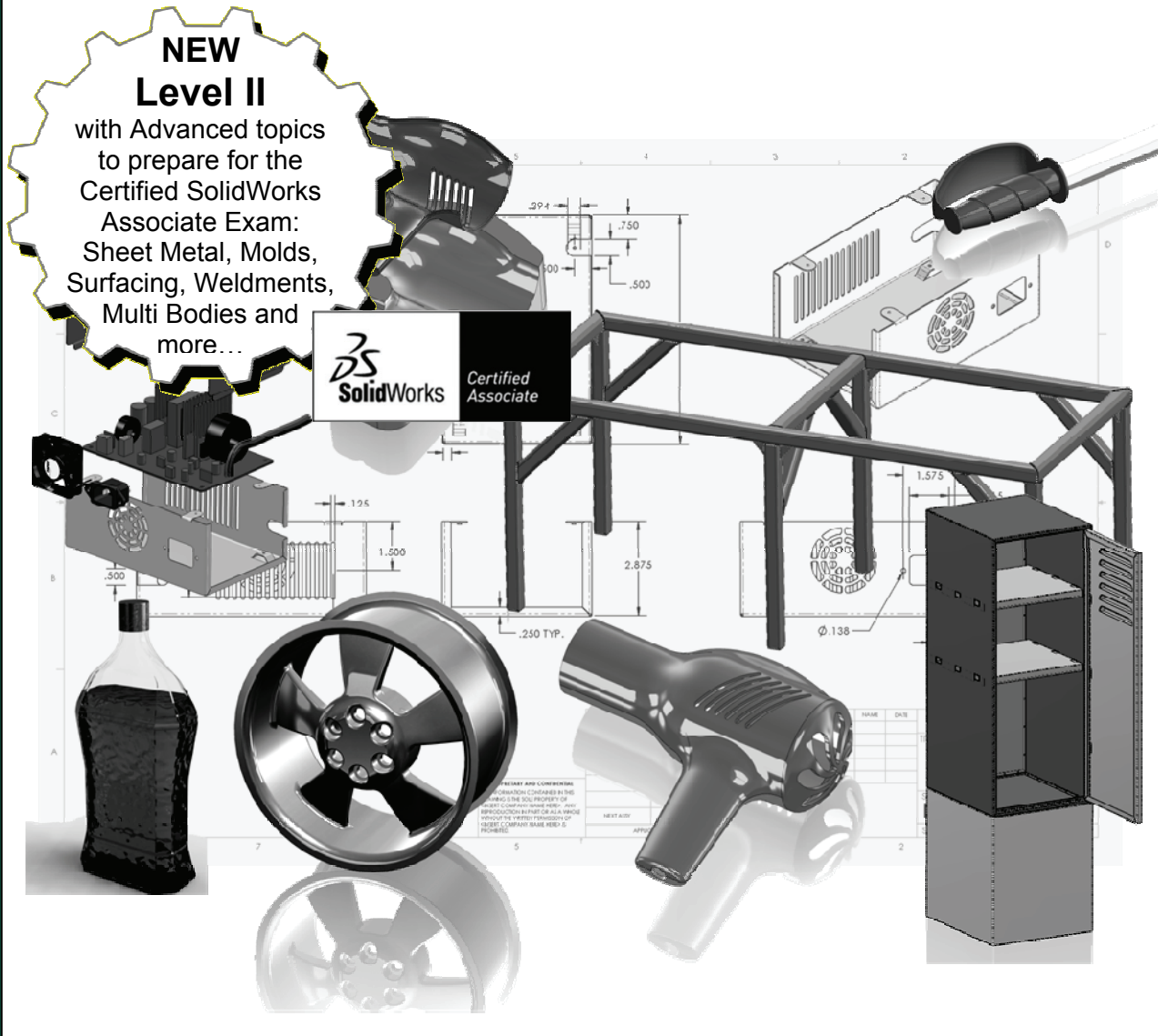


Beginner's Guide to SolidWorks 2011 – Level II



Alejandro Reyes, MSME
Certified SolidWorks Professional



SDC
PUBLICATIONS

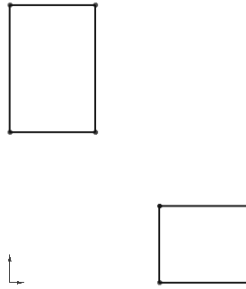
www.SDCpublications.com

Schroff Development Corporation

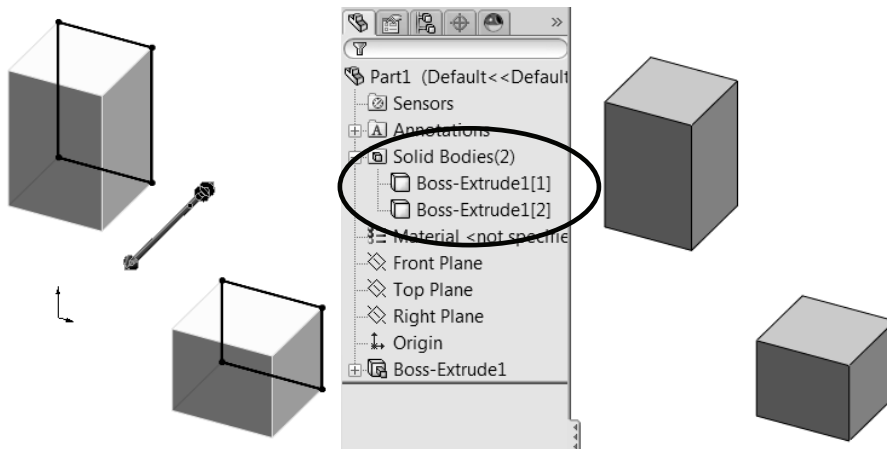
Multi Body Parts

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, that is purposely not merged to it, or by splitting an existing body. To show this, make a new part, and add a sketch to the Front plane as