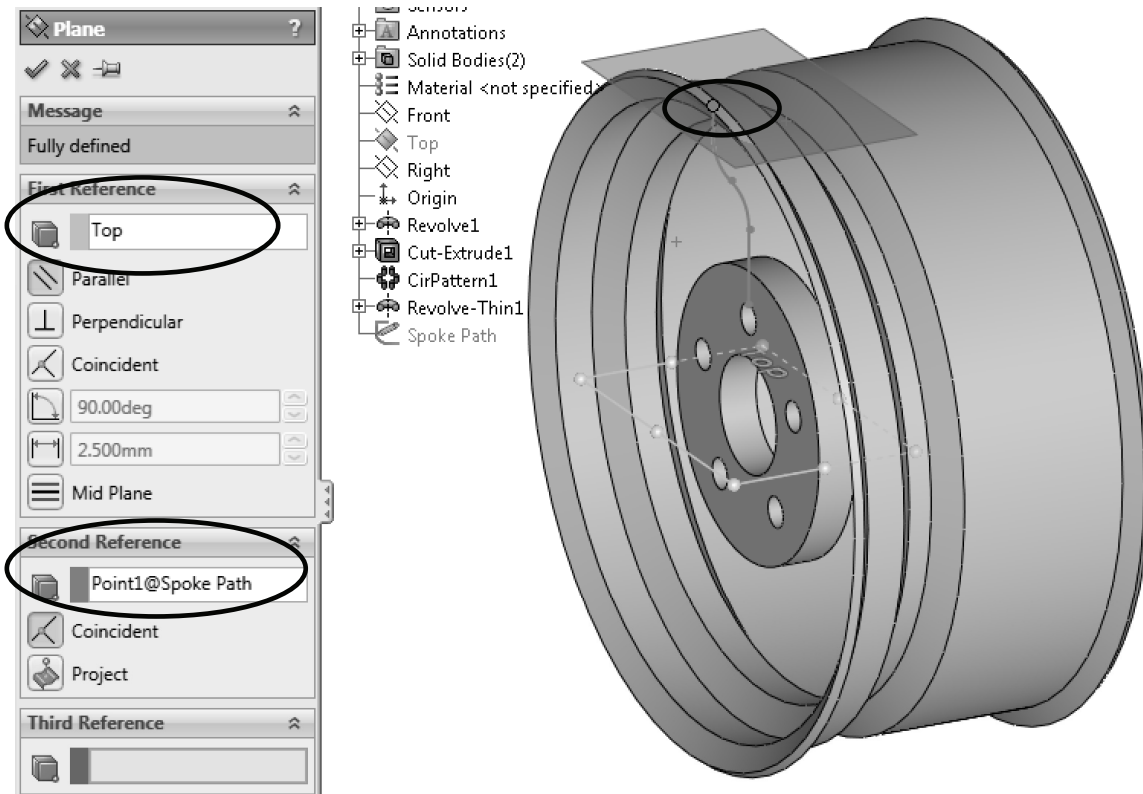
After finishing the revolved feature, we can see that now we have two bodies.



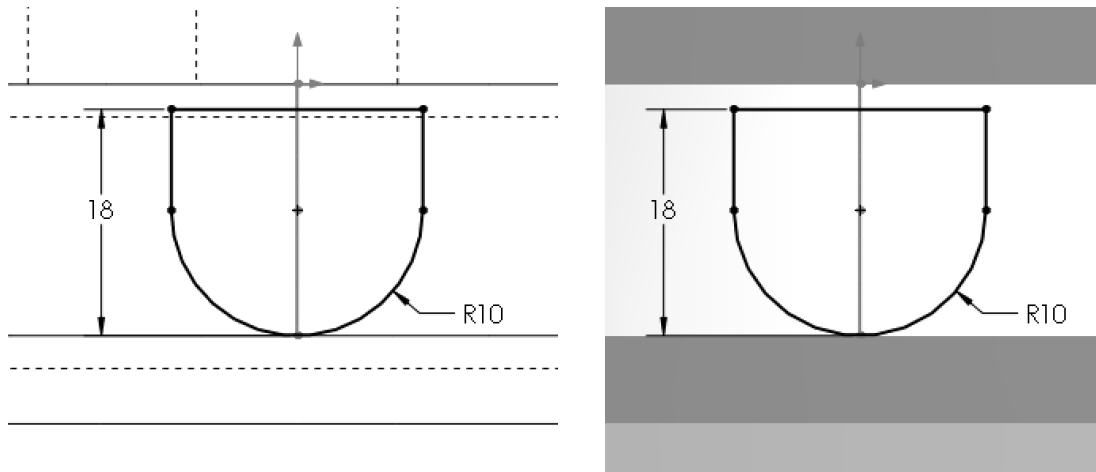
15. – Now we need to make the spoke in the wheel; this is the part where we can get creative. For this exercise we'll assume our limitations are defined by the hub and the rim where the tire mounts. We'll design the first spoke and will not merge it to any other body; we'll have to make a couple of local operations on the spoke and after it's done, make a body pattern with it. Switch to a Right view and add a sketch in the Right plane. Change to "Hidden Lines Visible" mode for visibility. A shaded model is shown for sketch clarity. Make both arcs tangent to the vertical lines, tangent to each other, and with equal radii. The vertical lines are coincident to the first hole in the hub and the rim's outside edge. This will be the path for a sweep. Exit the sketch and rename it "Spoke path."

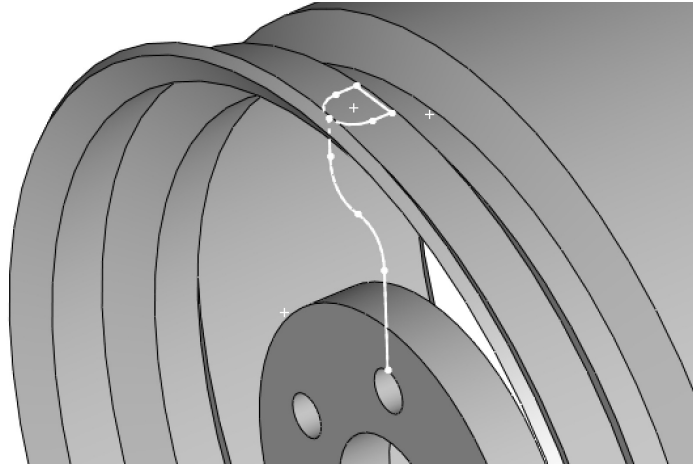


16. – Make a new plane parallel to the Top plane and coincident to the top endpoint of the previous sketch. This plane will be used to draw a new sketch that will be the sweep's profile.

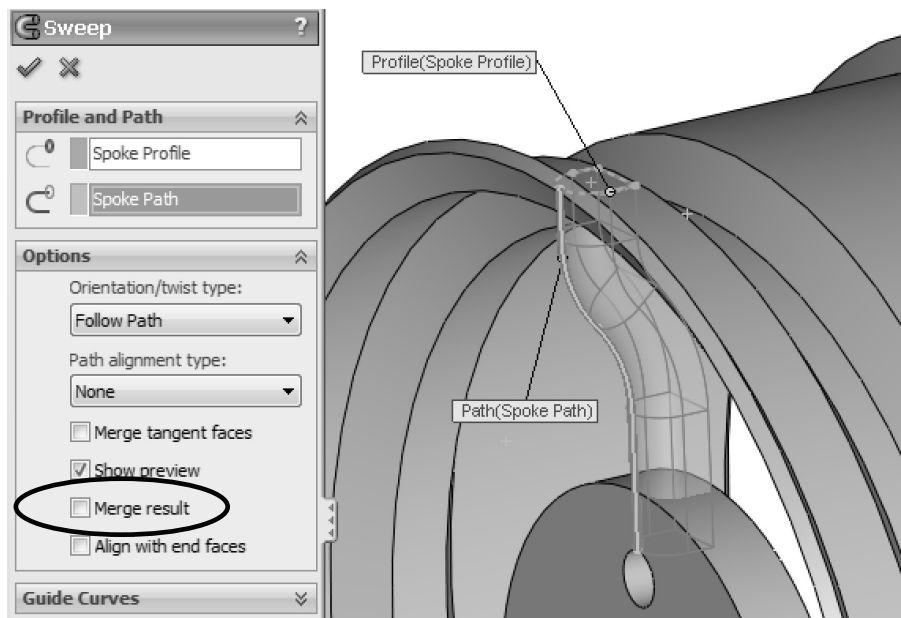


17. – Switch to a Top view, add a new sketch to the new plane and draw the following sketch. This sketch will be the Profile for the sweep. The arc is tangent to the edge where the path starts. Exit the sketch when done and rename it "Spoke Profile."

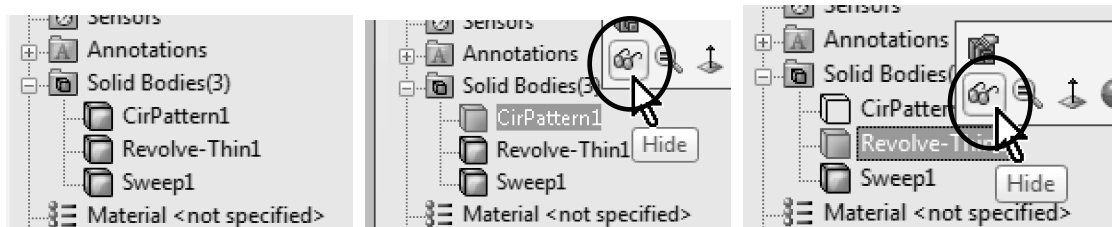




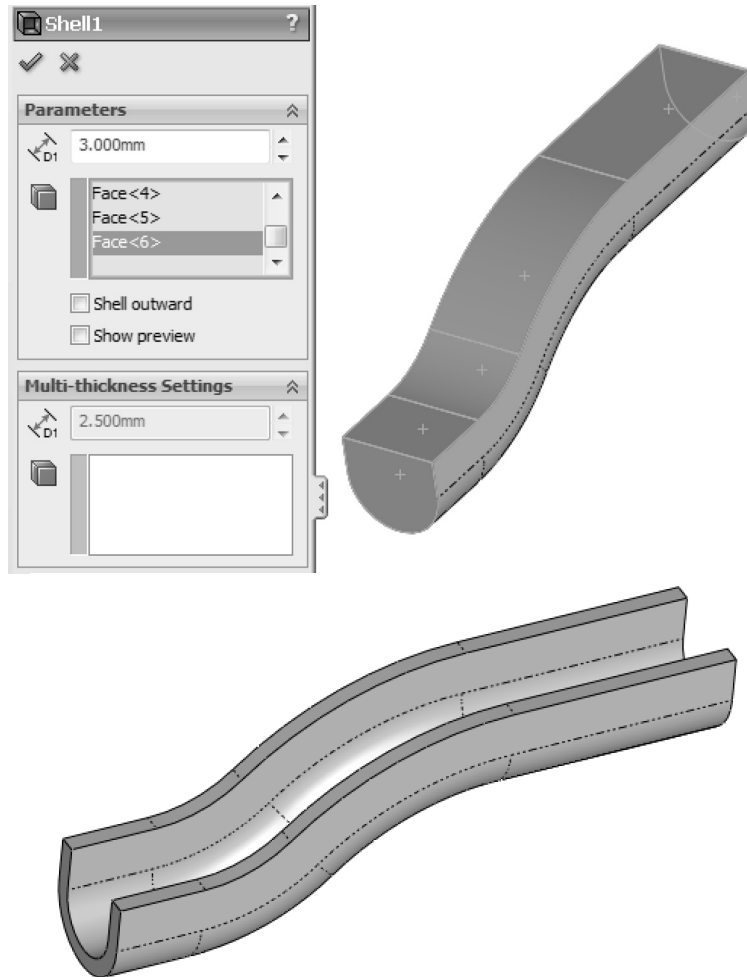
18. – Select the **Sweep** command. Select the "Spoke Profile" sketch and "Spoke Path" to make the sweep. In the Options, uncheck **"Merge Result"** as we still need to make a couple more operations before merging all bodies. Click **OK** to finish.



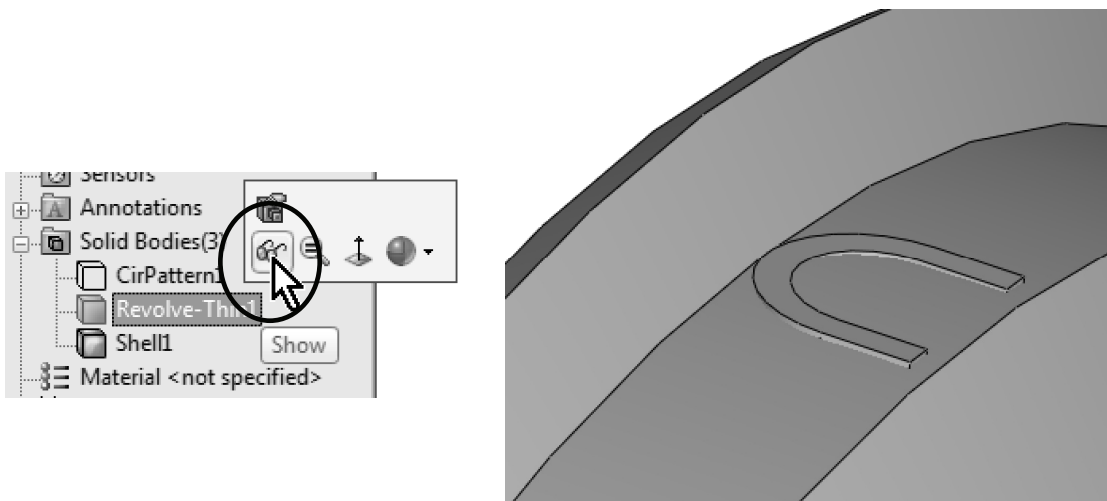
19. – Now we have three bodies. To continue, we'll hide the first two and only leave the Sweep body visible. Select each of the other two bodies and hide them.



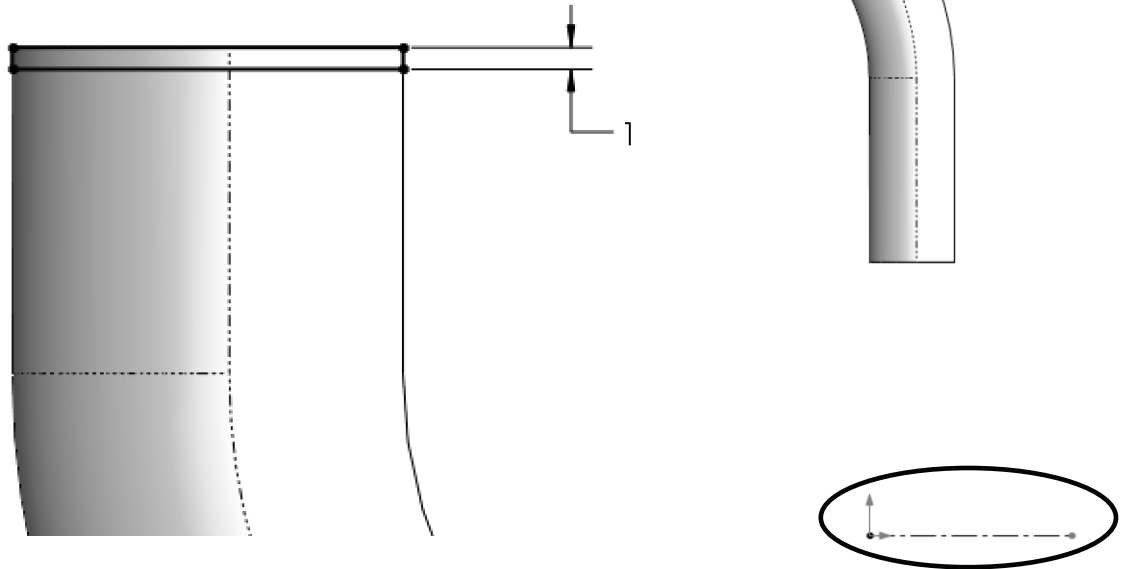
20. – After we are left with only the Sweep body, select the **Shell** command and remove the ends and back of the body. Make the shell 3mm thick.



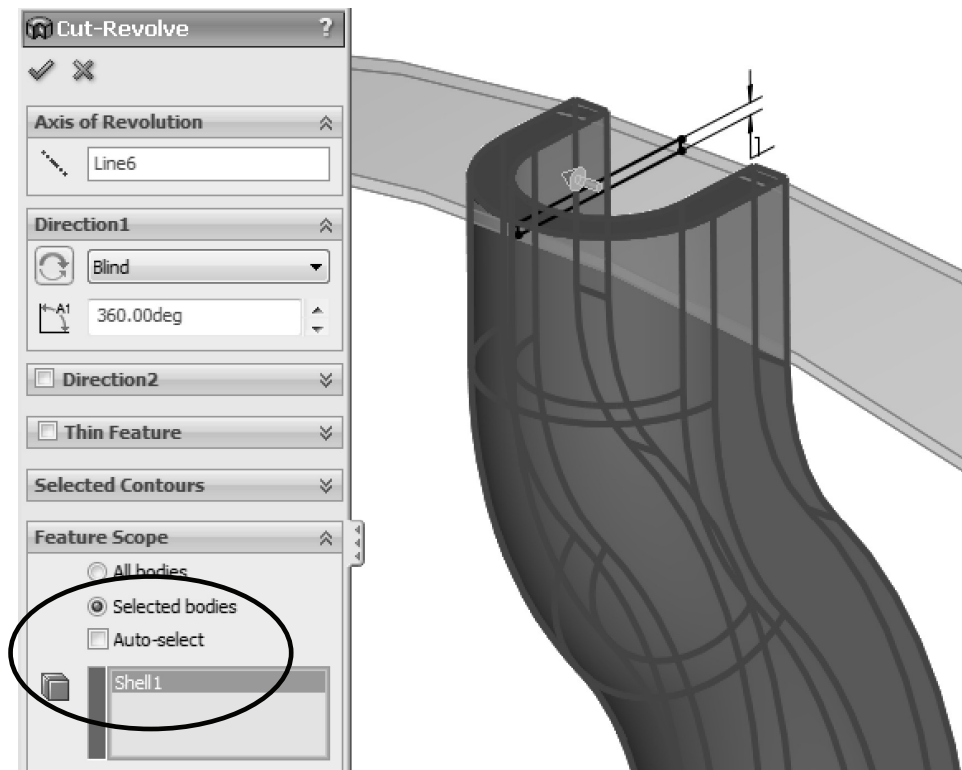
21. – We need to make a cylindrical cut to the top; otherwise we'll have a part of the Sweep coming through the top. Show the Revolve-Thin body to see it.



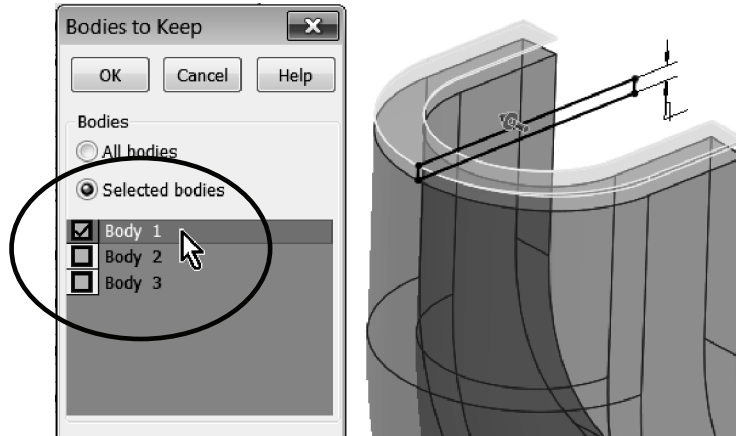
22. – Hide the Revolve-Thin feature again and switch to a Right view. Add a new sketch in the Right plane as shown. We are making the cut just under the top to make sure the Sweep body (now named Shell1) will not go through the top. Add a horizontal centerline at the origin to make a Revolved Cut about it.



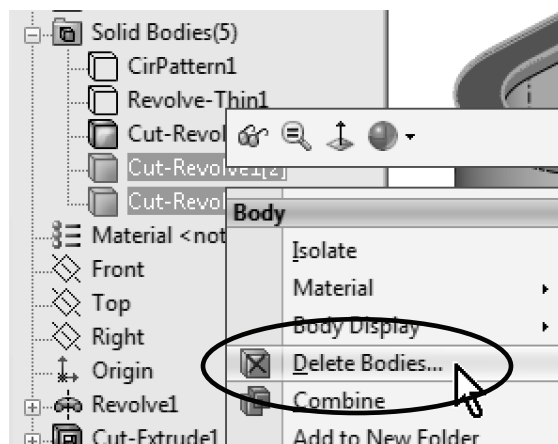
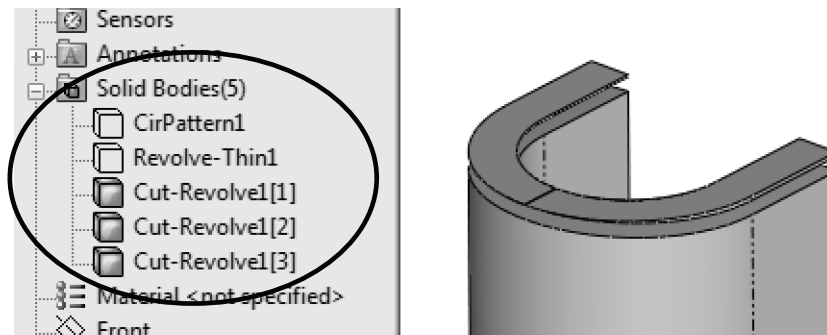
23. – Select the **Revolved Cut** command. What we want to do is to cut only the spoke body, and nothing else. Under **Feature Scope** uncheck “Auto-select” and select the spoke body only.



24. – After clicking **OK** we get a new dialog box asking us which bodies we want to keep, since the cut we are making divides the spoke in three pieces. Use the “**Selected bodies**” option, and select the spoke’s largest body which is the one we want to keep. Click **OK** to finish. In this case we still have our three original bodies and have discarded the two smaller ones.



25. – A different approach, assuming had we opted to keep all three bodies after the Revolved Cut, would be that we have five bodies in our part, and to eliminate the extra bodies we don't need we would have to delete them. To do this, select the two bodies, right mouse click and select “**Delete Bodies.**” Make sure the correct bodies are selected and click **OK** to finish.

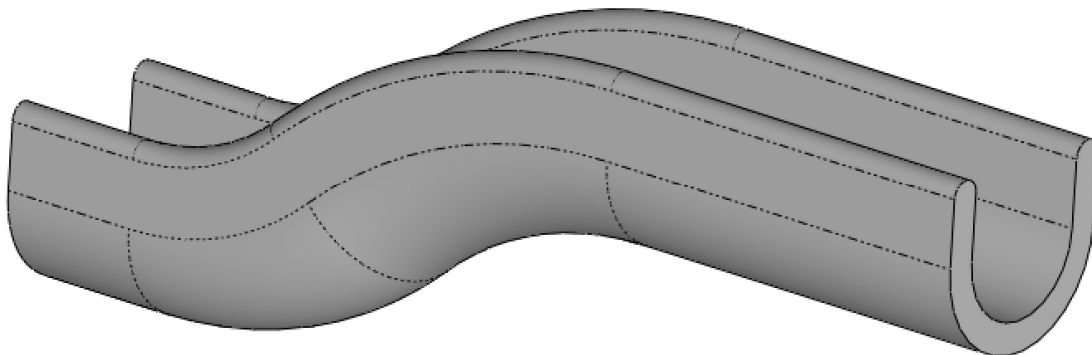
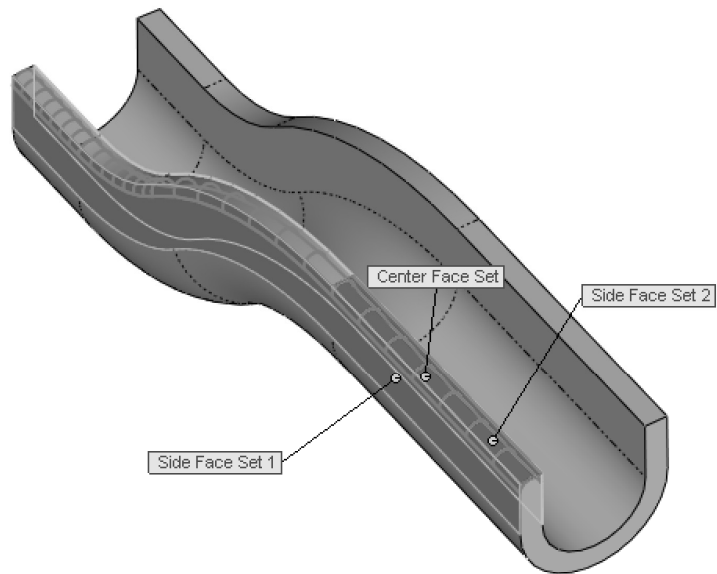
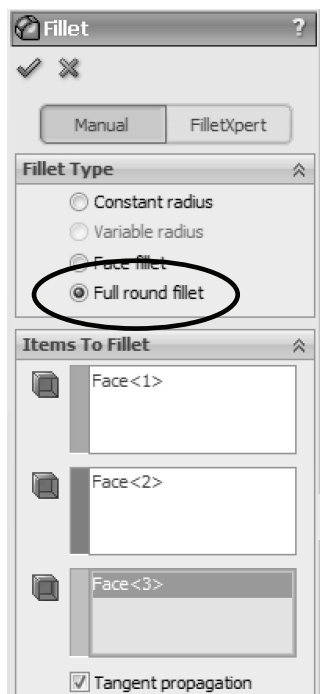


A new feature “**Body-Delete**” would be added to the FeatureManager.

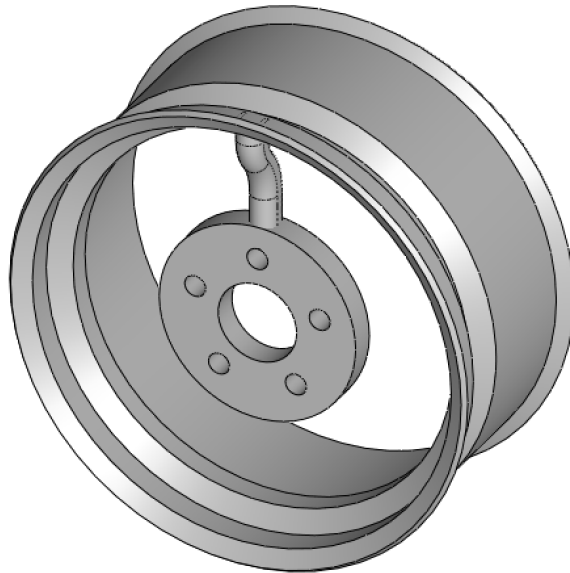
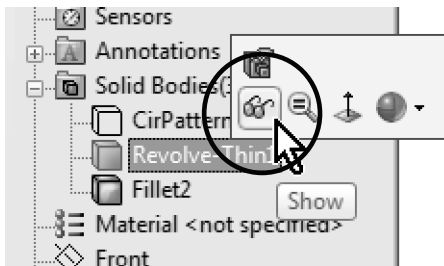


For our example we'll use the approach to select the bodies to keep at the time of making the Cut-Revolve instead of deleting the remaining bodies with an additional operation.

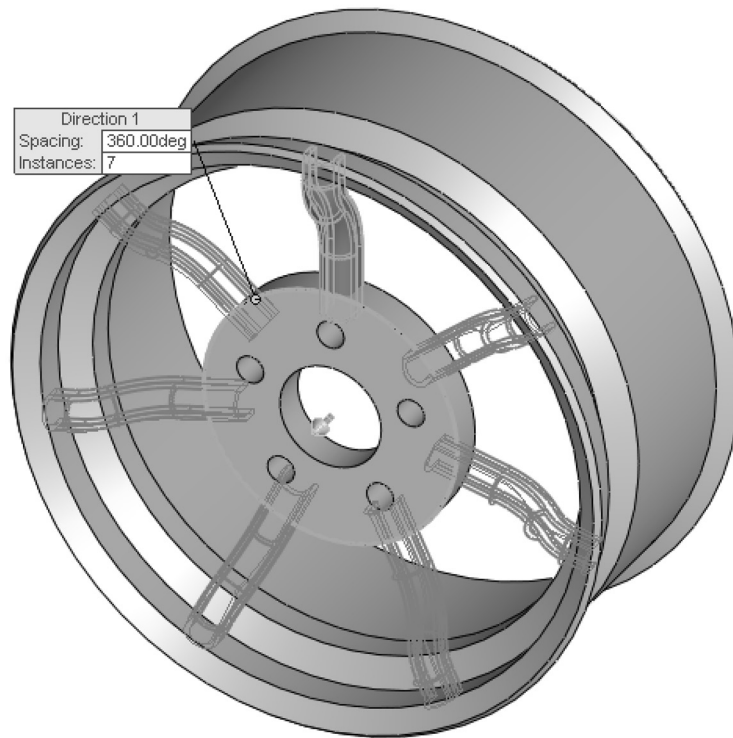
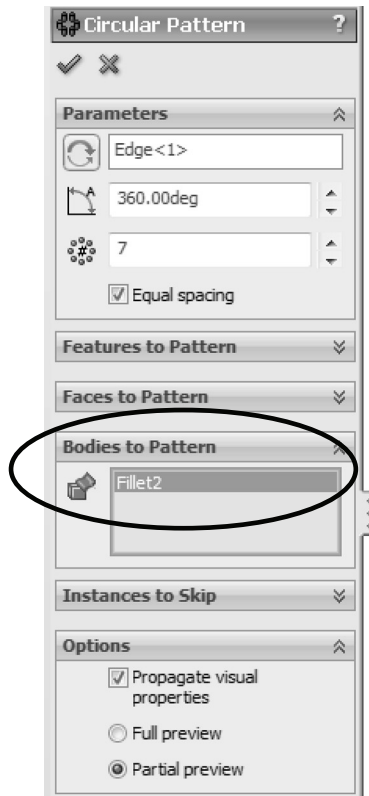
26. – Now add a Full Round Fillet to each side of the Spoke. Select the **Fillet** command with the **Full Round** option. Select both side and center faces to round both sides of the spoke. Two fillets will be added.



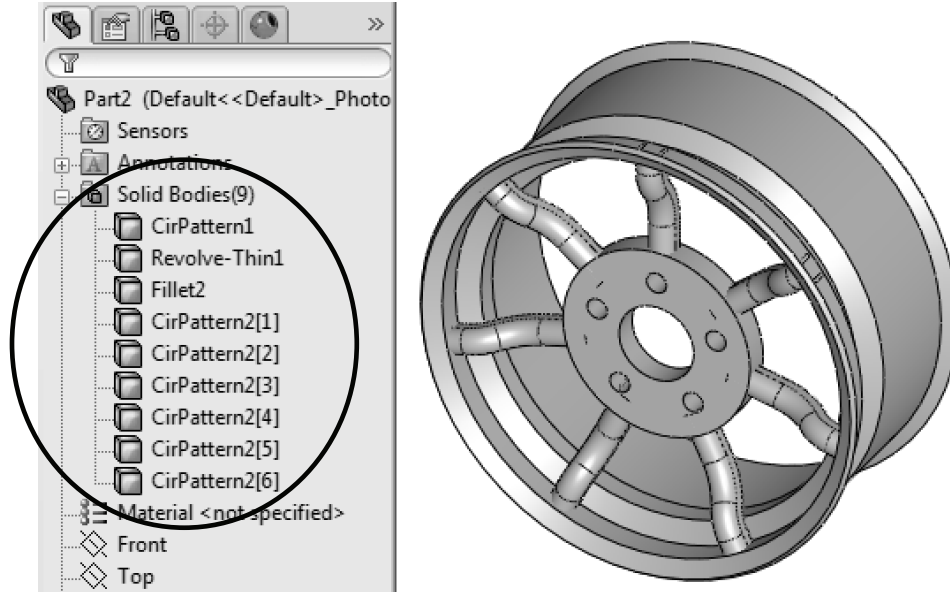
27. – We are now ready to pattern the spoke body. To show the hidden bodies, select them in the “**Solid Bodies**” folder and click in the **Show** command.



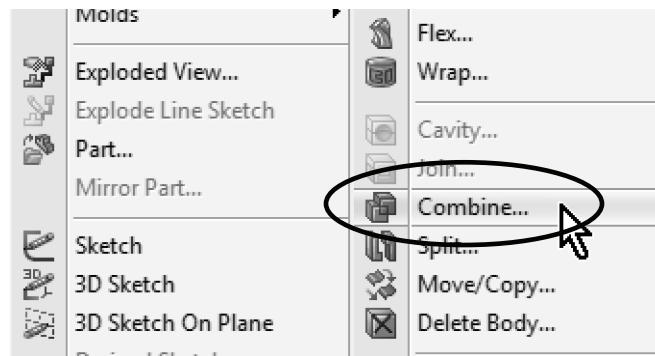
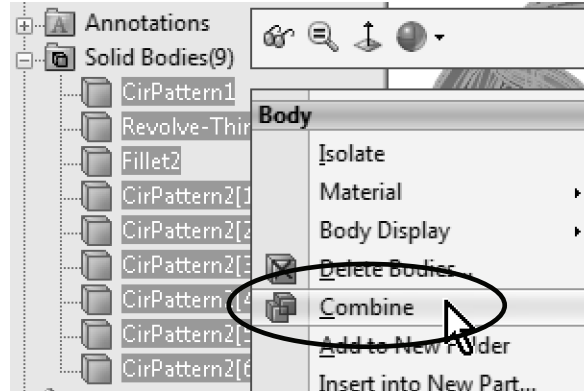
28. – Select the **Circular Pattern** command, but instead of making a pattern of features, we'll make a pattern of bodies. Select a circular edge to define the pattern's axis and expand the “**Bodies to Pattern**” selection box. In the graphics area, select the Spoke and make 7 copies equally spaced in 360 degrees. Click **OK** to finish.



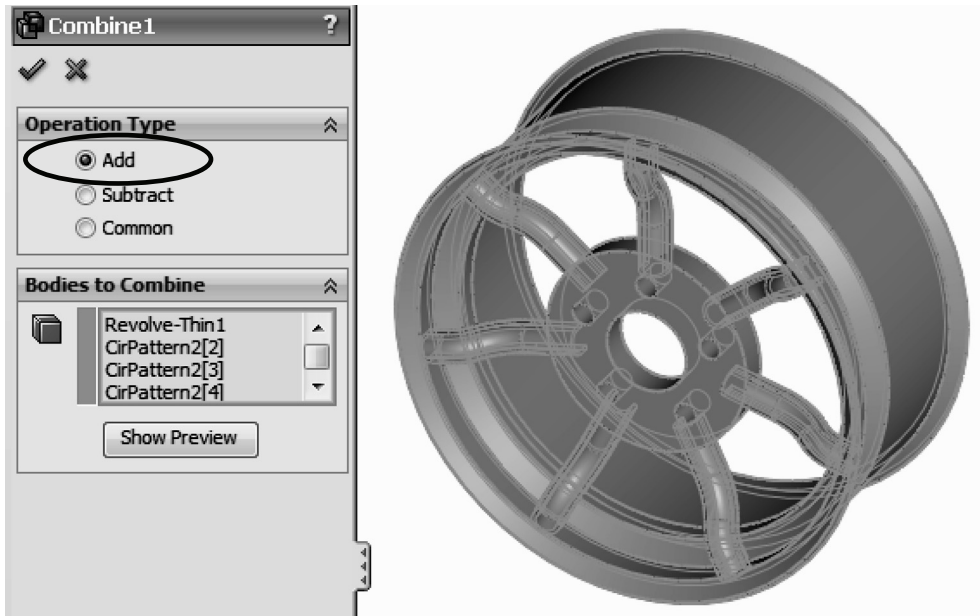
29. – Now we have nine solid bodies in our part.



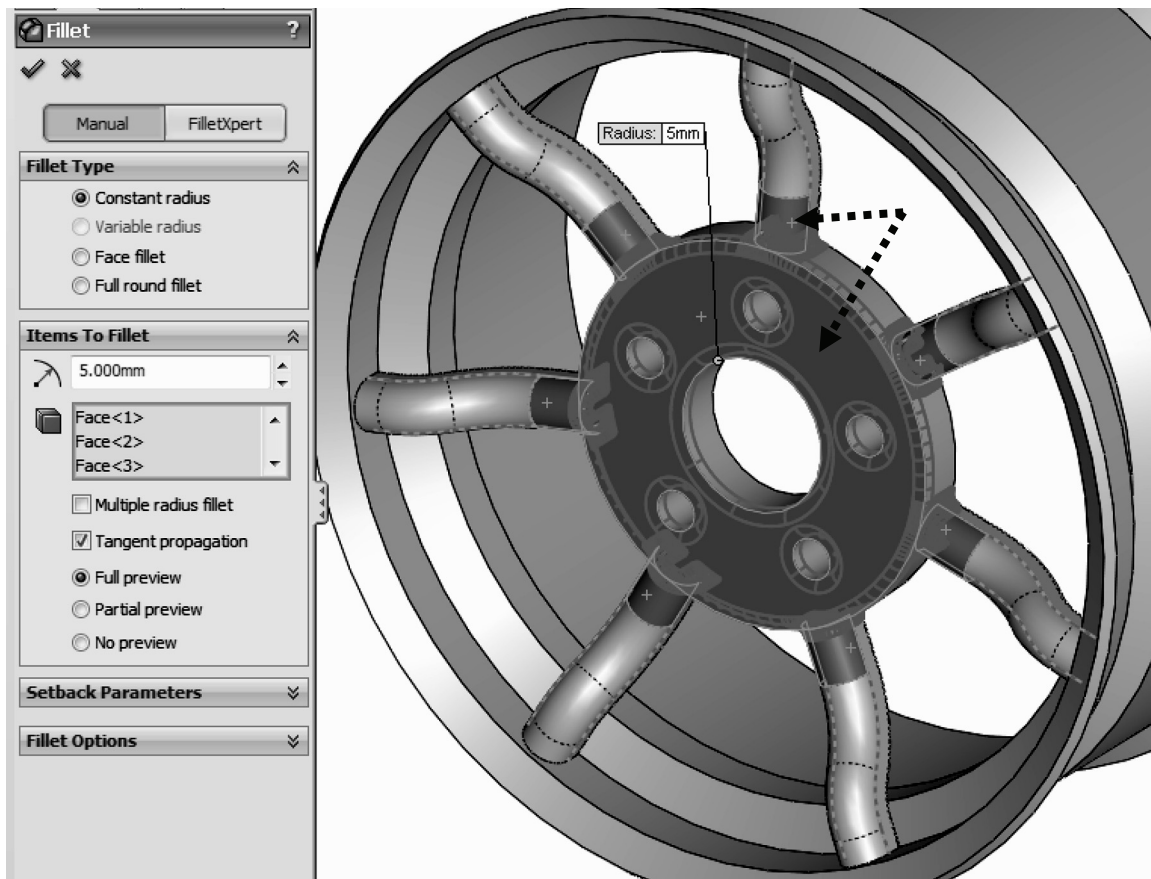
30. – At this time we are ready to combine all solids into a single body. Select all bodies in the FeatureManager, right mouse click on them, and select “**Combine Bodies**” from the pop-up menu, or from the menu “**Insert, Features, Combine,**” and select all the bodies in the graphics area.



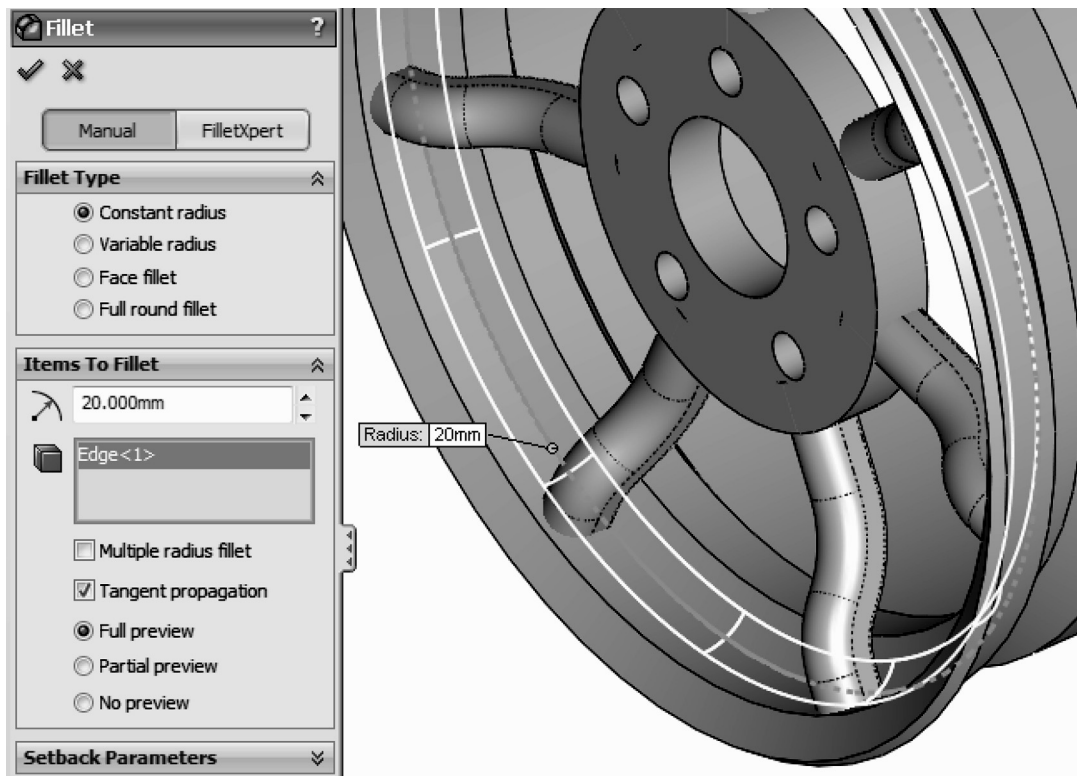
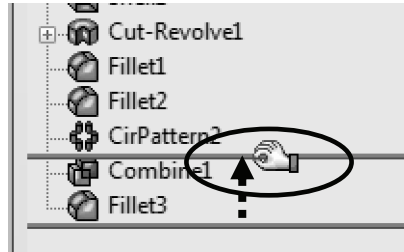
31. – Select the “Add” option and click **OK** to finish. All bodies are now combined into a single solid.



32. – Add a 5mm fillet to the faces indicated at the center and each spoke.



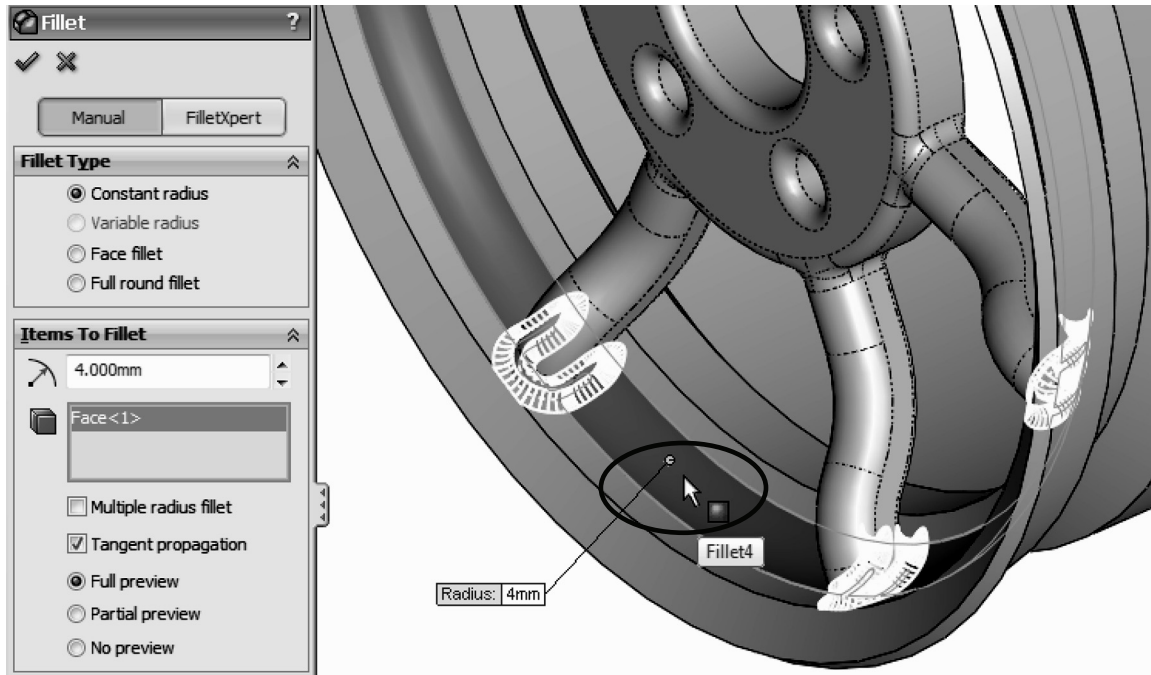
33. – To add a similar fillet to the other side of the Spokes, first we need to add a fillet to the Rim's body. If we try to add the fillet after the bodies are combined, the fillet will fail. Move the Rollback bar from the bottom of the FeatureManager before the “Combine1” feature. What we need to do is to add a 20mm fillet to the indicated edge of the rim's body *before* the solid bodies are combined.