shown. We will not worry about dimensions here as we are just illustrating the concept of how multi bodies work.

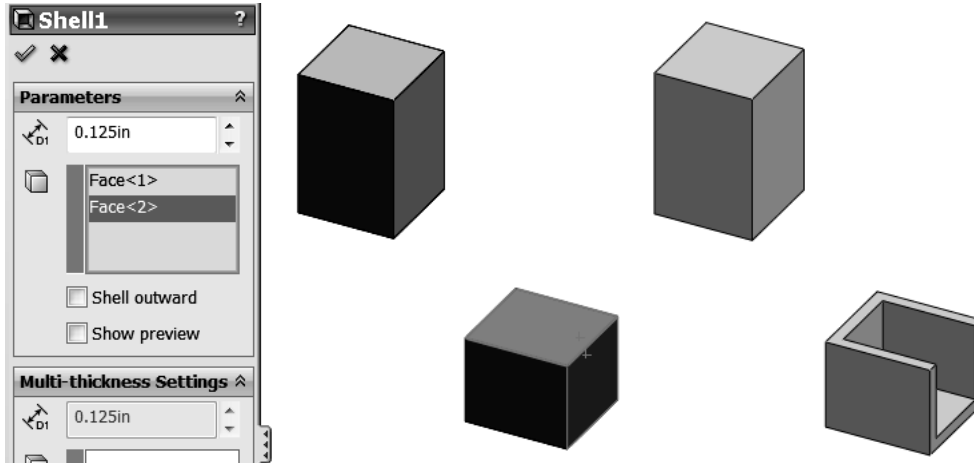


2. – Select the “Boss Extrude” command. Notice that we are not given a warning about having two separate bodies; it just works. Extrude any size that looks similar to the following image (dimensions are not important at this time) and click **OK** to finish. (Making two or more extruded/revolved/swept/lofted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