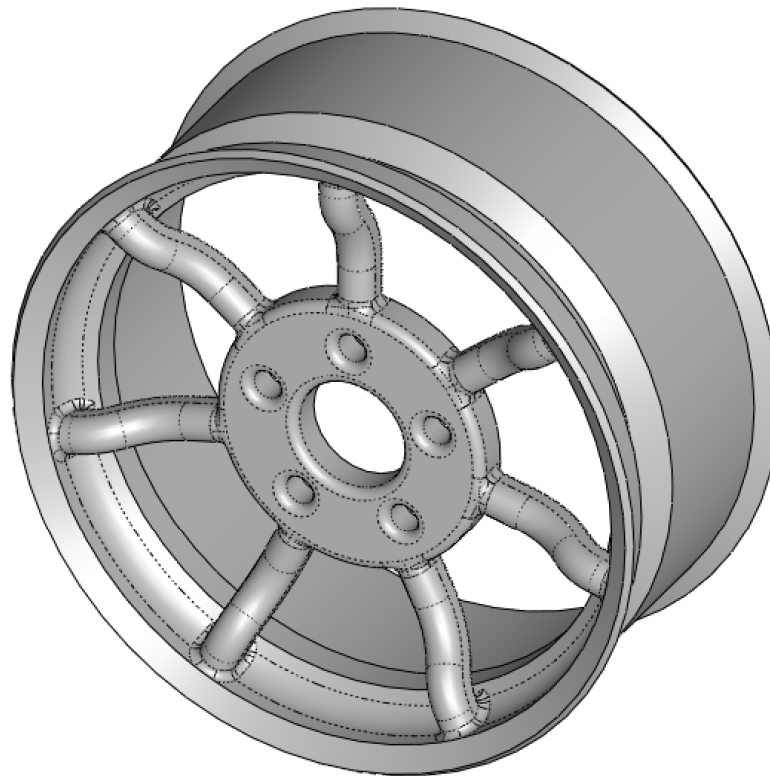
34. – Drag the Rollback bar down to the bottom when finished to have the "Combine1" and "Fillet3" features resolved, and have a single body again.



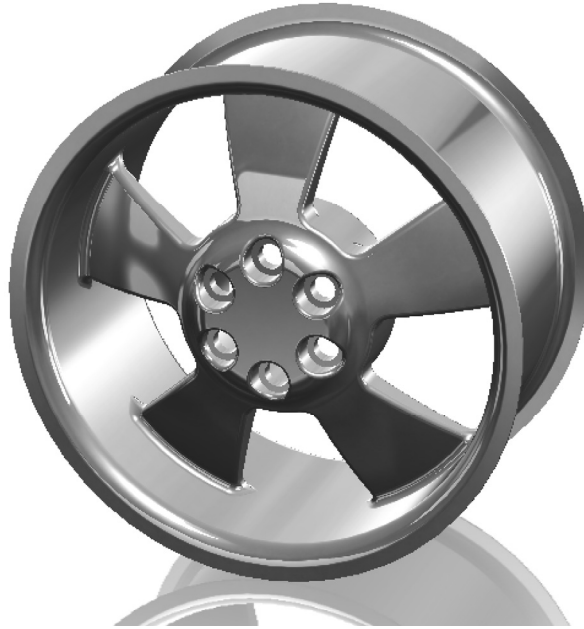
35. – Add a 4 mm fillet to the face of the previous fillet to propagate it to all Spokes at the same time.



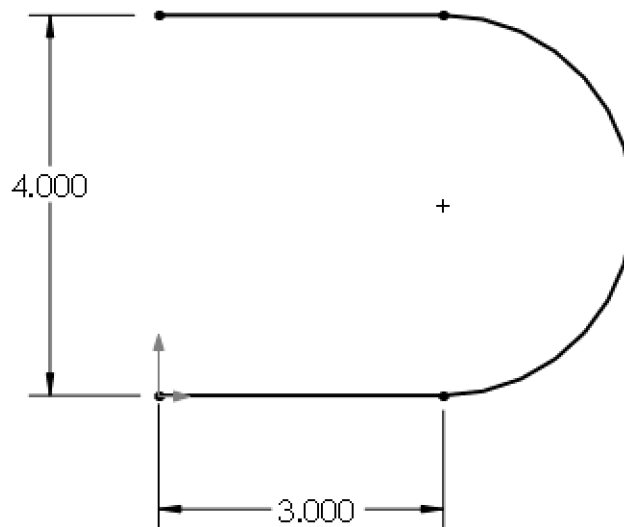
36. – Your finished wheel should now look like this:



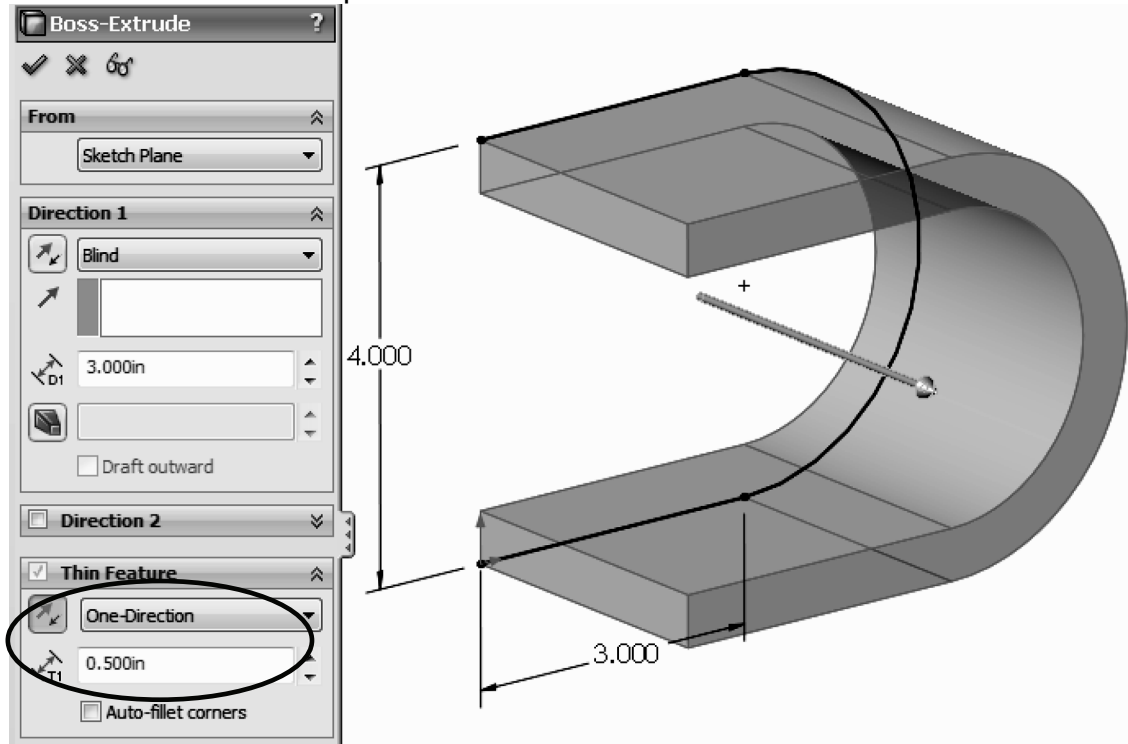
I know this may not be the best looking wheel, but it's a good example to show how to work with multi body parts and perform local operations. We'd love to see your designs; please email us a picture to alejandro@mechanicad.com



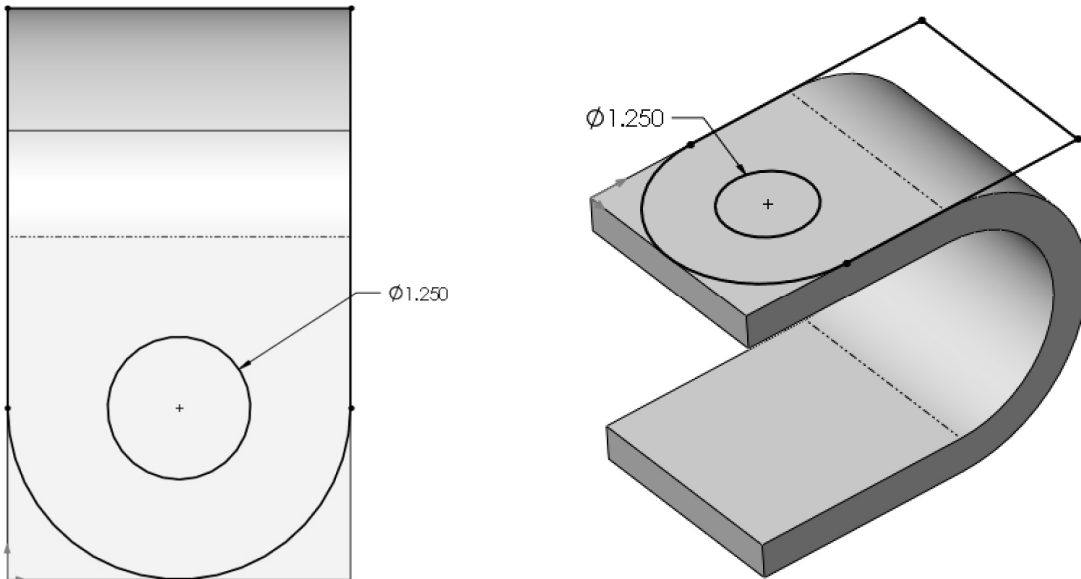
37. – The next area to cover in the multi bodies topic will be to learn more about combining bodies. What we did in the wheel exercise was to add bodies, in essence merging or fusing them. The other two operations we can do when combining bodies is to subtract one or more bodies from another, or get the common volume between them. To obtain the common volume between bodies, first we need two bodies that intersect. Open a new part and draw the following sketch in the Right plane.



38. – Make a Boss-Extrude as a Thin Feature. The extrusion's depth will be 3" and the thickness of the part 0.5" inside.

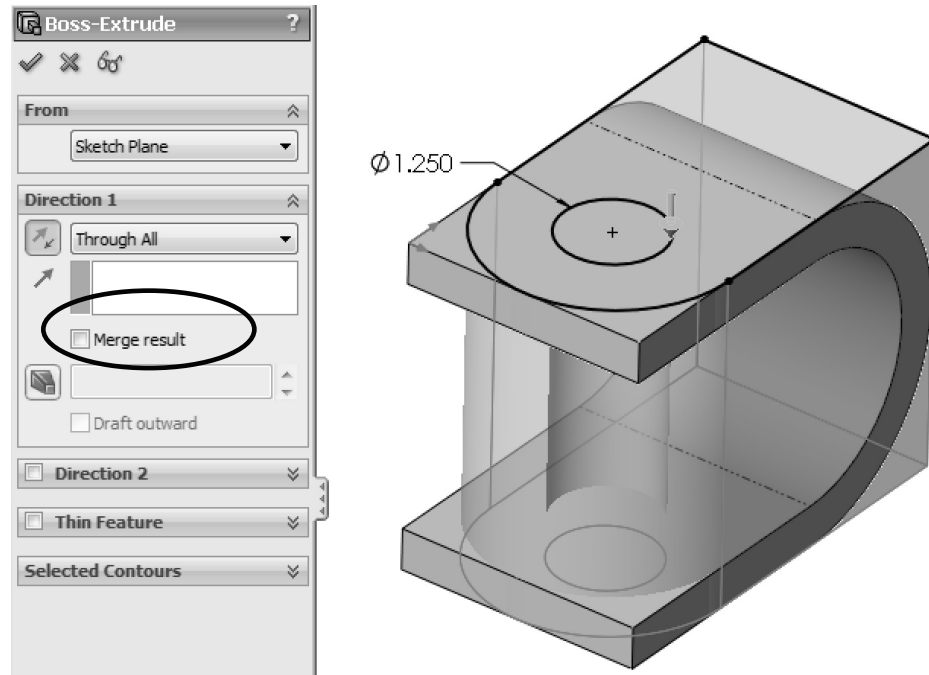


39. – For the next feature, add a sketch in the Top face of the first feature. Looking at it from a Top view should look like this:

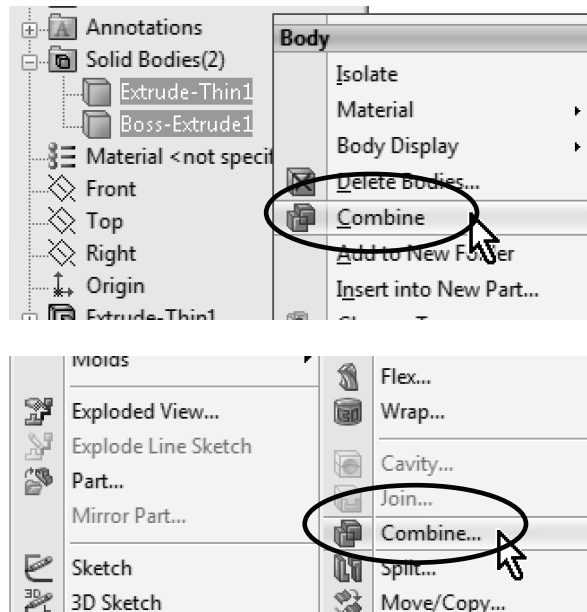


Notice the only dimension we need to add is the hole's diameter; everything else is defined with geometric relations.

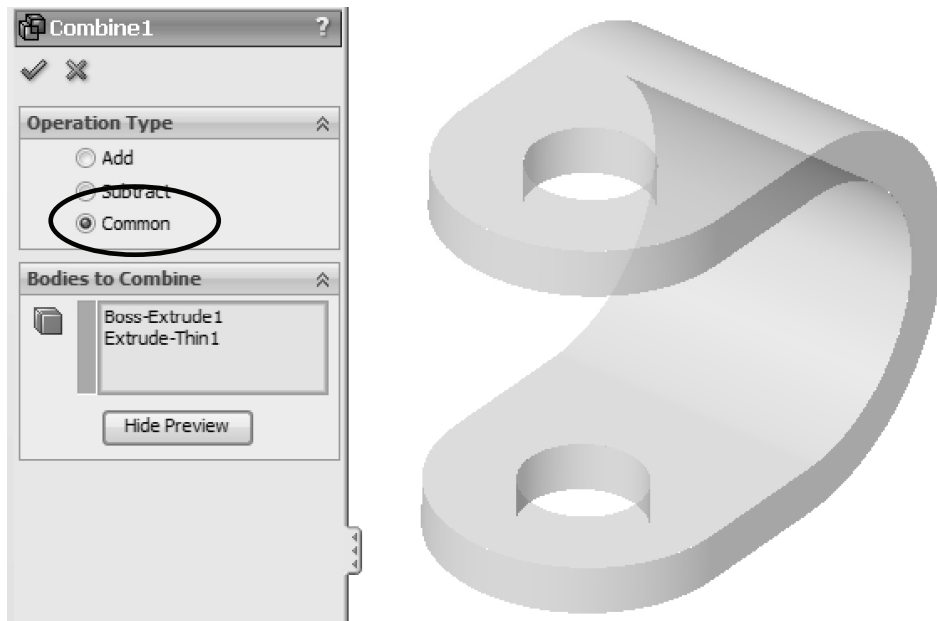
40. – Extrude the second sketch downwards with the “**Through All**” end condition, and uncheck the “**Merge result**” checkbox. We want to have two overlapping bodies.



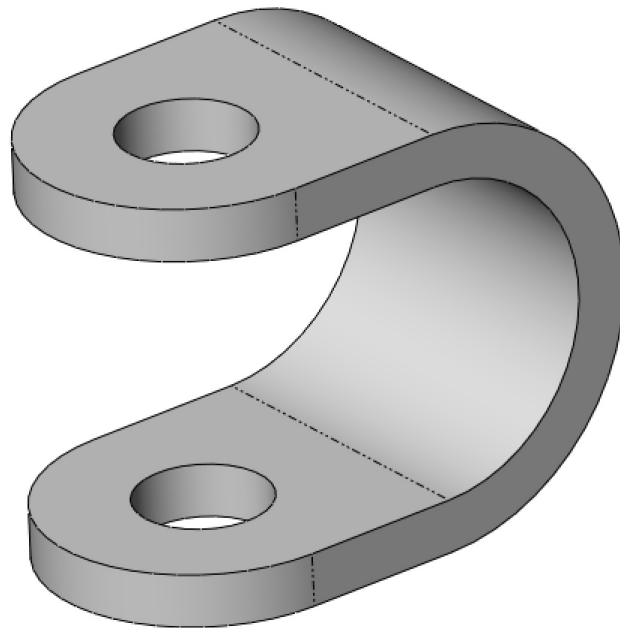
41. – Now that we have two separate bodies, we can combine them to get the common volume. Select the menu “**Insert, Features, Combine**” and select both bodies, or from the “**Solid Bodies**” folder in the FeatureManager pre-select BOTH bodies; right mouse click and select “**Combine**” from the pop-up menu.



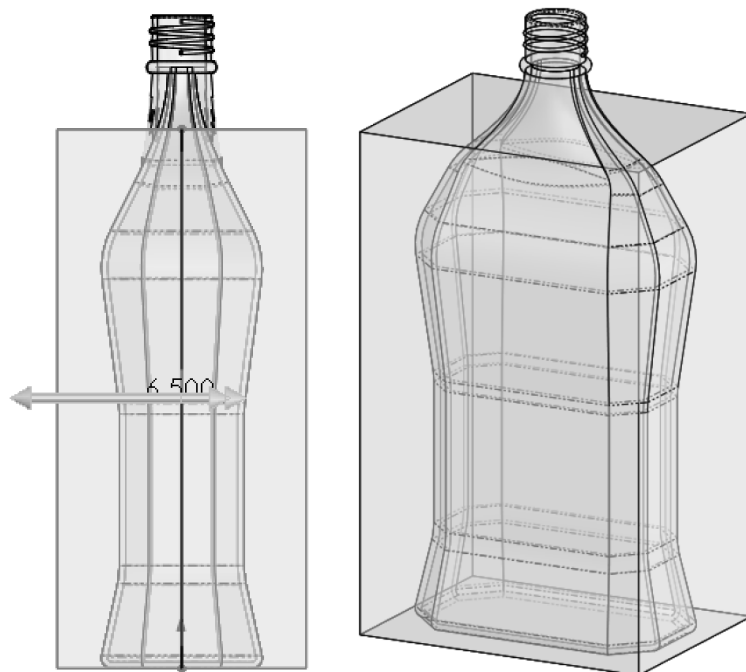
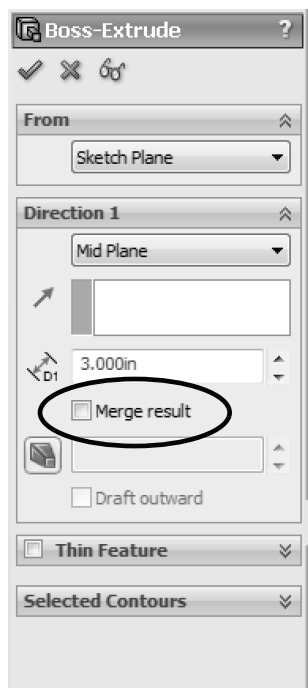
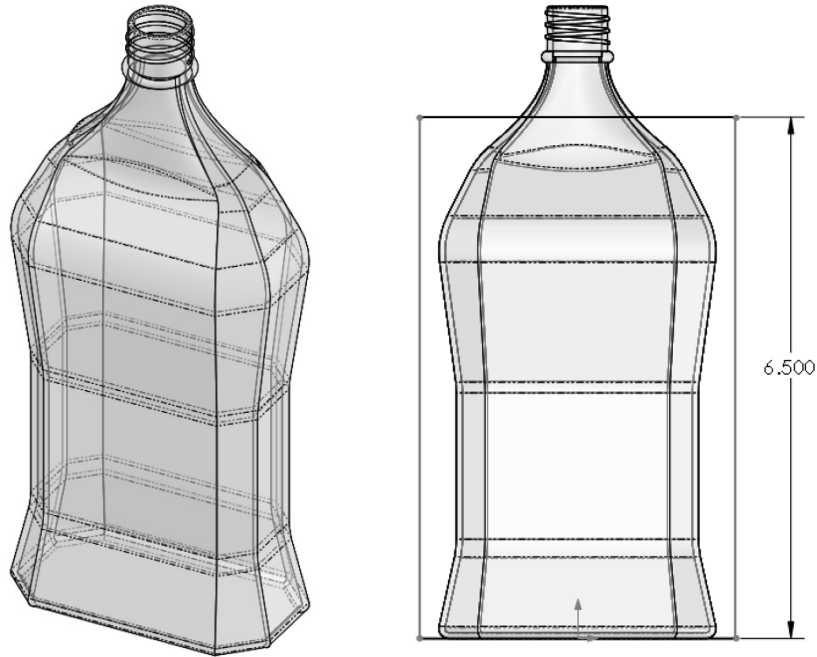
42. – In the Combine operation select the “**Common**” option under “**Operation Type.**” Click on the “**Preview**” button to see what the resulting body will look like and click **OK** to finish.



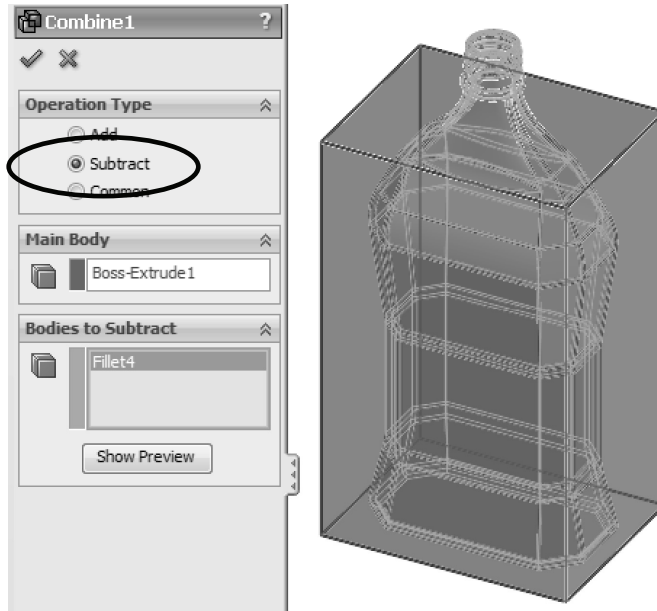
The finished part will look like this.



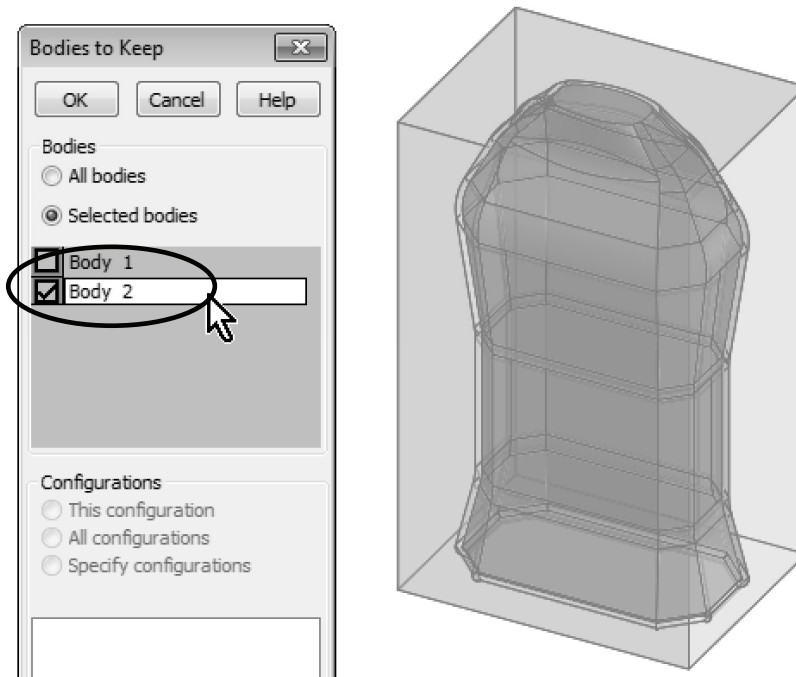
43. – The next multi body operation we'll cover is the difference between bodies. One common use for a body difference is to obtain a mold's core and/or cavity, but here we'll show how to obtain the volume capacity of an irregularly shaped bottle. Download the part *Bottle.sldprt* from our website and open it (www.mechanicad.com/download.html). In order to obtain a volume with the inside capacity of the bottle, we need to make a new solid body that will enclose the bottle up to the fill level. Make a new sketch in the Front plane and draw a rectangle as shown. Make the extrusion big enough to cover the entire bottle and uncheck the **"Merge result"** option.



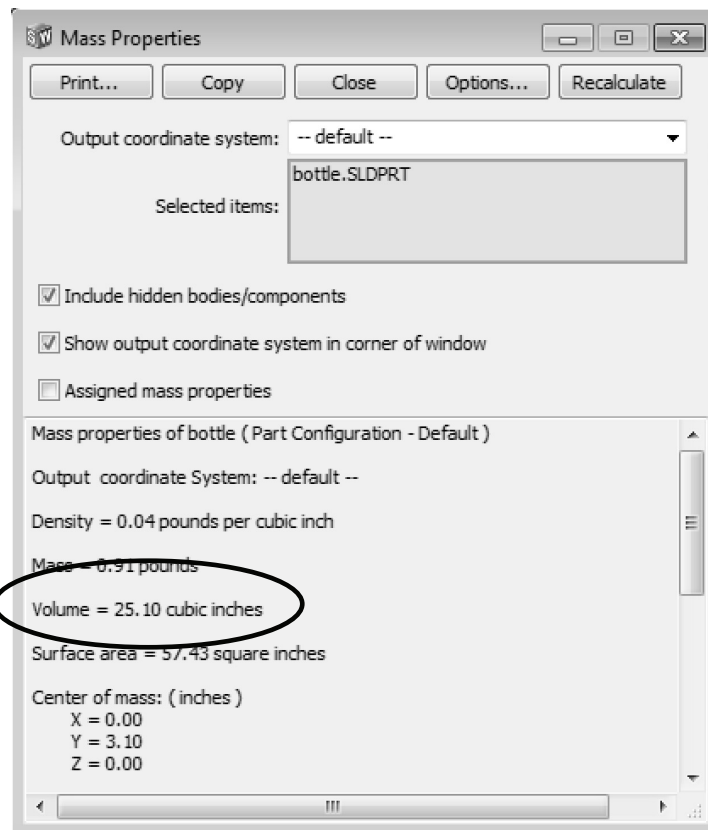
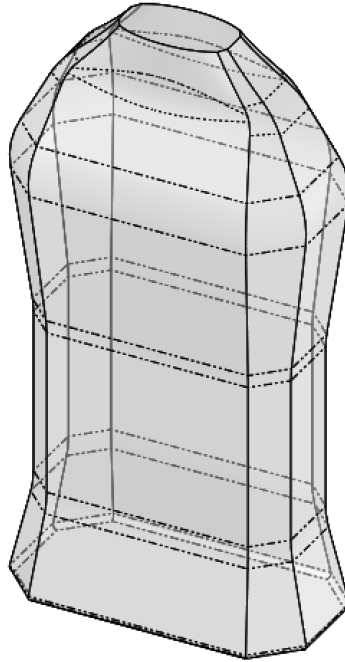
44. – Now that we have the two solid bodies, select the menu “**Insert, Features, Combine.**” Select the “**Subtract**” option under “**Operation Type.**” In order to get a difference, we need to select the body that we want to remove material from (Main Body) and the body(ies) that we want to remove from it. In the “**Main Body**” selection box, select the body just created, and under “**Bodies to Subtract**” select the Bottle. Click **OK** when done selecting.



45. – We are immediately presented with the “**Bodies to Keep**” dialog. There are two bodies resulting from the operation, but we are interested only in the inside body. Select the inside body and click **OK** to finish.



46. – Now we have the actual volume of the bottle up to the fill line. Notice that when we subtract one body from another, the original bodies are consumed and we are left only with the difference. After running a “**Mass Properties**” analysis, we can see that the volume of liquid inside the bottle up to the fill line is 25.1 cubic inches. Save and close the part.

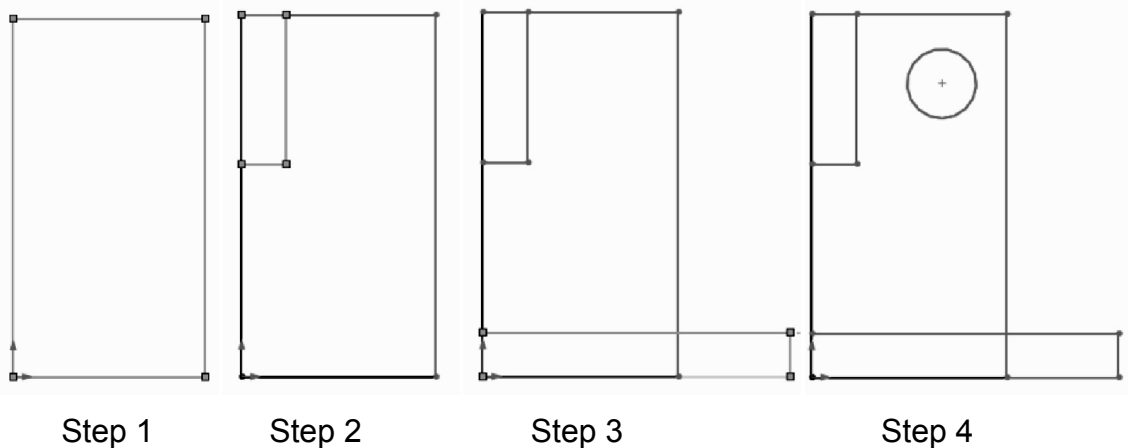


Notes:

Contour Selection

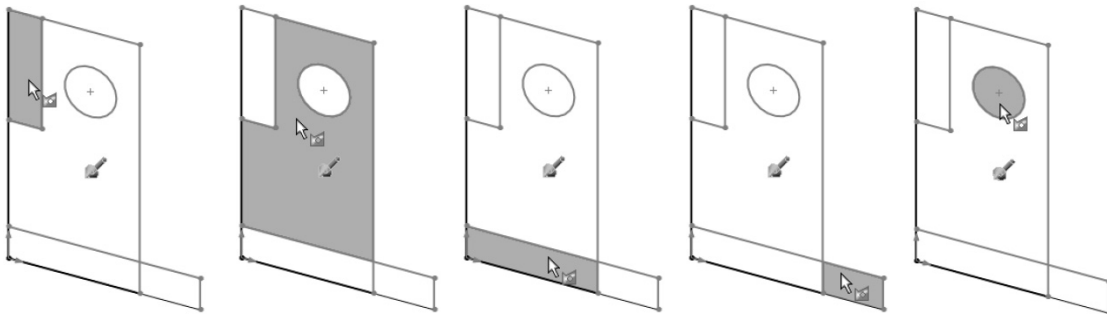
Contour Selection is a way to work with a sketch that has intersecting entities, endpoints shared by multiple entities and many of the common problems that prevent us from using a sketch for a feature. We'll also learn how to reuse a sketch for multiple features and a previously unused Extrude/Cut option.

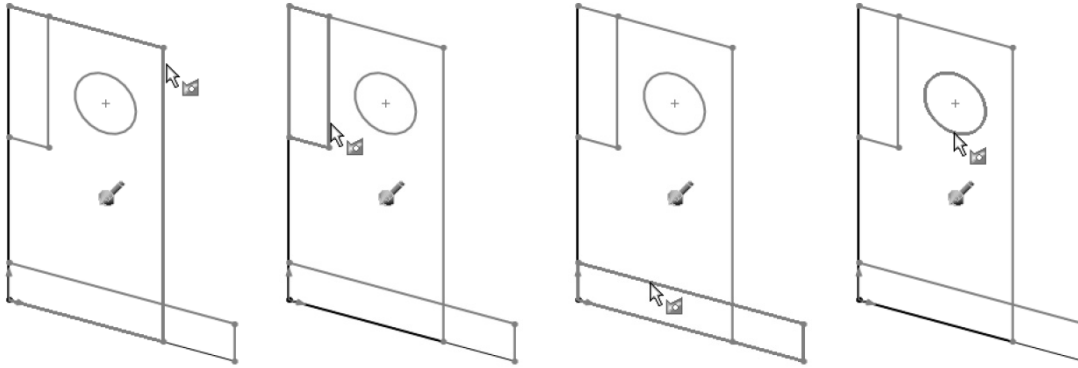
47. – Make a new part and add a sketch to the Front plane as shown. In this example we will not worry about dimensions to simplify the explanation and concentrate on how contour selection works.



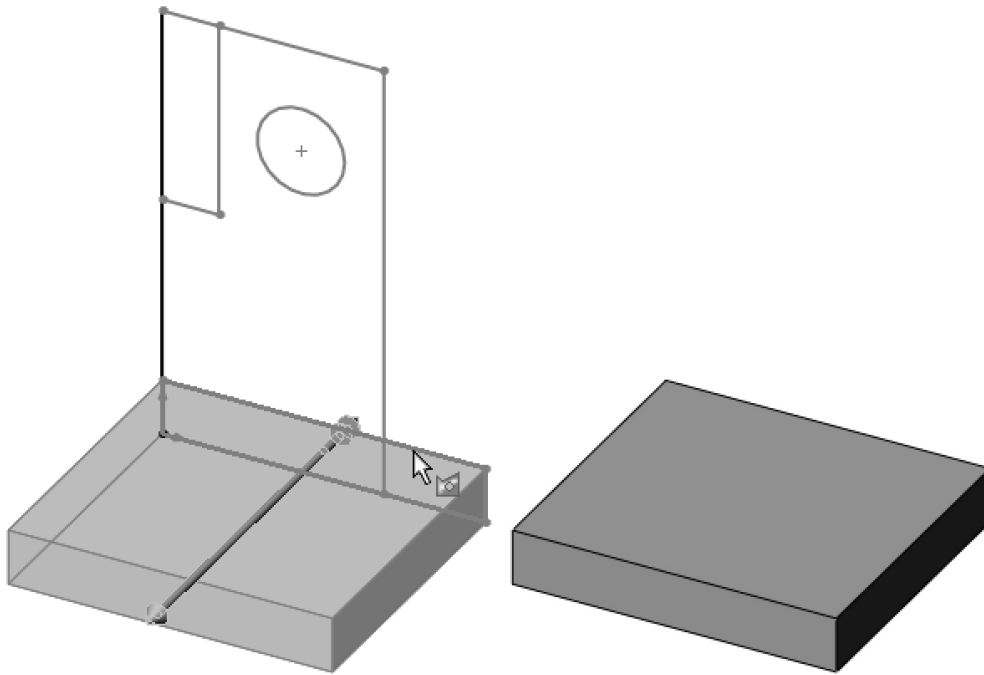
In this sketch we are drawing three rectangles and one circle. Note that the rectangles are overlapping and sharing endpoints.

48. – When we try to create a feature using a sketch with overlapping and/or shared endpoints, the Contour Selection tool is automatically activated and the “**Selected Contours**” selection box is open. Also notice we don't get an Extrusion/Cut preview until we select the region(s) or contour(s) that we want to use in the feature. Select the Boss-Extrude icon. In the graphics area move the mouse pointer around and see the different regions available for selection. We can select single or multiple regions *and/or* complete closed contours.

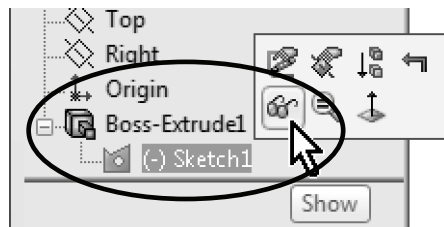




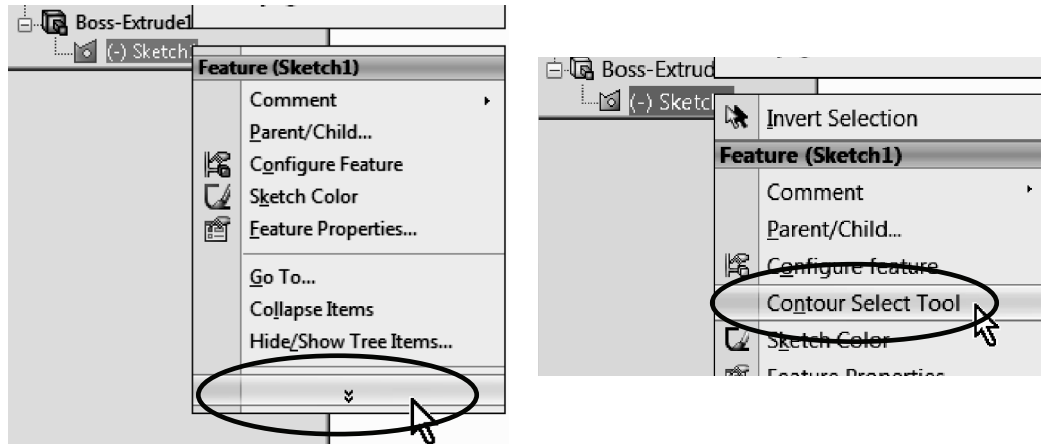
49. – Select the bottom square profile (or both bottom regions) and extrude *approximately* as shown. Just like with any other feature the sketch is automatically hidden after we finish.



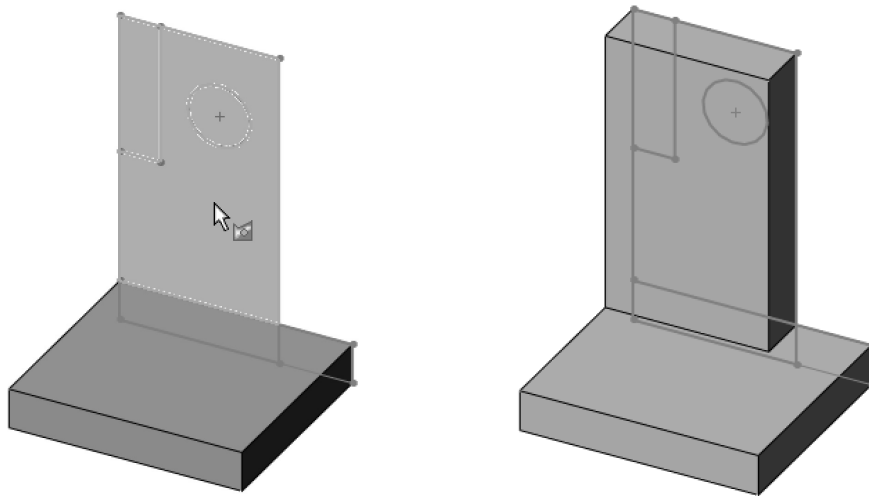
50. – To re-use the sketch for more features, select the sketch in the Feature-Manager and click “**Show**” in the pop-up menu. Note the “**Contours**” icon next to the sketch’s name.



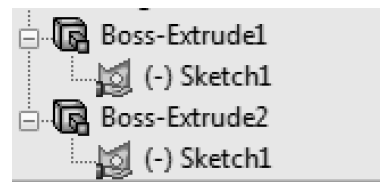
51. – We will now use the same sketch again for a new feature. To activate the Contour Selection tool we have to make a right mouse click in the graphics area (or the sketch itself). Be aware that by default the “**Contour Select Tool**” option is not available in the pop-up menu; you have to expand the menu at the bottom to make it visible.



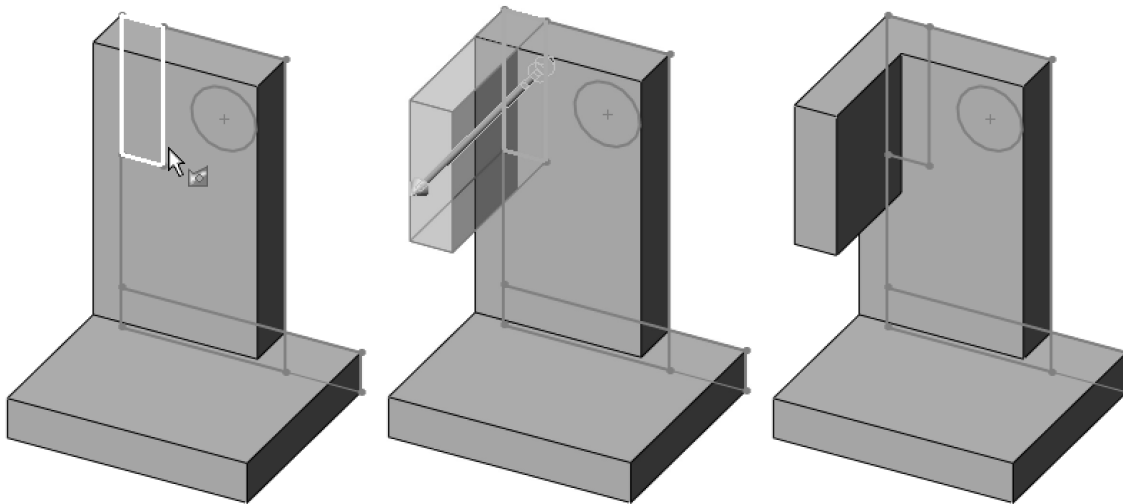
52. – After selecting the “**Contour Select Tool**” option select the regions indicated and extrude *approximately* as shown. You may have to click in the sketch to enable (activate) selection of regions, and either: hold down the Ctrl key to pre-select all three regions and then extrude, OR select the Extruded Base command *and then* select the regions. Either way works the same.



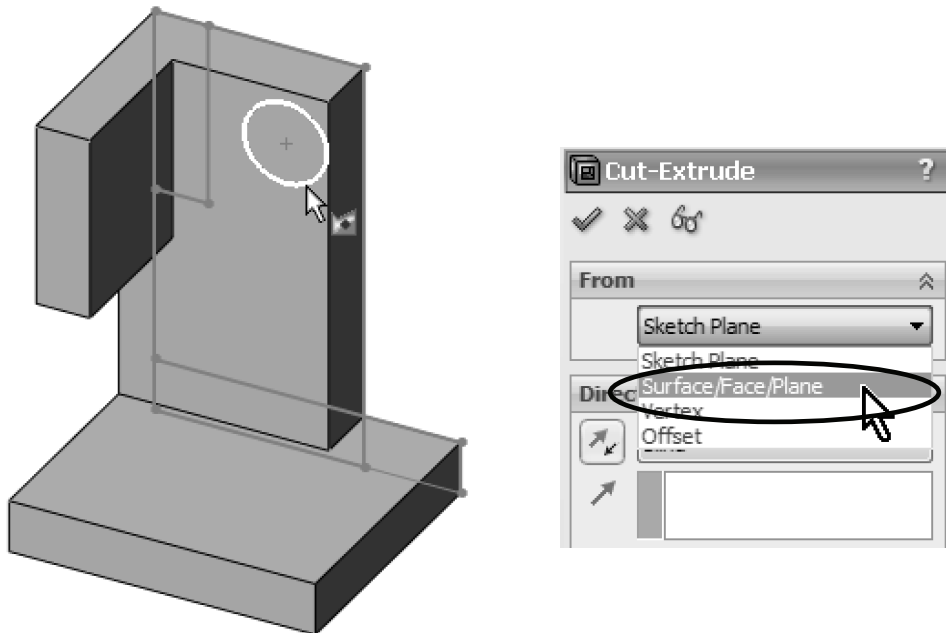
After using the same sketch for two or more features we can see a slightly different icon next to the sketch name with a little hand under it. This means the sketch is “shared” by more than one feature, and the sketch name is the same. Also note that the sketch remains visible after we use the “Show” command.



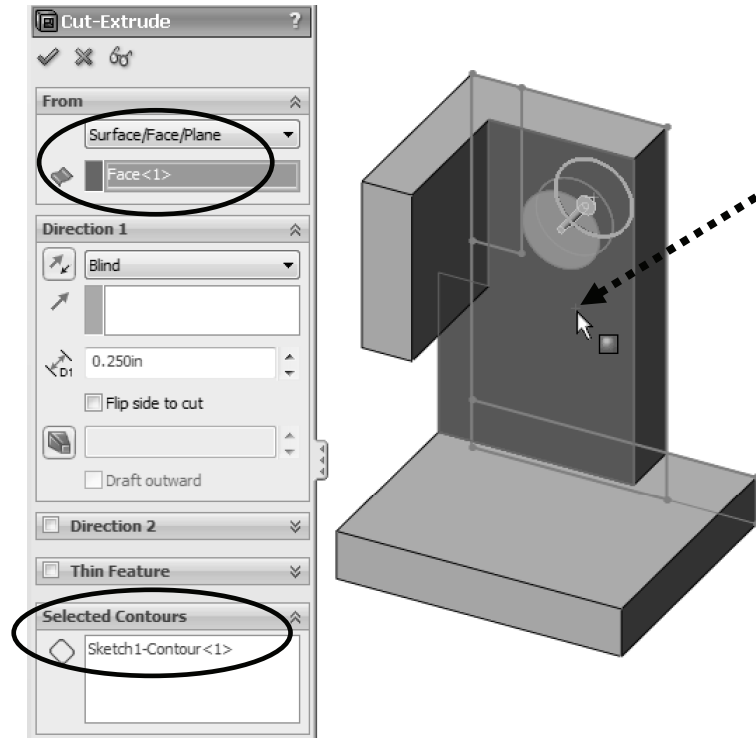
53. – Repeat the previous process to select the next contour and extrude as shown. Remember at this time we are only showing how it works and are not concerned about the dimensions.