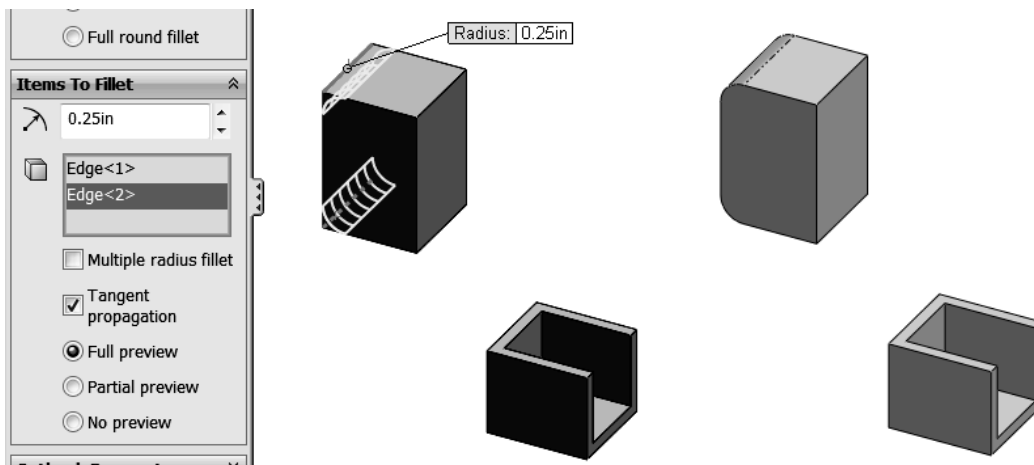



The first thing we notice is a new folder in the FeatureManager called “**Solid Bodies(2)**”. This folder is automatically added when SolidWorks detects multiple disjointed bodies in a part and lists the number of bodies found in the part (in this case, 2). If we expand the folder, we can see the two 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 as 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.

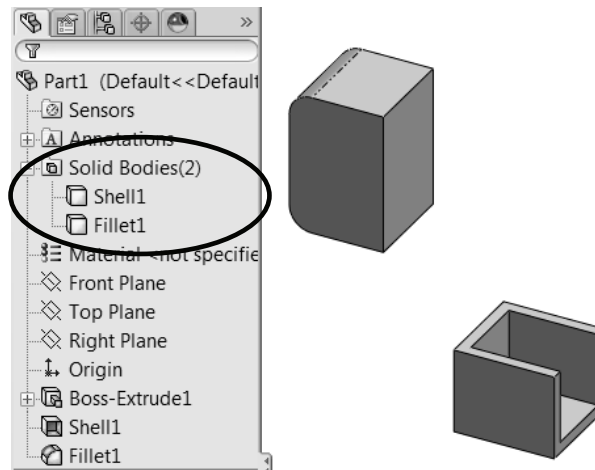
3. – The next step is to add a new feature. When working in a multi body part we can make ‘local’ operations; for example, a shell feature affecting only one body. Select the “Shell” command from the Features toolbar, and shell the bottom body as indicated.



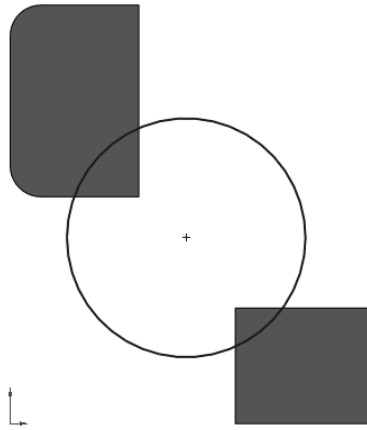
4. – Select the **Fillet** command and round two corners to the top body as shown.



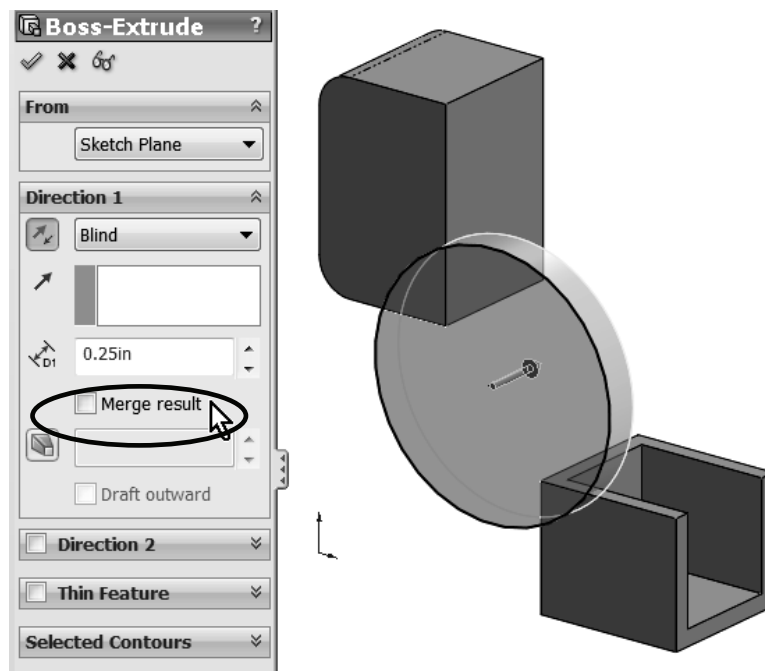
 To help the reader understand the concept of local operations better, “local” means that we can add applied features (features that don't require a sketch like Shell, Fillet, Chamfer, Draft, etc.) to each body separately. Notice the name of a body changes to the last feature applied to it.