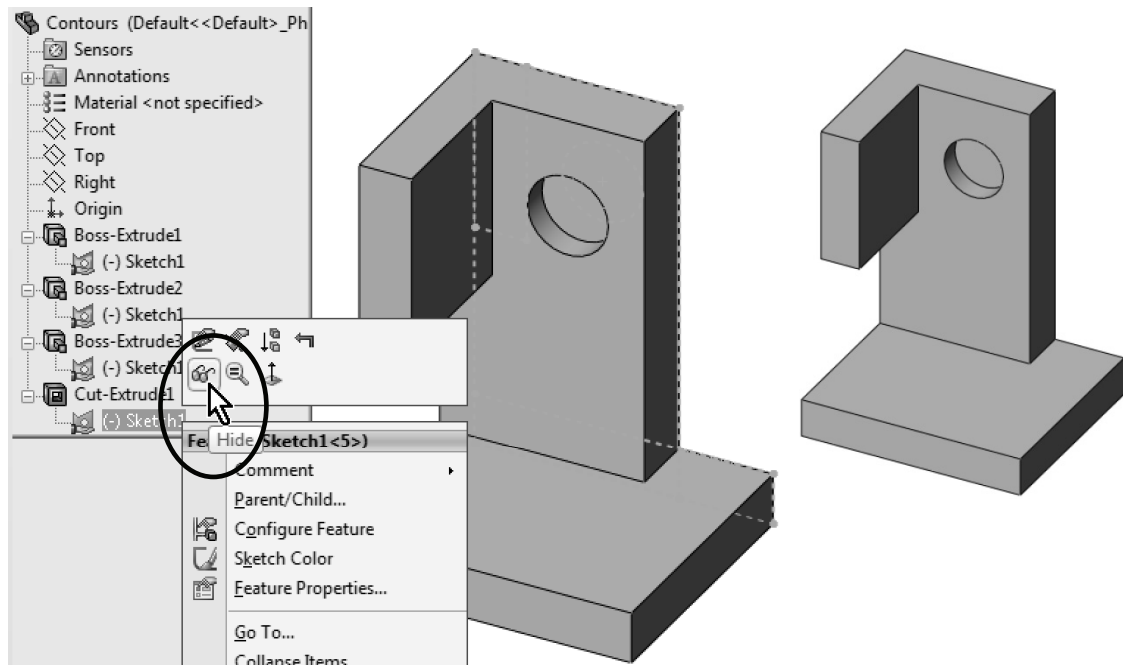
54. – For the final feature select the **“Contour Select Tool”** and select the circle. What we have done up until now is to make features starting in the sketch plane. SolidWorks has a powerful (yet sometimes under used feature) that allows us to start the feature somewhere *other* than the sketch plane. After selecting the circle using "Contour Select," click in the **Extruded Cut** command; in the **“From”** start condition's drop-down menu select **“Surface/Face/Plane”** (we can also use a vertex or distance offset).



55. – Now select the front face of the second boss as indicated. Our Cut feature will start here. Make the feature's depth *about* half way deep, and click **OK** to finish. By using the “**Start Condition**” option for feature creation we can easily save time by not having to create auxiliary planes or geometry.

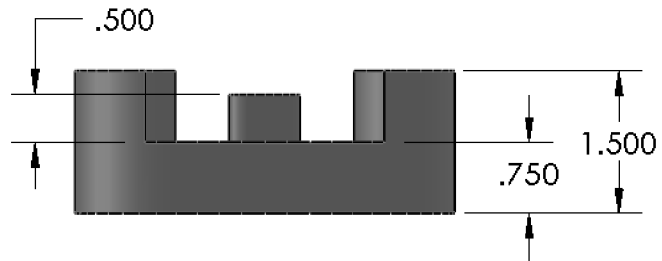
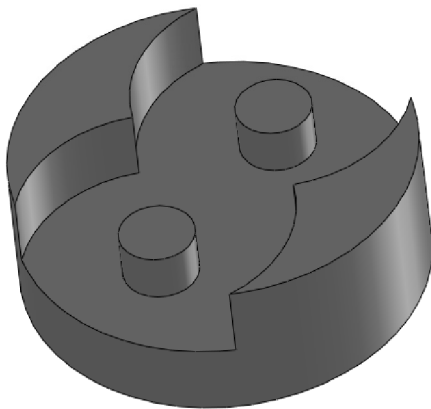
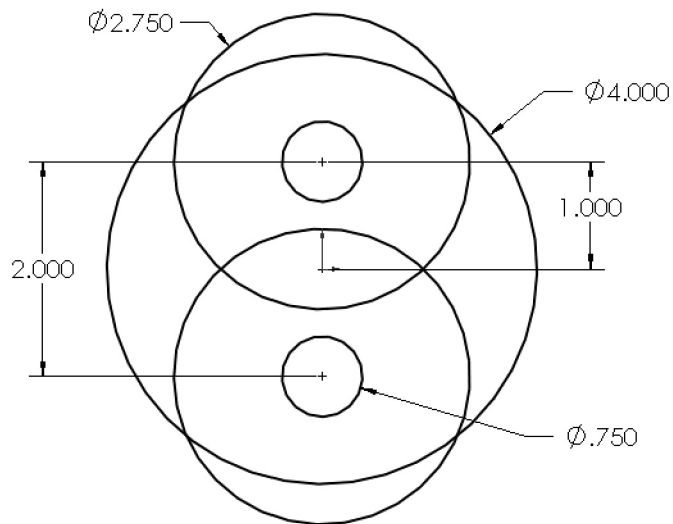


56. – Now that we have made four features from the same multiple contour and self-intersecting sketch, we can hide it.



Contour Selection Review

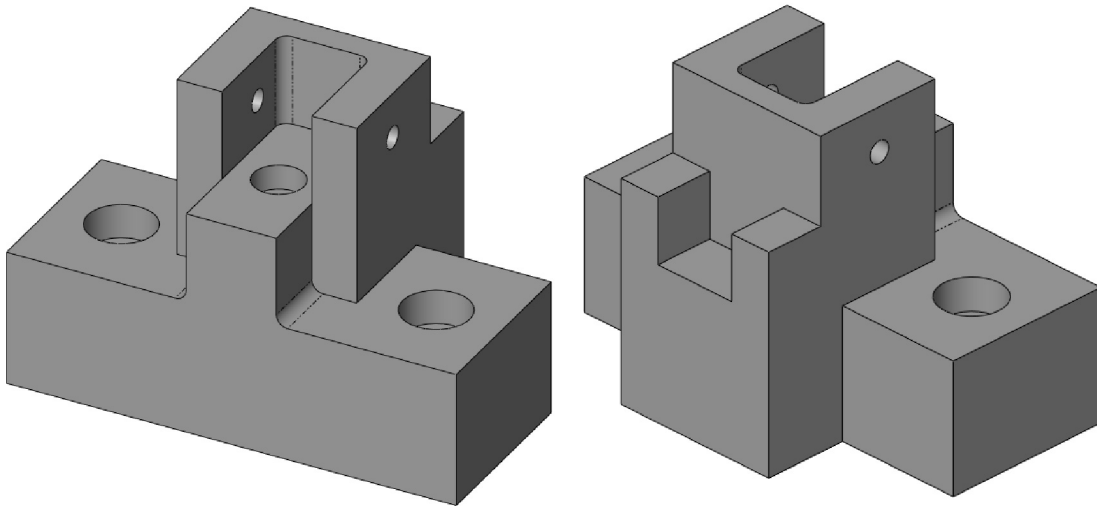
Make the following sketch and make all four features off of it using contour selection.



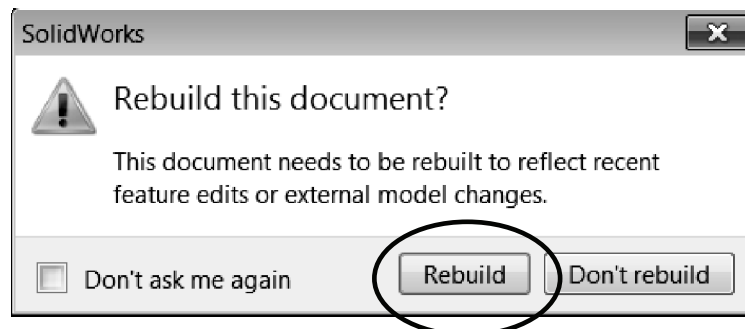
Part Editing

A very important skill to have when modeling in SolidWorks (or any CAD package for that matter) is to be able to change a model and fix errors. Let's face it: the only constant in design is change, and when we make changes to our model, chances are we may cause errors down the road, that is, the Feature-Manager. For example, if we have a part with round edges (fillets) and we change a previous feature and eliminate an edge, the fillet will give us an error because it cannot find it. That's the type of error we are talking about.

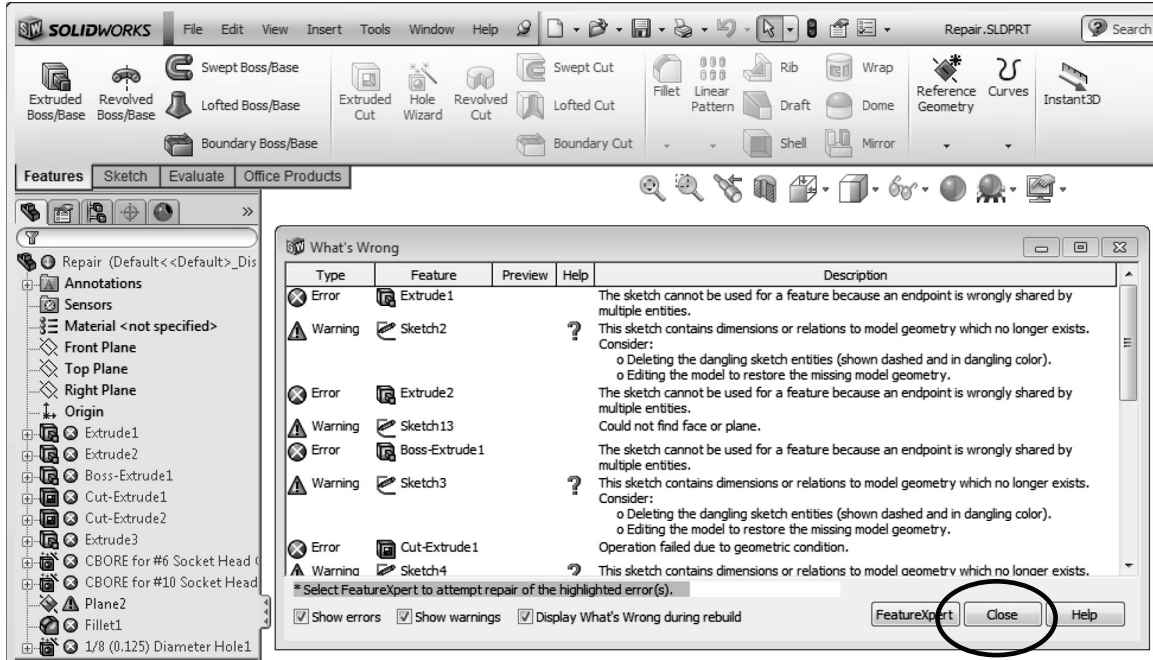
57. – To practice editing and fixing errors, download the part *Repair.sldprt* from www.mechanicad.com/download.html and open it. After we are done modifying and fixing the part, it should look like this:



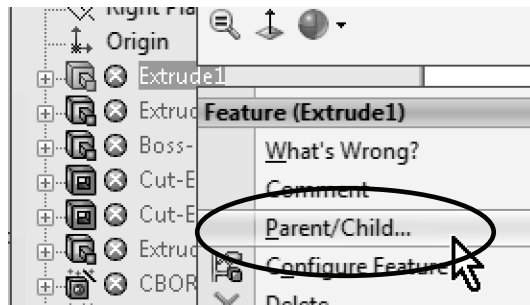
58. – When we open the file we are asked if we want to rebuild it. Select **“Rebuild”** from the dialog to continue.



59. – When rebuilt, we get a list of error messages and no geometry at all.

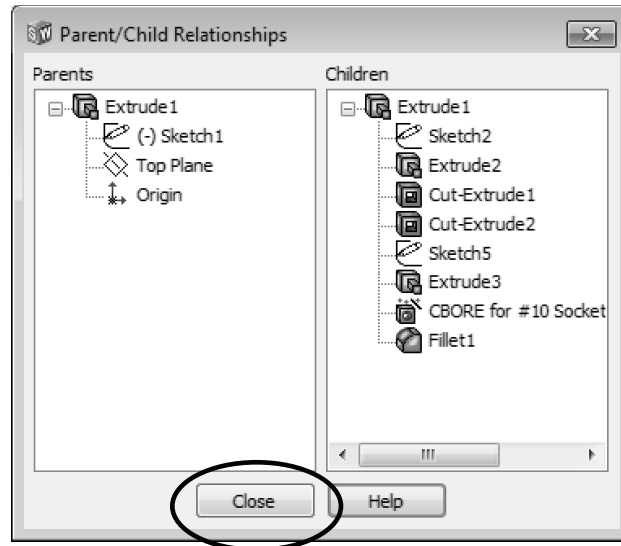


60. – This part has so many errors that no geometry can be generated, and the “What's Wrong” dialog contains the full list of things that need to be fixed. Click the “Close” button for now. Since features are added chronologically starting at the top in the FeatureManager, the logical order to start fixing errors is from the top and work your way down. The reason is if a ‘parent’ feature has an error, it may cause problems in a ‘child’ feature, therefore the dependency between features is referred to as “Parent/Child” relations. For example, if a sketch is added to a face of another feature, or a dimension references another feature’s geometry, a Parent/Child relation is generated. To identify these relations, click in the “Boss-Extrude1” feature with the right mouse button and select “Parent/Child.”

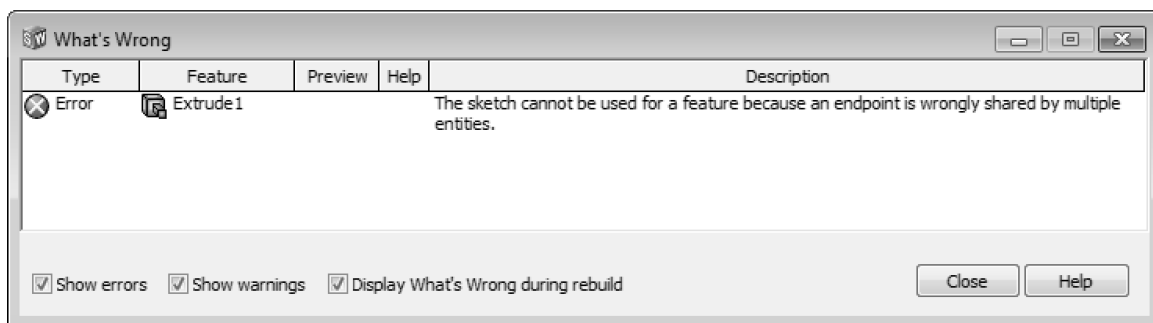
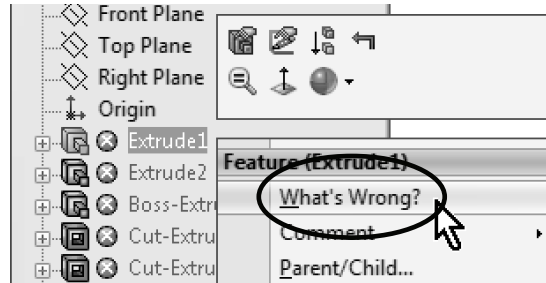


There are two types of errors: a red X means the feature failed to build; the yellow warning triangle means the feature has an error, but SolidWorks was able to build it.

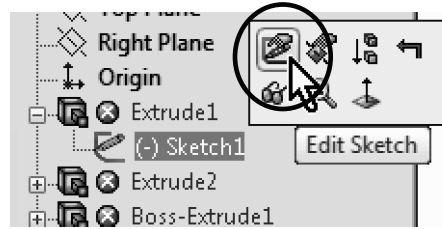
61. – Here we can see the features that “Boss-Extrude1” depends on (Parents), and which features depend on it (Children). The higher we go in the FeatureManager, the more Children a feature may have. Click “Close” to continue.



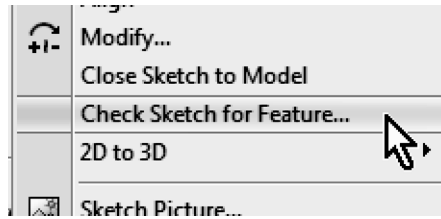
62. – To start fixing errors, we’ll check what the error at the “Extrude1” feature is. Right mouse click on it and select “What’s Wrong?”



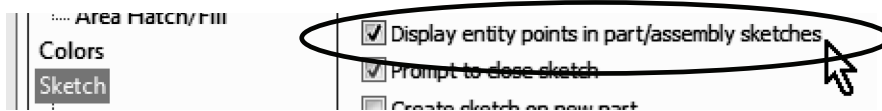
This error means that the sketch has intersecting lines or more than 2 lines connected to the same endpoint. One common cause of this problem is when sketching we accidentally add overlapping lines. Close the “What’s Wrong” dialog, expand the “Extrude1” feature and edit the sketch.



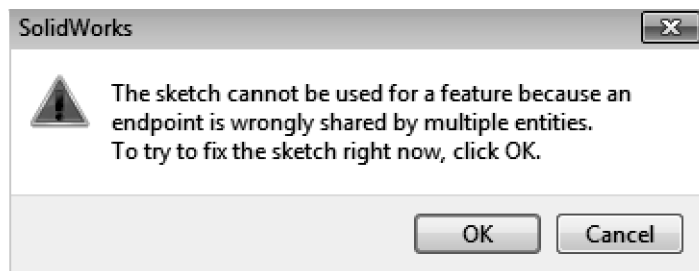
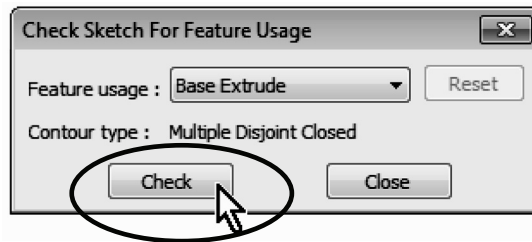
63. – Sometimes it's easy to see the geometric elements causing the problem in a sketch and we can correct it, but sometimes it's not that obvious. To help us identify the problem, select the menu **"Tools, Sketch Tools, Check Sketch for Feature."**



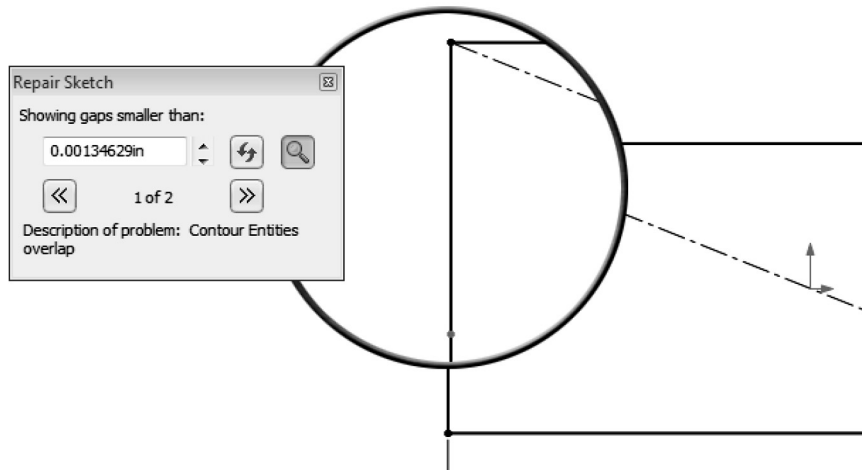
Setting the option to show sketch endpoints is usually a good option. Go to the menu **"Tools, Options, System Options, Sketch, Display entity points in part/assembly sketches."**



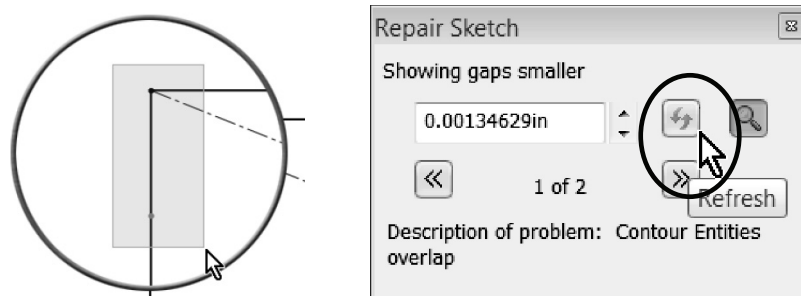
If this sketch has already been used for a feature, that feature type will be pre-selected in the drop-down menu. If the sketch has not been used, we have to select the type of feature that we intend to use it for. In our case "Base Extrude" is pre-selected. Click on **"Check"** to analyze the sketch. Immediately we see the same error message that we got using **"What's Wrong."** Click **OK** to dismiss it and continue.



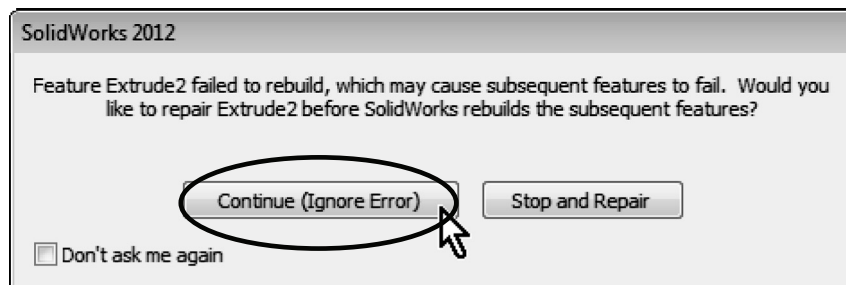
64. – SolidWorks immediately reorients the sketch and places the magnifying glass on top of the geometric element suspected of causing the problem. We are given two areas of concern: the first one is that we have overlapping entities, and the second one multiple (more than 2) elements sharing the same endpoint.



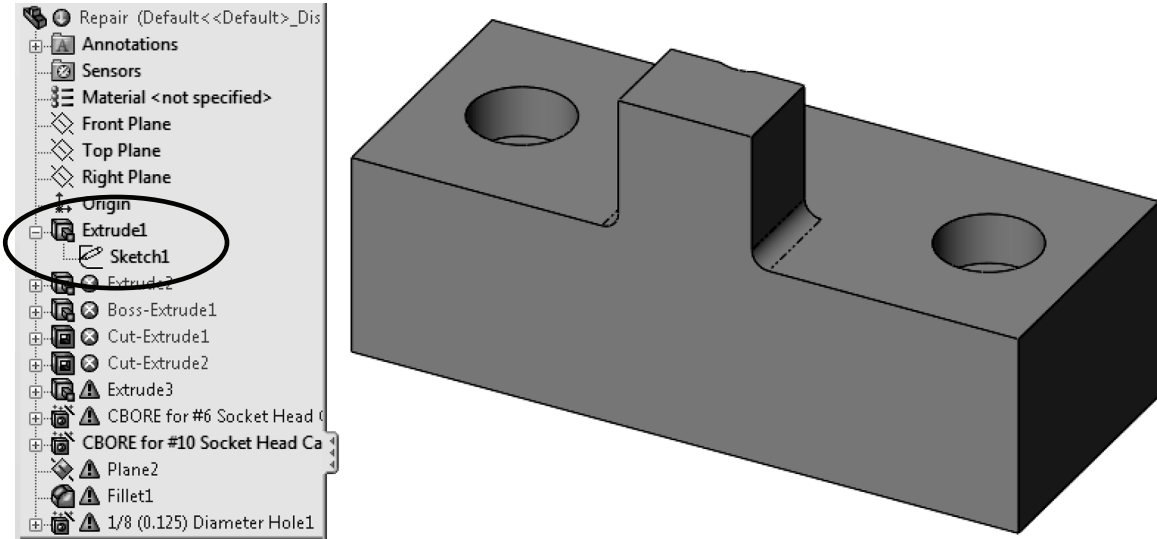
The advantage of using this tool is that we can quickly identify where the problem is and correct it, instead of hunting down small line segments in a sketch. Now that we know where the problem is, window-select the overlapping line and delete it. Click **“Refresh”** to confirm that we don't have any more problems and close the “Repair Sketch” window.



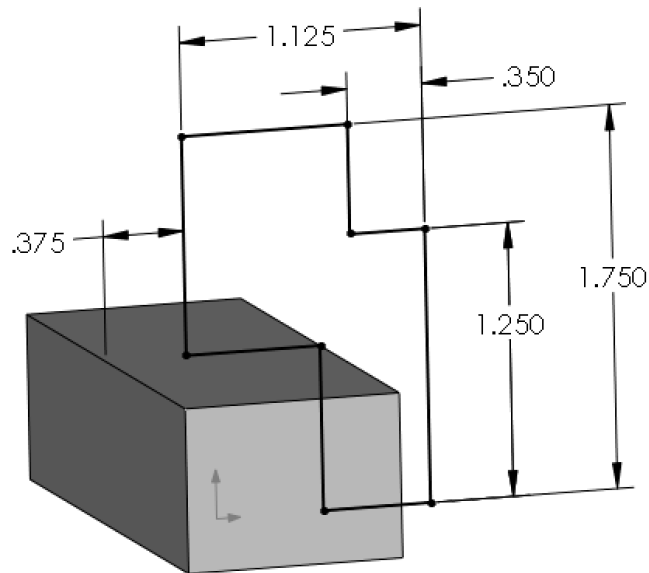
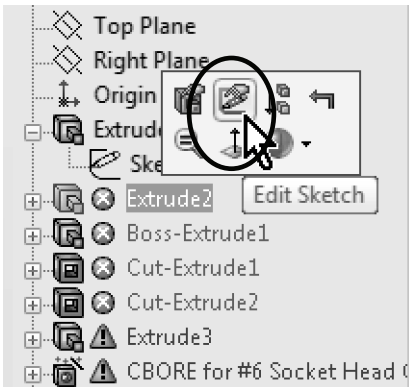
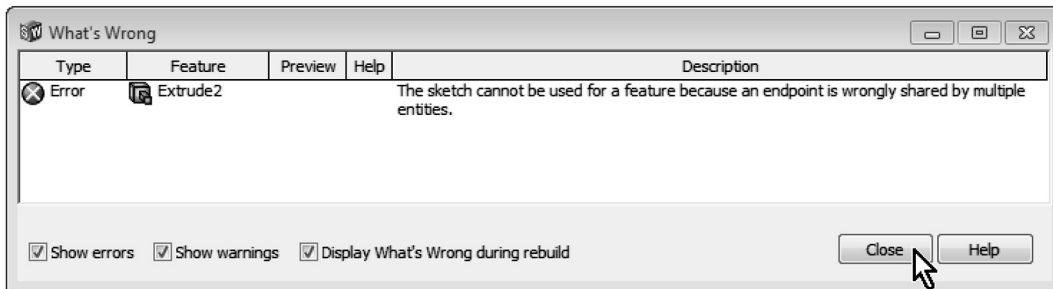
65. – Exit the sketch (or rebuild the model) to continue. We are notified that a subsequent feature has an error and we are asked if we wish to repair it now or continue with the error. Click on **“Continue (Ignore Error)”** and also close the **“What's Wrong?”** message.



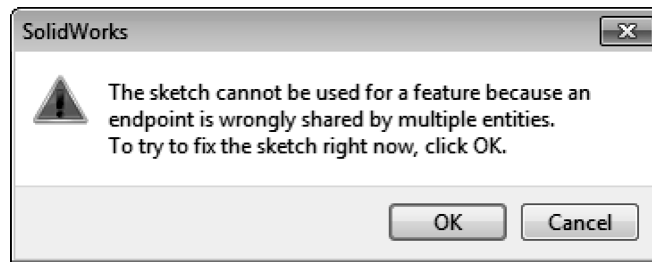
Now we can see some features and (more importantly) the error from the first feature is gone.



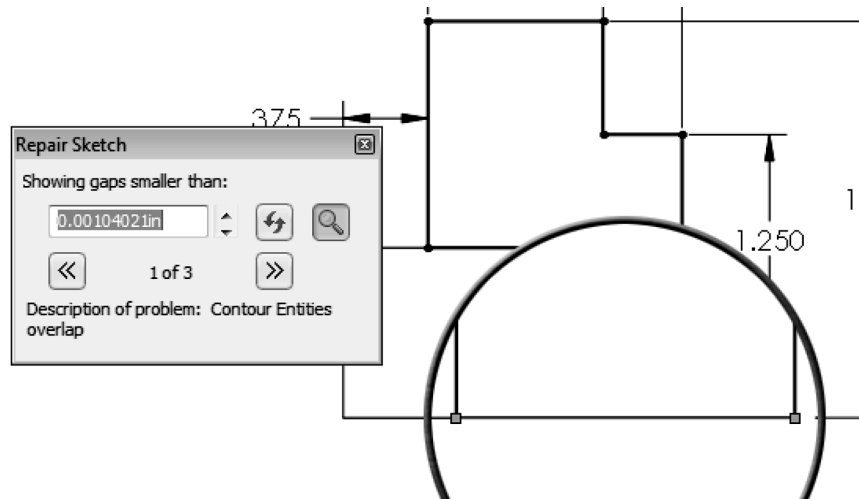
66. – Select the “Extrude2” feature, right mouse click and select “**What's Wrong?**” to view the error. Essentially we have the same problem as the first feature. Edit the “Extrude2” sketch to fix it.



67. – Once we are editing the sketch, select the menu “**Tools, Sketch Tools, Check Sketch for Feature**” to find out what the problem is. Close the error message when it comes up.

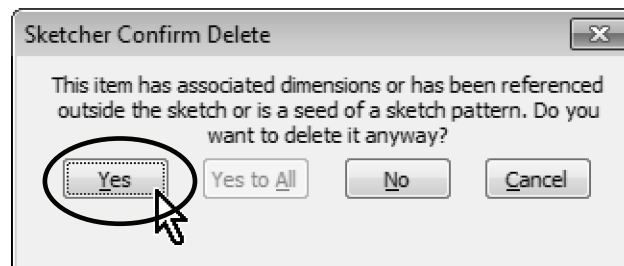


In this case, looking at all three problems listed in the "Repair Sketch" dialog (one overlapping entity and two with more than two entities at an endpoint), we can tell that we have two identical lines overlapping. Close the dialog and delete one.



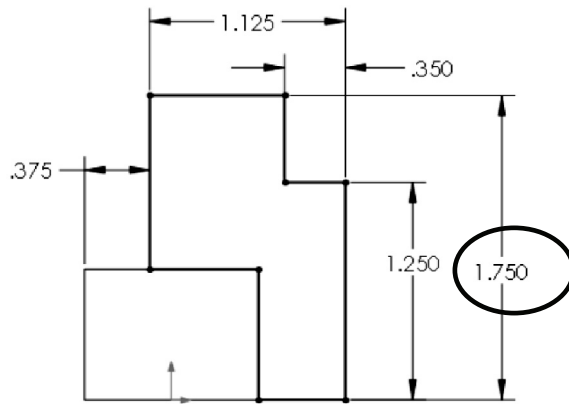
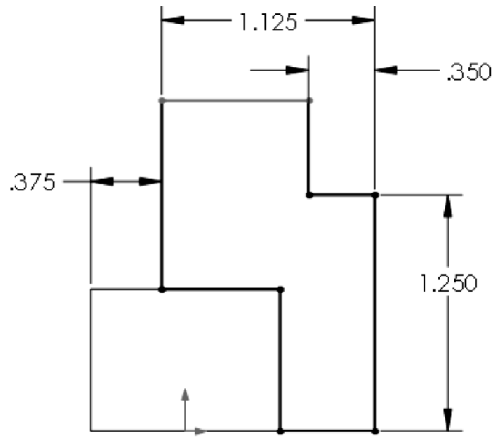
If we use window-selection we'll delete both lines; if we click to select one, only one is selected.

68. – After selecting one of the overlapping lines and deleting it, we may be warned that other entities will also be deleted, most likely dimensions attached to the line selected.

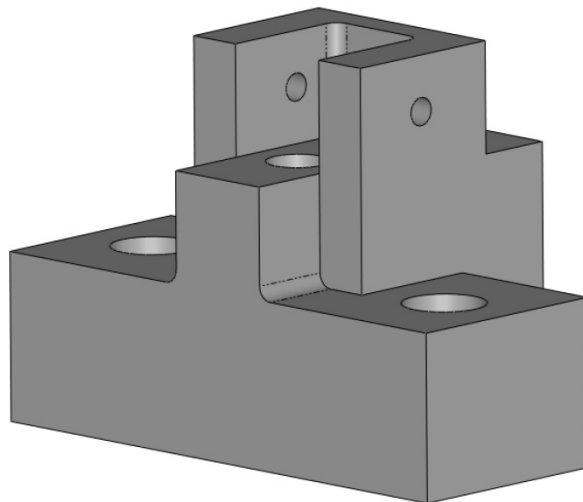


69. – If this is the case click “Yes” to continue. Notice the top horizontal line's dimension is also deleted. This is because that dimension was referencing the

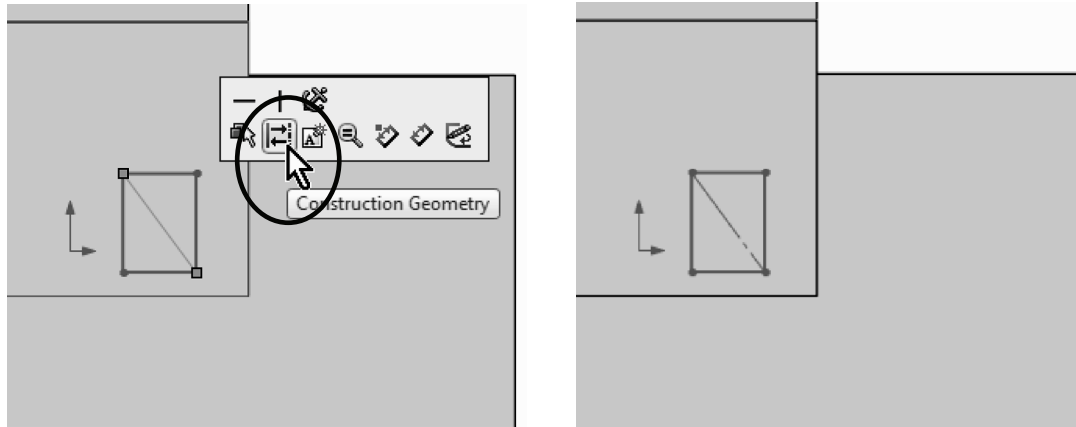
line we deleted. Add the missing dimension and exit the sketch. Select “Continue (Ignore Error)” and close the “What's Wrong?” message.



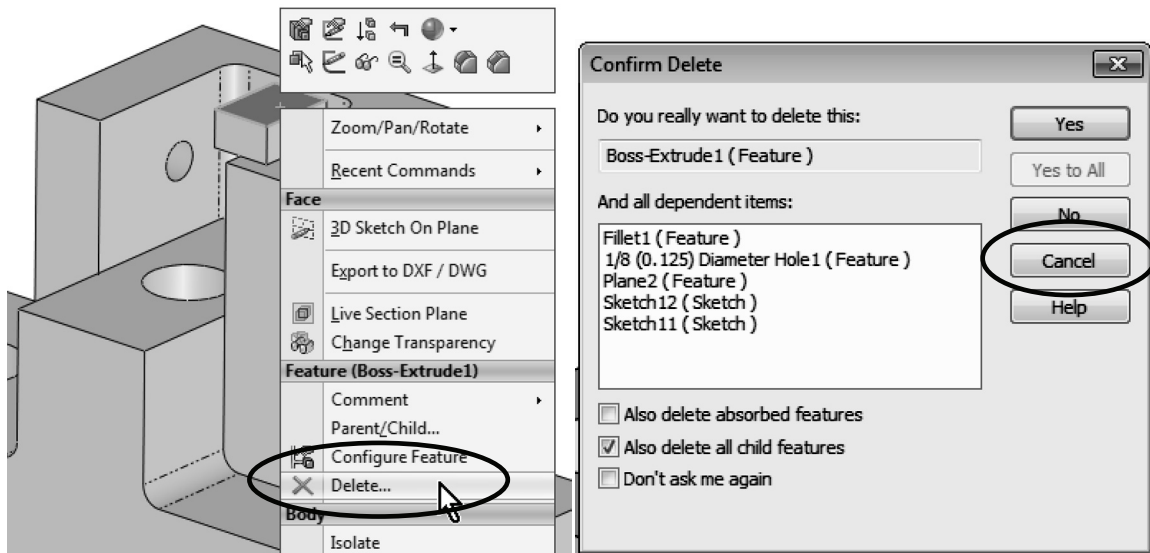
70. – Now our model is starting to look better. Notice that we have fixed only two features and we have cleared many errors, better illustrating the importance of understanding parent/child relations.



71. – After reviewing the next error with “**What's Wrong?**” we see that it’s the same error as the two previous features. Editing the sketch we can see an extra diagonal line. We can either delete it, or convert it to **Construction Geometry**. The second option is usually safer, as we could lose dimensions and/or relations if we delete it, and we can always revert it if needed. Select the diagonal line and convert it to construction geometry from the pop-up toolbar.

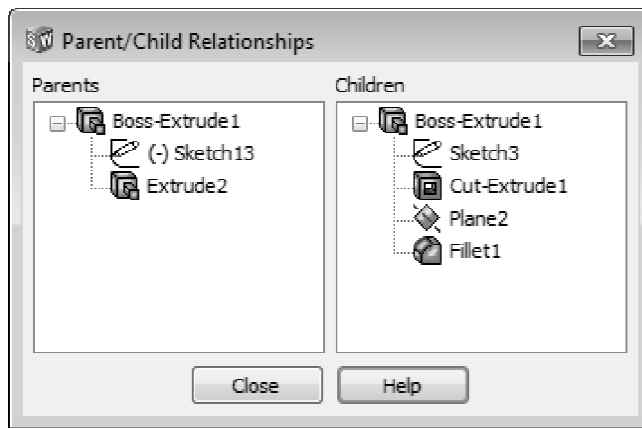
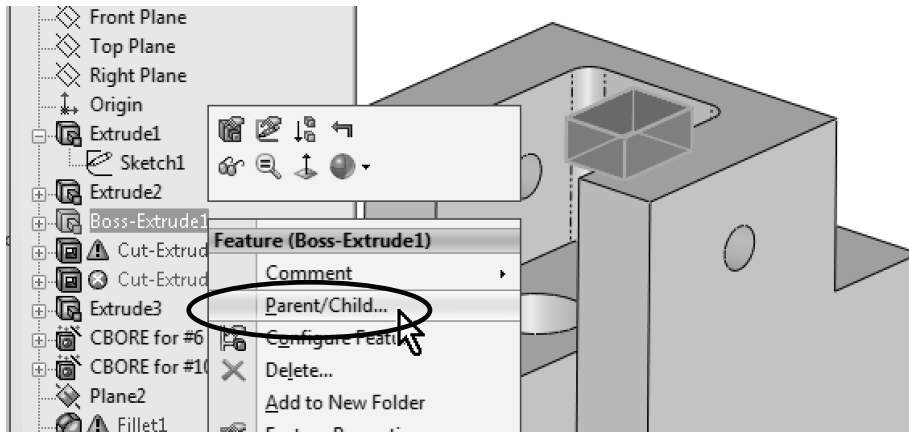


72. – Exit the sketch to rebuild the part. Since the newly created feature does not seem to be part of the original design, the logical step would be to delete it. Select the feature with the right mouse button and after selecting “**Delete**” we get the confirmation dialog:

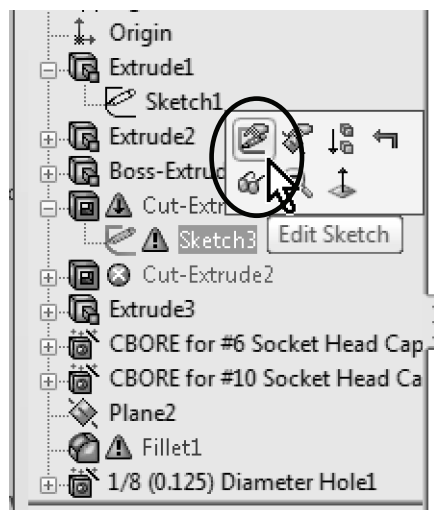


73. – In the confirmation we can see that if we delete this feature we’ll also delete its dependent features, but we need to have them in our part. The reason those features would also be deleted is because they are “children” features of “Boss-Extrude1.” Select “**Cancel**.” We don’t want to delete anything at this time; we’ll

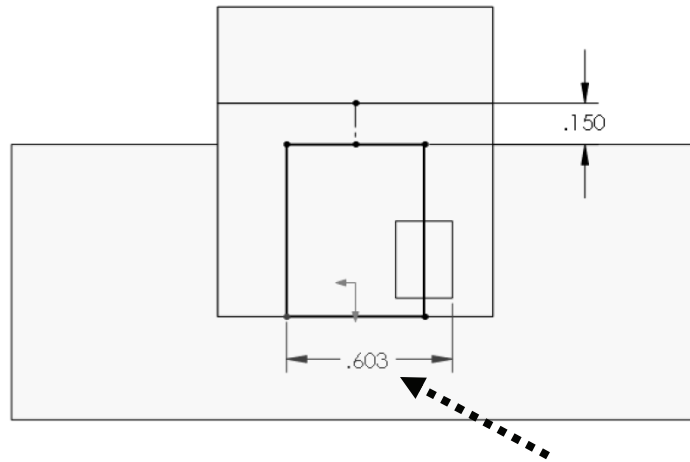
edit those features to remove the dependencies *and then* delete this feature. Select the extrusion's “**Parent/Child**” relationships.



74. – We can see that the first dependant feature is “Sketch3” (from “Cut-Extrude1”). Select it in the FeatureManager and edit it.



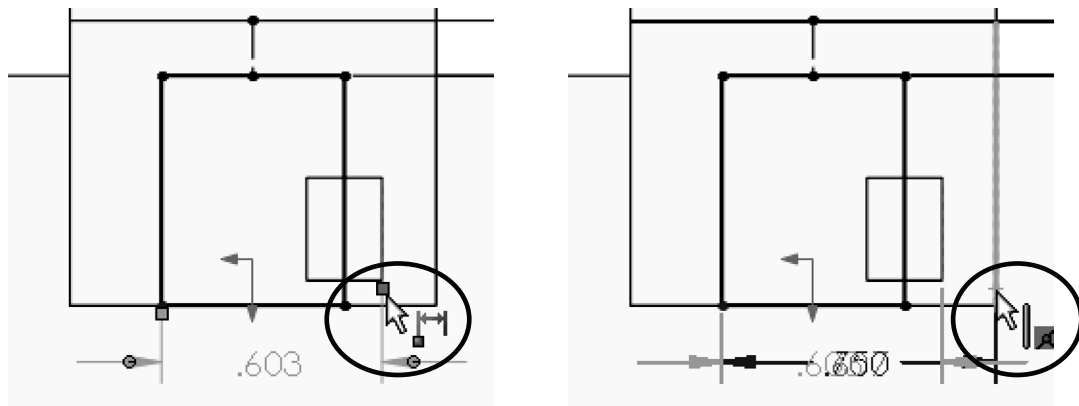
75. – Change to a Top view for clarity. We can see a dimension colored in brown (0.603). The brown colored dimension (or geometric relation) tells us it is '**dangling**,' which means that it is referencing something that no longer exists. What we need to do is to fix the dimension's reference.

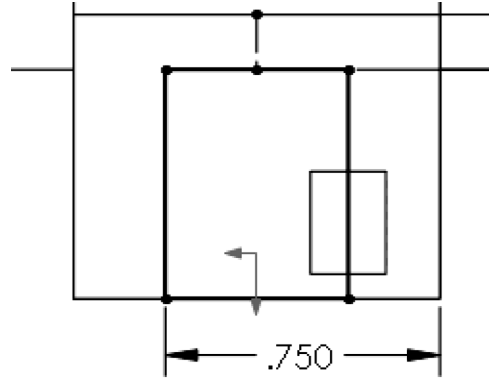


Here we can do one of two things:

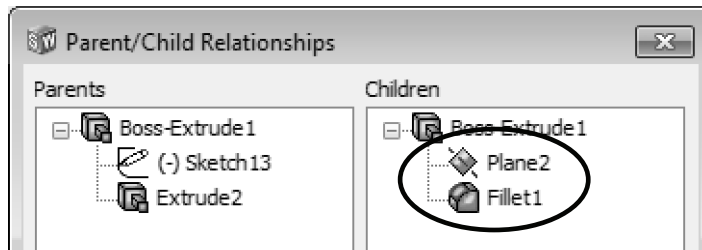
- Delete the dimension and add it again to a valid reference, or
- Re-attach the dimension to a valid reference.