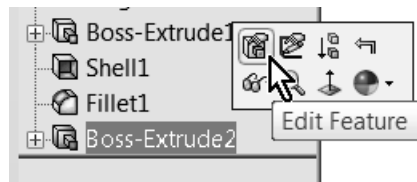
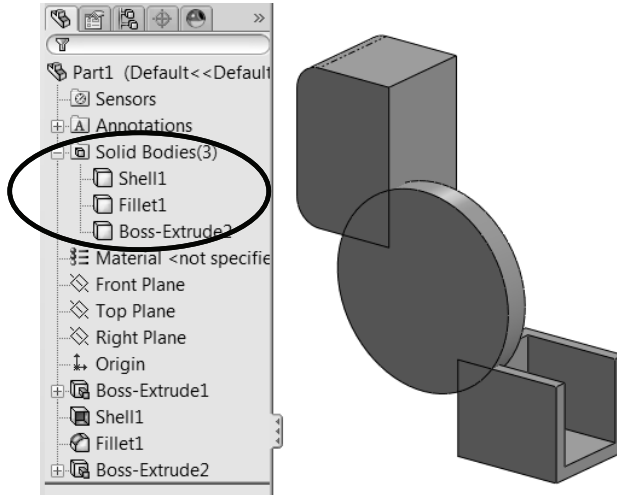
5. – When working with multi body parts, adding more features automatically selects existing bodies to modify; this is the default behavior. Optionally, we can select which bodies to “merge” (or fuse with) to make a single body, or select which bodies to cut. Here we'll make a new boss and explore the option to merge two existing bodies. Select the front face of a body and make the following sketch.

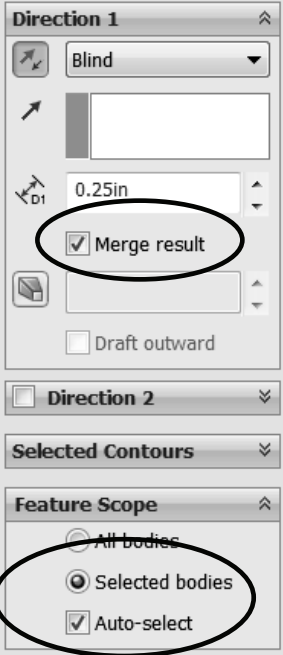
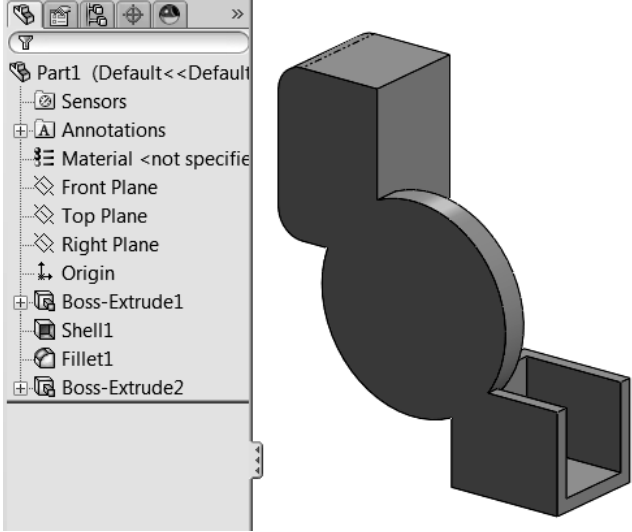


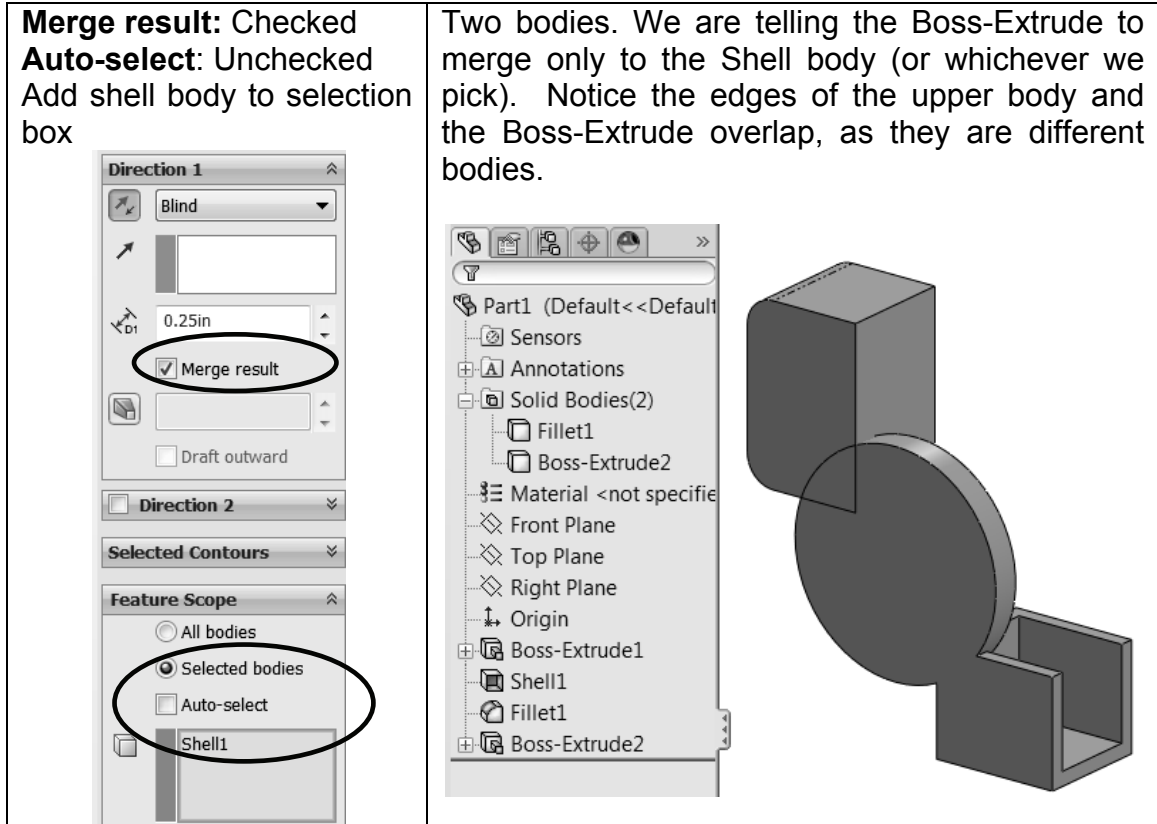
6. – Extrude the sketch *into* the existing bodies. Notice the “**Merge result**” option in the Extrude command. It's always been there (except when there are no existing bodies), and by default is always checked. A new option at the bottom of the Extrude command is “**Feature Scope**”. This is where we can select which bodies to affect, either “All bodies” or “Selected bodies” and either is automatically or manually selected. By default Feature Scope 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. Uncheck “**Merge result**” and click **OK** to finish (when unchecked, “**Feature Scope**” is automatically removed.)