Deleting the dimension is straightforward and often a good solution, so we'll talk about the second option. After selecting the dimension, we see a witness line ending with a red dot; this is the witness line missing the reference. It may look as if it was referencing the existing edge, but in fact it lost the edge it was attached to, even if it was rebuilt. To re-attach it, drag the red dot onto a valid reference. Here we'll use the right side edge. After re-attaching, note that the geometry does not change; the value of the dimension is what updates.

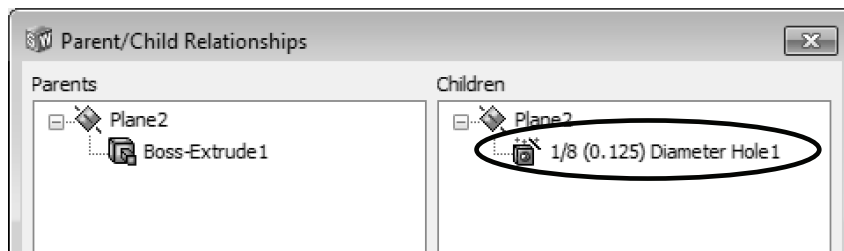




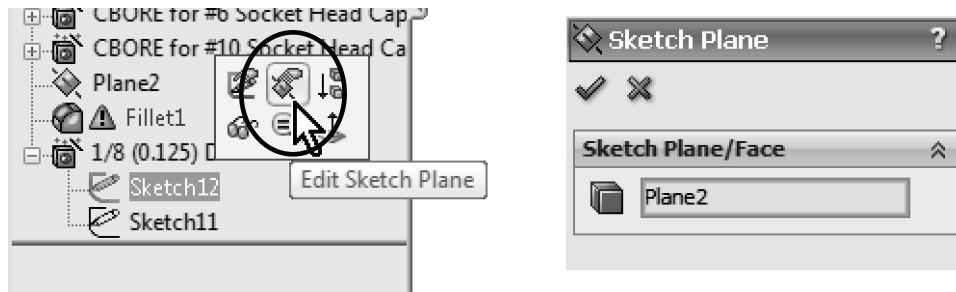
76. – Exit the sketch and review the “**Parent/Child**” relationships again (dismiss the “**What's Wrong**” dialog). As we can see, Sketch3 and Cut-Extrude1 are no longer listed as children features of Boss-Extrude1, since we changed the dimension to reference a different edge, in essence, breaking the relation.



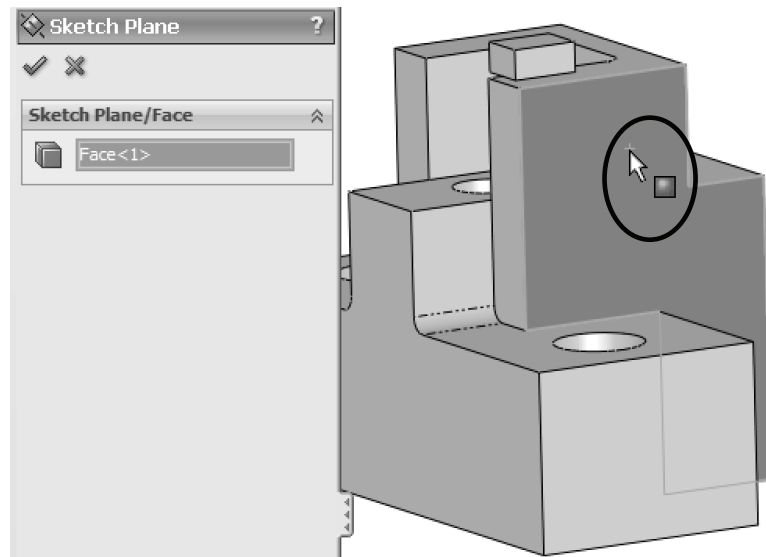
77. – From the relationships dialog we can see that the only children left are Plane2 and Fillet1. Reviewing the Parent/Child relations for Plane2, we see that the only child feature is the “1/8 (0.125) Diameter Hole1.” A Hole Wizard feature has two automatically made sketches; the first sketch locates the hole’s position, and is created in the plane/face where the hole is made, the second sketch is the hole’s profile.



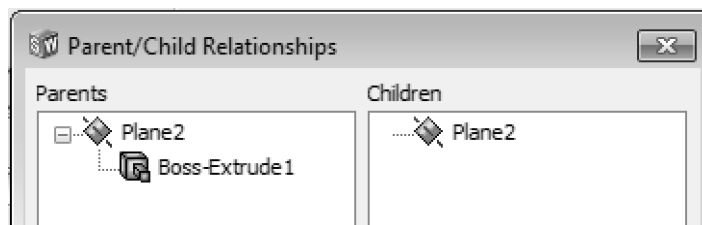
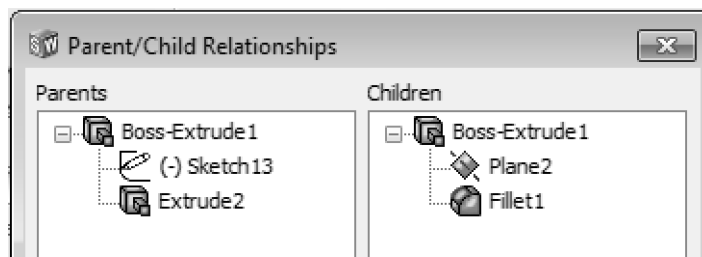
The most likely reason why the “1/8 (0.125) Diameter Hole1” feature is a child to Plane2 is that it was probably made in Plane2. We can find out in which plane or face a sketch is made by editing the sketch plane; to do this expand the Hole Wizard feature, select the first sketch (Sketch12) which is the hole's location sketch, and from the pop-up menu select “**Edit Sketch Plane.**” Since Plane2 is listed it confirms it as the current sketch plane.



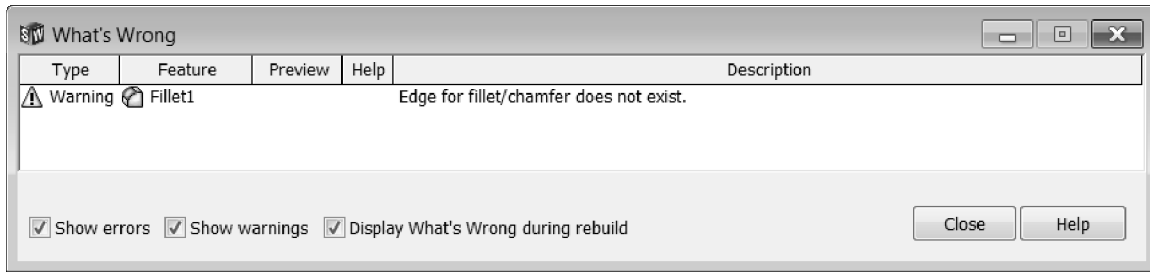
78. – To change the sketch to a different plane (or face) and delete the relationship to Boss-Extrude1, select a new face for the sketch. Select the face indicated and click **OK** to finish. Close the “**What's Wrong?**” dialog to continue.



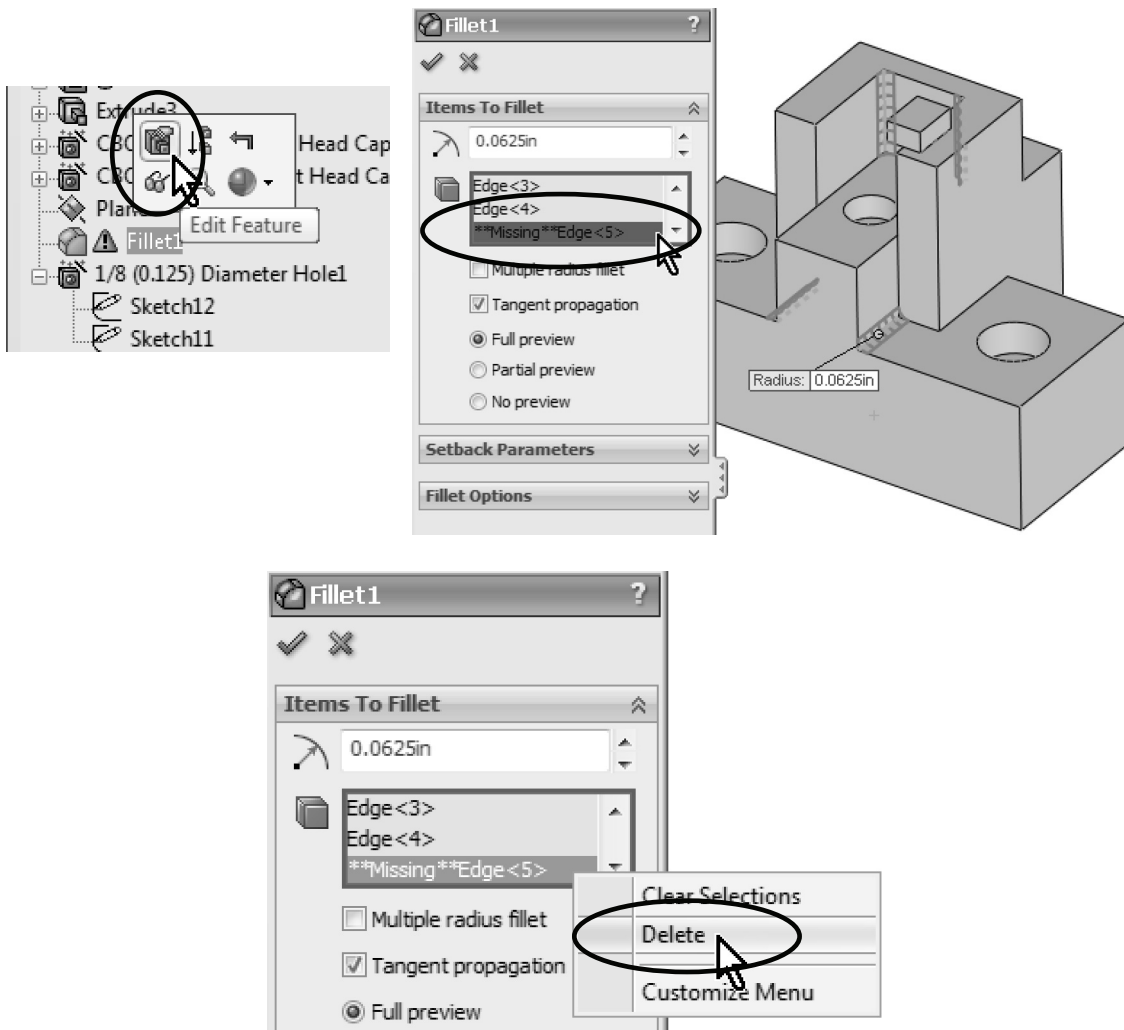
79. – Review the Parent/Child relationships again for Boss-Extrude1 and Plane2. Now the only children of Boss-Extrude1 are Plane2 and Fillet1, and Plane2 has no children. Close the dialog to continue.



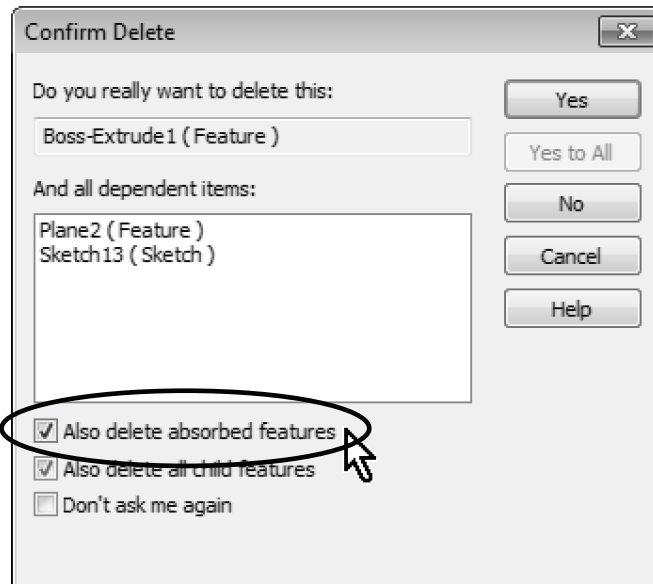
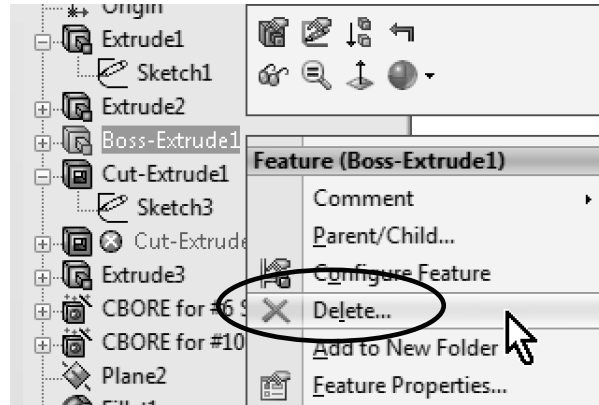
80. – Select Fillet1 and check **“What's Wrong?”** Note that the icon is different; in this case the error is a warning. The Fillet was built but it's missing one or more edges. Close the dialog and edit the Fillet feature.



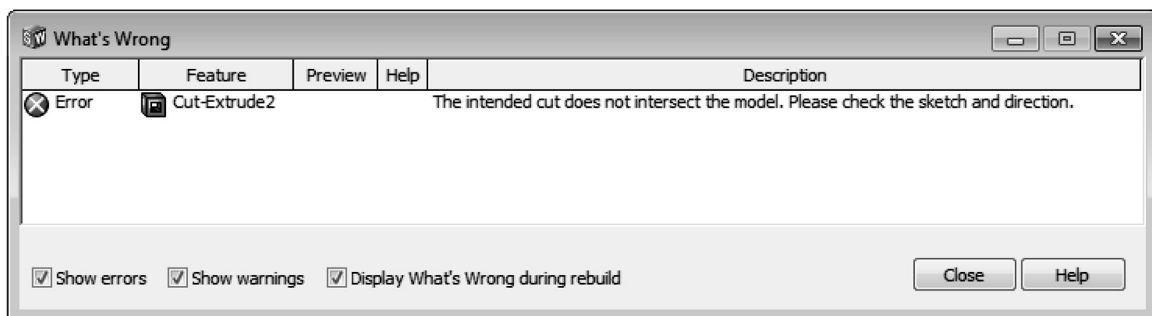
81. – The selection box is listing a missing edge in the same ‘dangling’ brown color, and the missing edge is shown with a faint red phantom line. Since we no longer need the fillet in that missing edge, delete it from the selection list and click **OK** to finish; the error will be cleared.



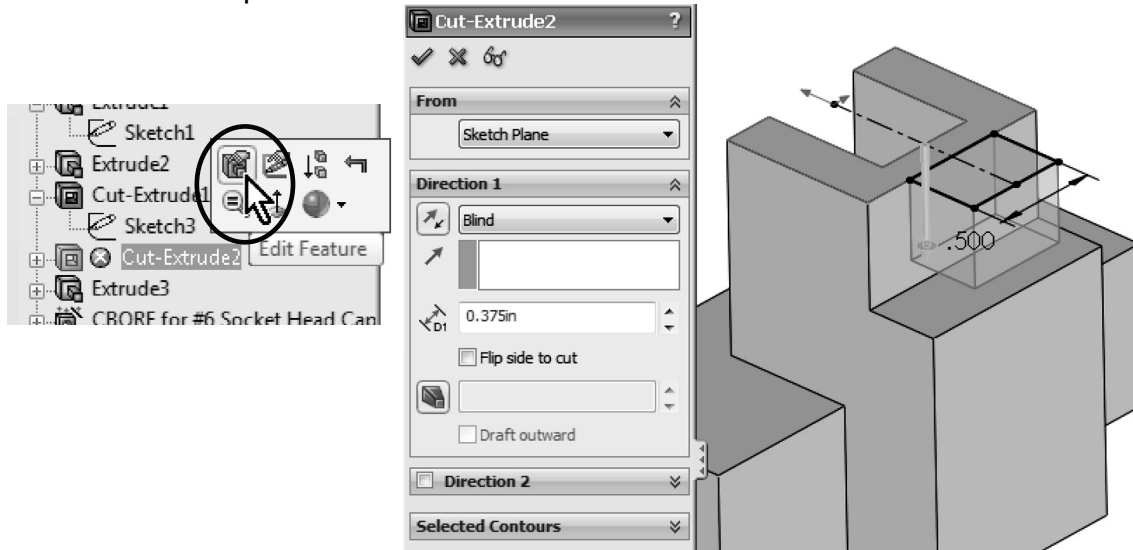
82. – After fixing Fillet1 we can delete the Boss-Extrude1 feature. Select it in the FeatureManager and delete it. This time the confirmation dialog only lists Plane2, which we don't want (or need) and it's OK to delete. Be sure to click on the option “**Also delete absorbed features**” to delete its sketch, too.



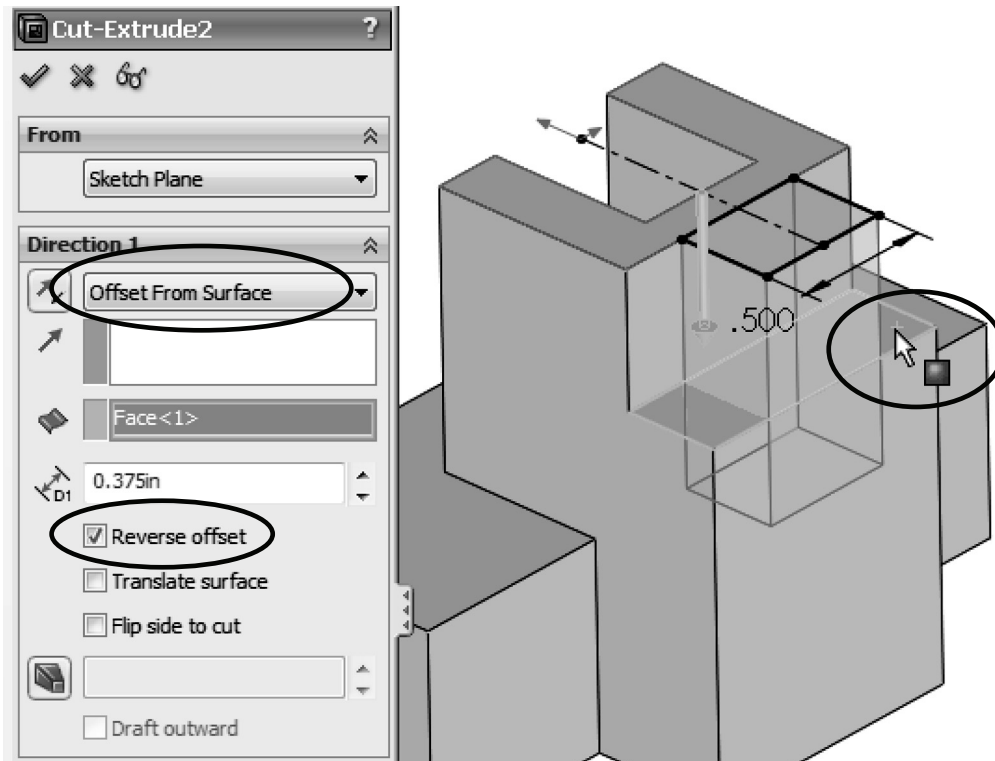
83. – Now we have one error left to fix. This error means that the cut is not cutting through the model.



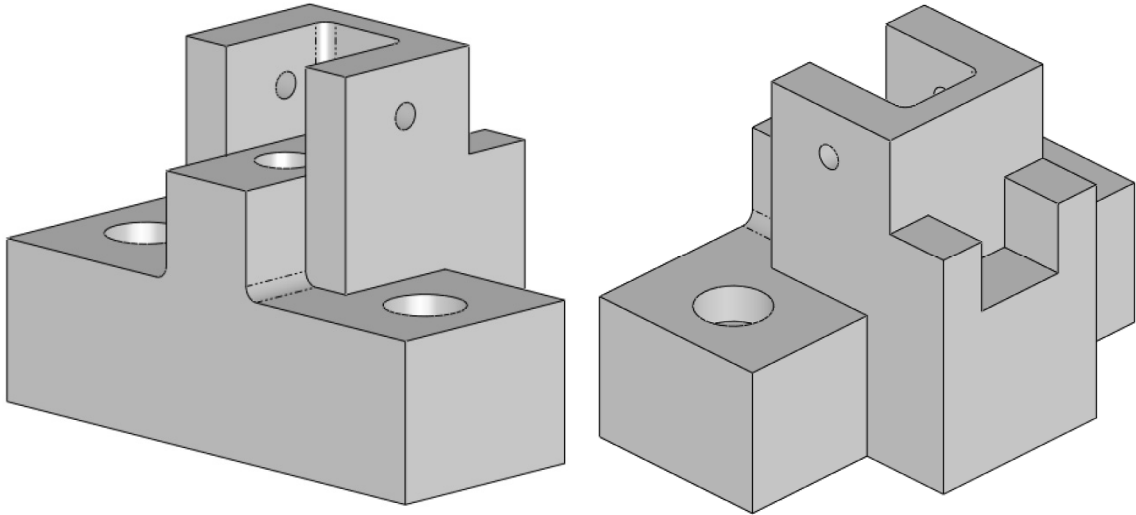
84. – Close the “**What's Wrong**” dialog and edit the Cut-Extrude2 feature. We can see the preview is not going deep enough and we want it to be 0.375” deep into the lower step.



85. – Change the cut’s end condition to “**Offset from Surface**” and select the lower step’s face as indicated. This end condition will make the cut the offset distance going to either side of the selected face; if needed, turn on the “**Reverse offset**” checkbox to cut into the part as shown. Click **OK** to finish.



The finished part fully repaired looks like this:



86. – When working in a sketch we can have different errors, so we decided to show them with a different part. Download the part *Sketch Relations.sldprt* from www.mechanicad.com/download.html. Just like the part before, we have a number of errors in this part's sketch. Close the “**What's Wrong**” dialog after opening the part and edit the sketch. We'll show the use of diagnostic tools to help us correct the errors.