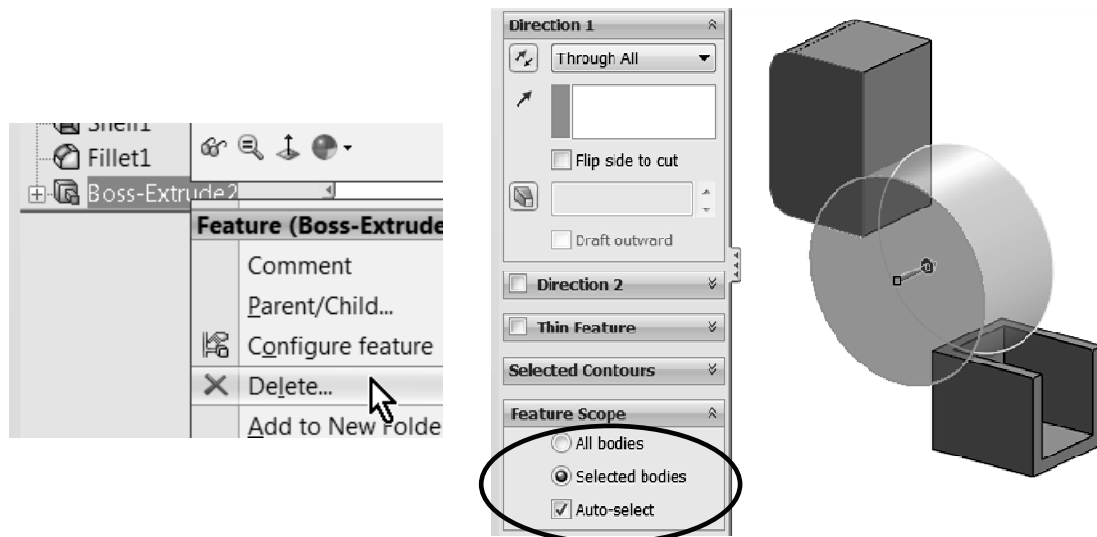
7. – The result is three bodies in our part. See how the different bodies' edges intersect each other. If we had left the “**Merge result**” option checked, we would not see these edges overlapping as they would have merged into a single body. Edit the definition of the last extrusion to explore the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



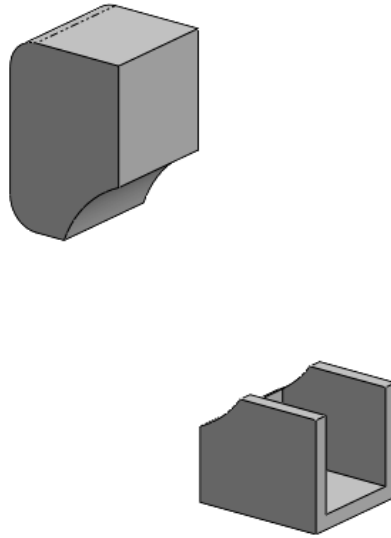
Combination:	Result:
<p>Merge Result: Checked Feature Scope: Auto-Select</p> 	<p>Single body. By touching both existing bodies “Auto-select” merges them, fusing them into a single solid body. The “Solid Bodies” folder is no longer visible.</p> 



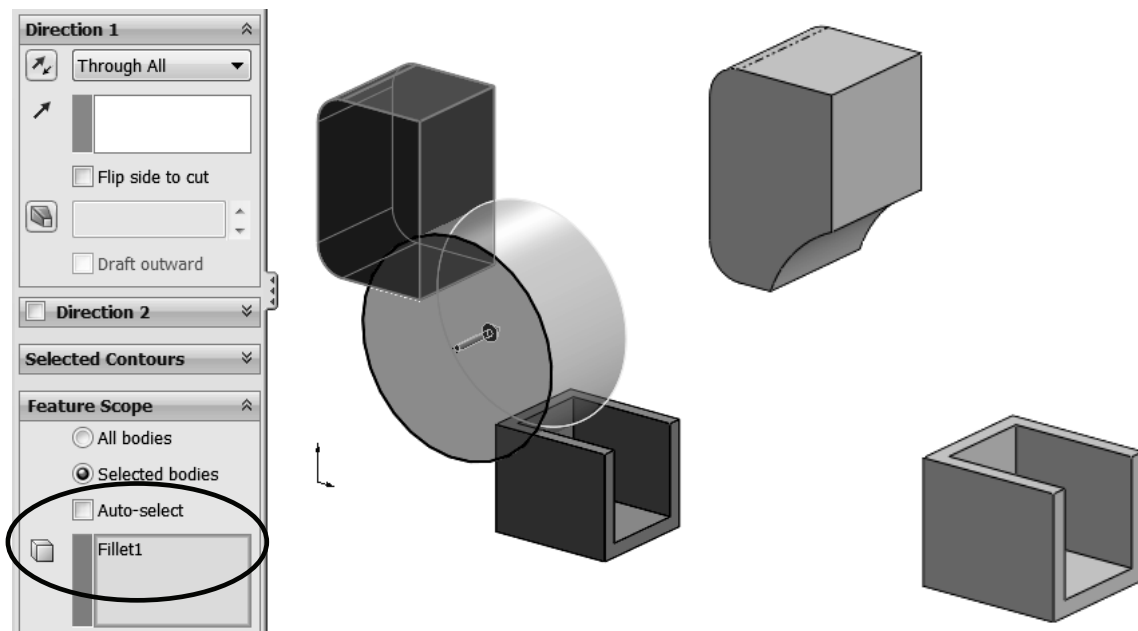
8. –The “**Merge result**” option works the same way with any feature that adds material to the part, including revolved boss, sweep, loft, etc. Now we’ll see how it works when we remove material. Delete the Boss-Extrude feature and keep the sketch. Select the sketch and click in Cut-Extrude using the “Through All” option. In this case the only difference from a boss extrude is that we only have the “**Feature Scope**” option at the bottom with the same selection options: “All bodies” or “Selected Bodies,” and with the “Auto-select” or manually selected bodies. 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.