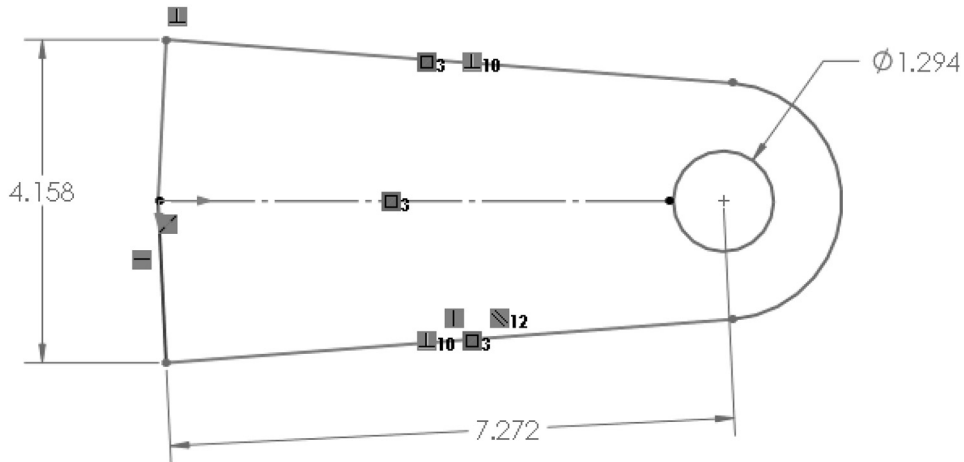
In a sketch we can have many types of geometric relations. What we are going to focus on is when relations are not solved correctly and generate warnings or errors.

An Under Defined or Fully Defined sketch can be used in a feature without a problem, the latter being the desired state. When we add conflicting relations that cannot be solved, a sketch's geometry can be in one of the following states:

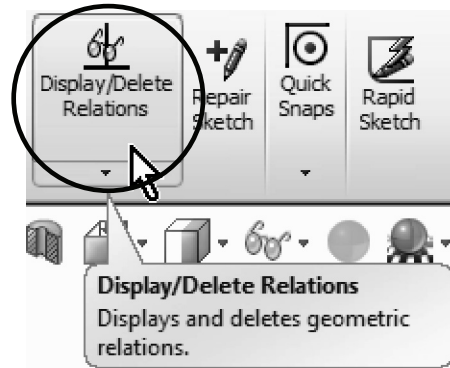
State	Color	Description
Over defined	Red	Conflicting relations cannot be satisfied.
Not Solved	Yellow	Relations cannot be solved; geometry cannot meet the required relations.
Dangling	Brown/Gold	Relations to geometry that no longer exists.
External	Can be under, fully or over defined, not solved or dangling.	Relations to geometry outside the sketch;
In Context *		Relations referencing other part's geometry added in an assembly.
Locked *	Can be any color	<i>'Frozen'</i> <i>In context</i> relations.
Broken *		<i>In context</i> relations that have been broken.

* Will be covered more in depth in the **Top Down Design** section later.

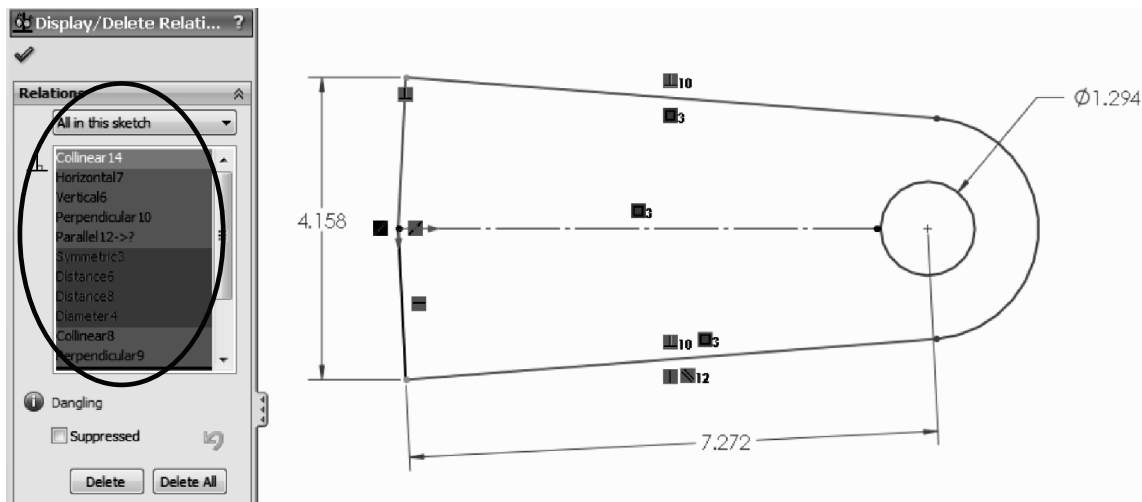
A sketch can become Over Defined, Not Solved or Dangling when geometry is deleted, modified, or when the user adds conflicting relations and/or dimensions.



87. – After editing the sketch we see red and yellow entities. There are two ways we can approach this: we can manually sort through the geometric relations and delete the conflicting ones, or we can use the **SketchXpert** tool to fix them. We'll show the manual mode first. Select the “**Display/Delete Relations**” icon from the Sketch toolbar, or the menu “**Tools, Relations, Display/Delete.**”

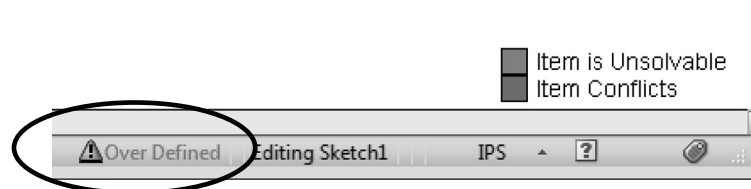
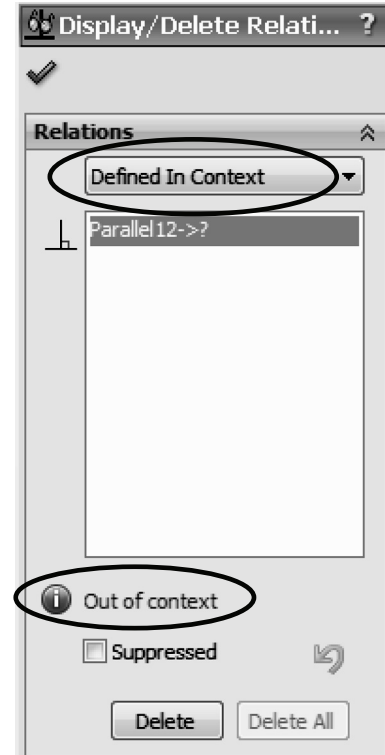


88. – In the PropertyManager we can see the list of existing relations in the sketch. We can see the relations highlighted with different colors, and their state listed under the list. At the top of the list we can filter the relations displayed by state, or if we select an entity, its relations will be listed.

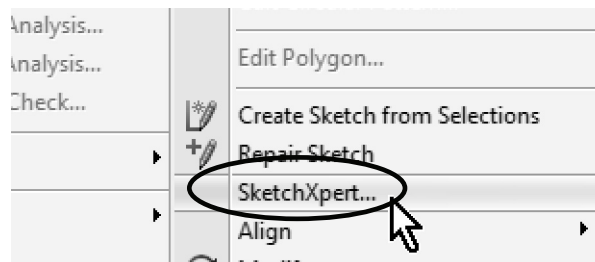


As the user can see, having a large number of conflicting relations can be difficult to sort manually. This is a good diagnostic option when we have problems with a few relations. On the other hand, this is a good tool to identify and/or delete relations, especially External, In context, Locked or Broken, if needed.

89. – Using this tool we'll remove the “**In Context**” relations. By deleting this relation, our part will no longer be referencing geometry outside the part (more on this in the Top Down Design section). Select the drop down filter menu and select “**Defined In Context.**” In our sketch we only have a Parallel relation and its status is “Out of Context.” Click the “Delete” button at the bottom or select the relation and press “Delete” on the keyboard. Click **OK** to close the “**Display/Delete Relations**” dialog.



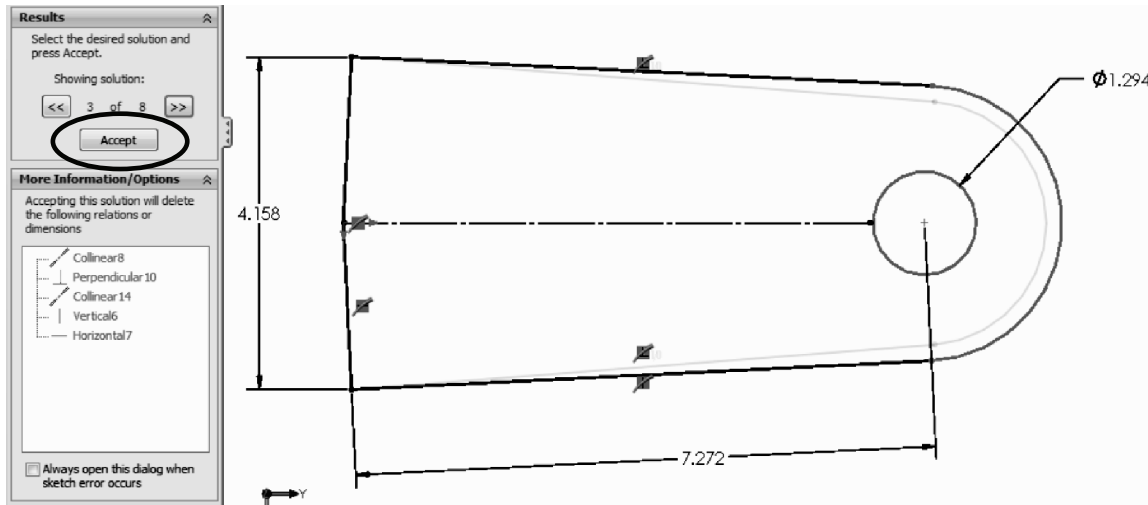
Our sketch still has errors, but we'll fix them using **SketchXpert**, which is a great tool to quickly review multiple possible solutions. We can load it by clicking in the “**Over Defined**” message in the status bar or the menu “**Tools, Sketch Tools, SketchXpert.**”



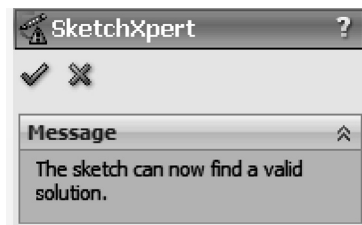
90. – In the **SketchXpert** dialog select “**Diagnose**” to automatically analyze the sketch and evaluate possible solutions.



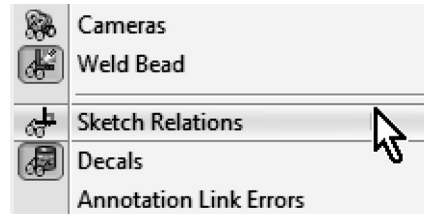
91. – **SketchXpert** quickly diagnoses the sketch and offers possible solutions. We can view each option by advancing in the “Results” box. In the “More Information/Options” box we are shown the relations and/or dimensions that would be deleted if we accept the solution displayed. Scroll until you see the following solution and click “**Accept**” when done.



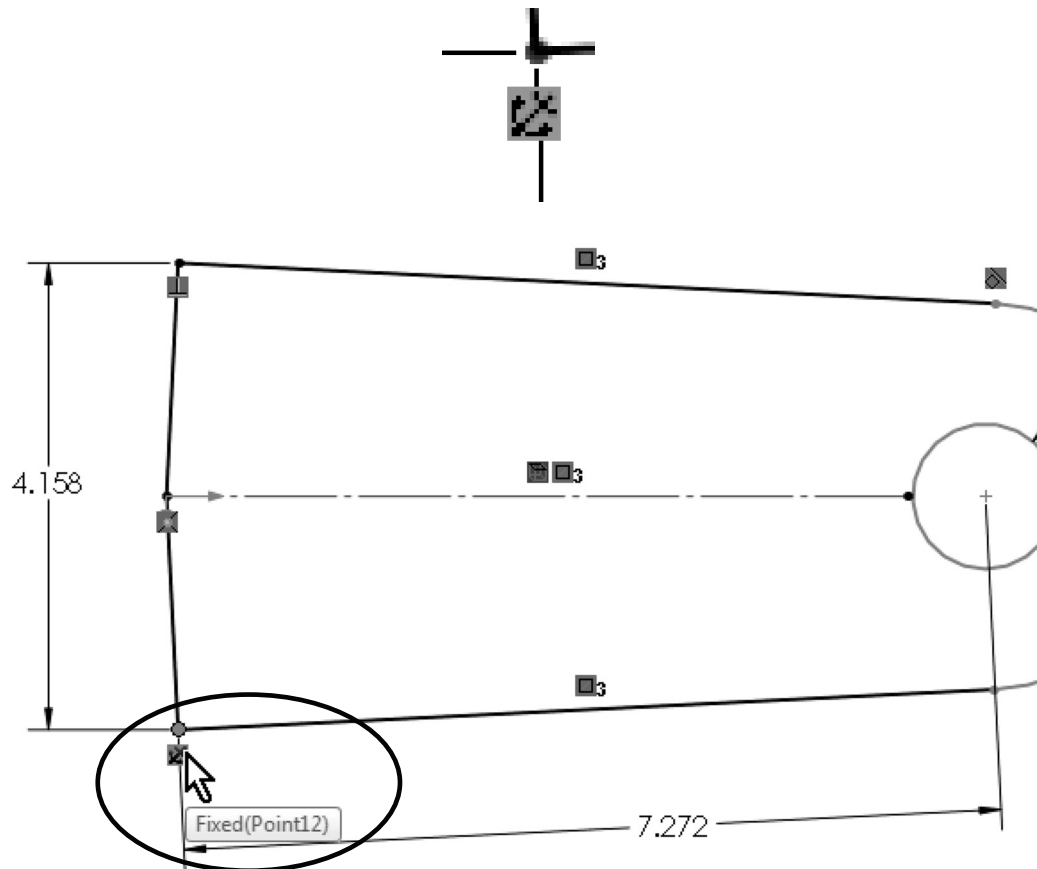
We see the message “The sketch can now find a valid solution” with a green background. Click **OK** to finish.



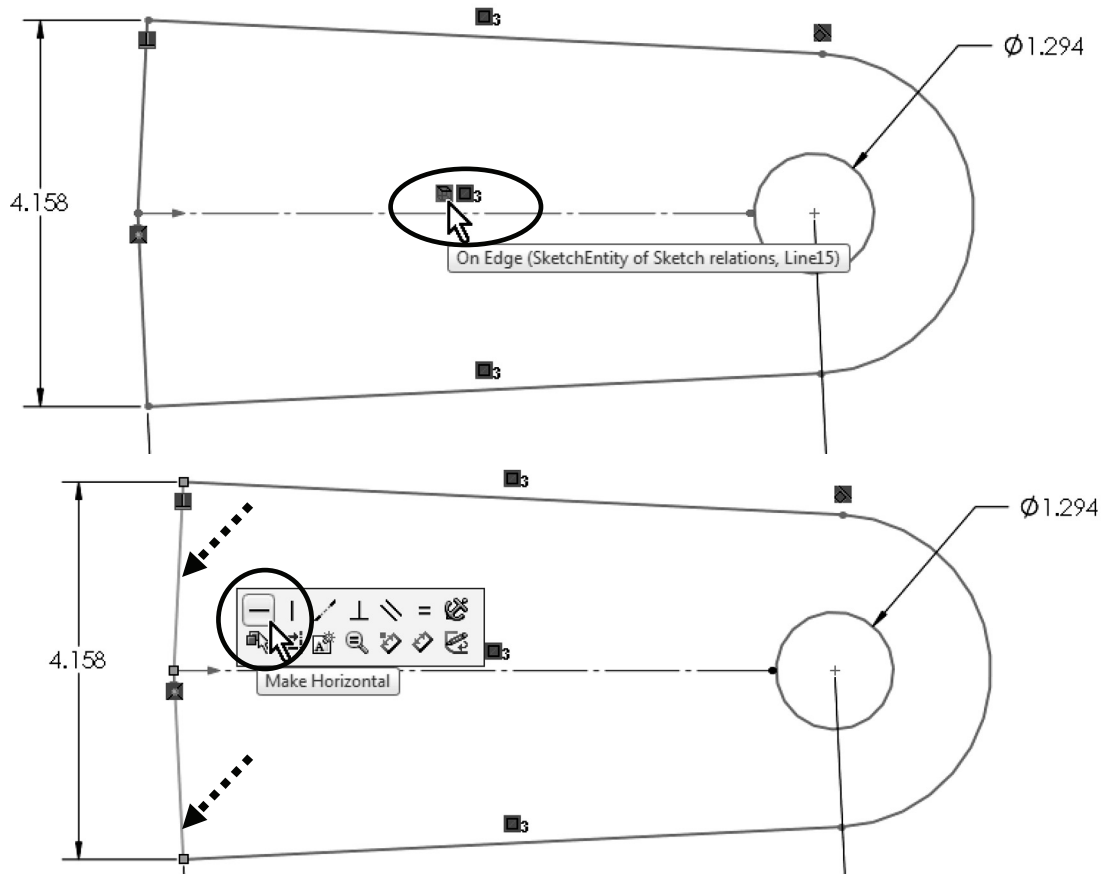
92. – We want to make the lines on the left to be collinear and horizontal (along the short origin arrow; the sketch is shown sideways to save space). What we need to do is to find what relations are keeping the lines fully defined, delete/edit them, and then make the lines horizontal. Turn on the display of sketch relations using the menu “**View, Sketch Relations**” if not already activated; this way we can see all the relations in every geometric element.



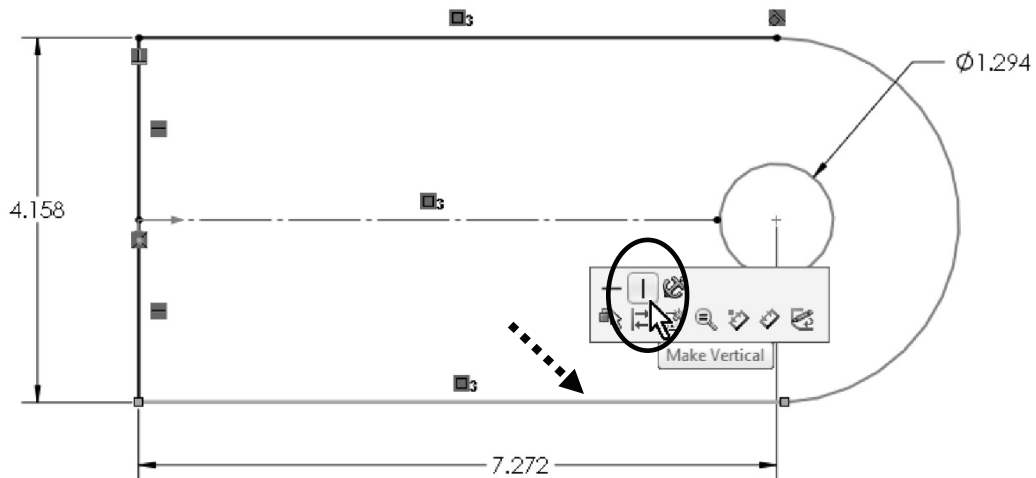
93. – We can see that there are only three relations in the two lines that we are interested in: a perpendicular, a coincident and a “**Fix**” relation. The Fix relation is the equivalent of artificially and arbitrarily constraining an element in space. Think of it as putting a nail in a geometric element and hammering it in. Read: brute force. Select this Fix relation and delete it.



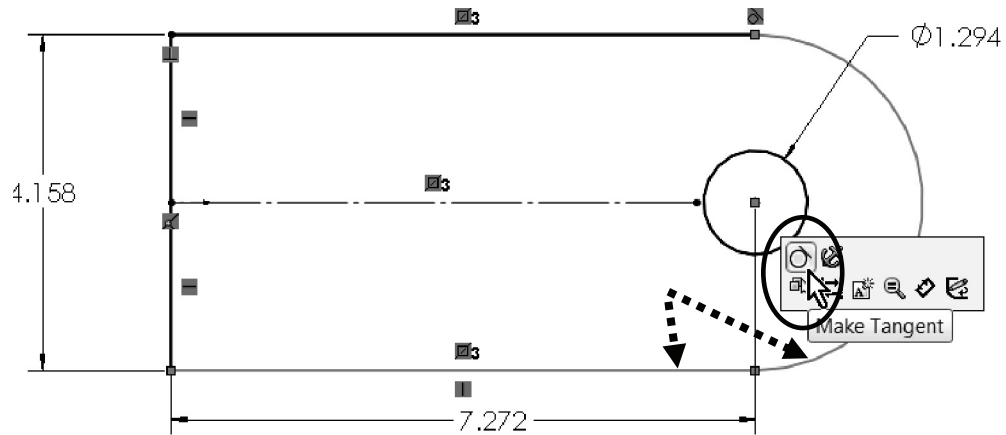
94. – If we pay attention, we'll see that we have a relation colored brown/gold in the centerline. This relation is “**On Edge**,” and is the type of relation created when we use the “**Convert Entities**” command in a sketch. The brown/gold color means that it is "dangling;" in other words, it lost the reference from where it was converted from, and we have to delete it. Delete this relation and make both lines on the left “**Horizontal**.”



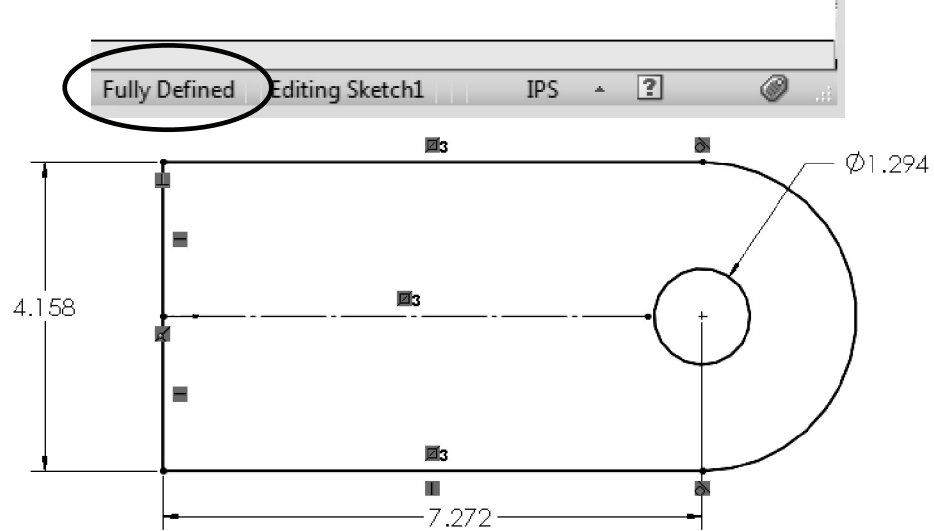
95. – Make one of the long lines “**Vertical**,” the other two lines will also become vertical as they have a “**Symmetric**” relation between them about the centerline.



96. – To complete the changes and fully define the sketch we need to add a “Tangent” relation between the lower long line and the arc. Note that the other side of the arc is already tangent to the top long line. If desired, click and drag the endpoint (blue) to see the effect of not having the tangent relation.



We can read in the status bar that our sketch is now “Fully Defined.”



97. – Now the part is complete. Exit the sketch to finish, save and close the file.