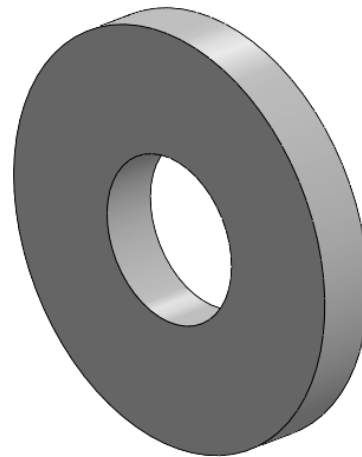
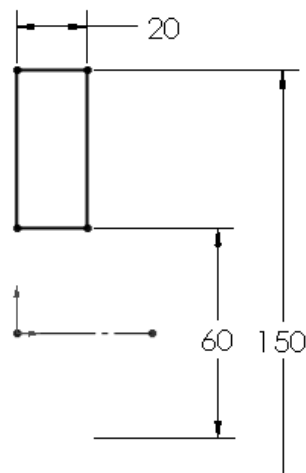
10. – Edit the Cut-Extrude definition, turn off the “Auto-select” option in the “Feature Scope,” and select only the top body. Click **OK** to finish. Now we are only modifying the top solid, even if the Cut-Extrude overlaps the lower body.



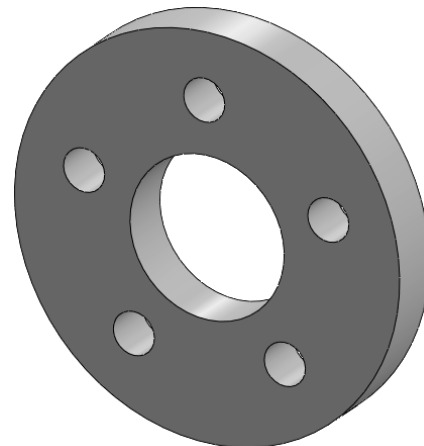
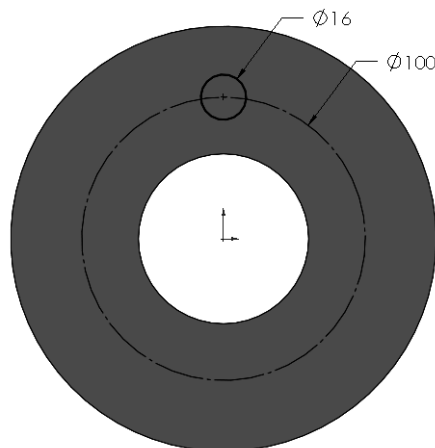
And just as with the features that add material, this technique works the same way with all the features that remove material including revolved cut, sweep cut, loft cut, etc.

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 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 will be the hub or mounting pad. Draw the following sketch in the Right plane and make a 360° Boss-Revolve. Tire dimensions are usually in millimeters. Since the sketch dimensions are given in millimeters, be sure to change your part’s dimensions accordingly (**Tools, Options, Document Options, Units**). Pay attention to the diameter dimensions (doubled about the horizontal centerline).

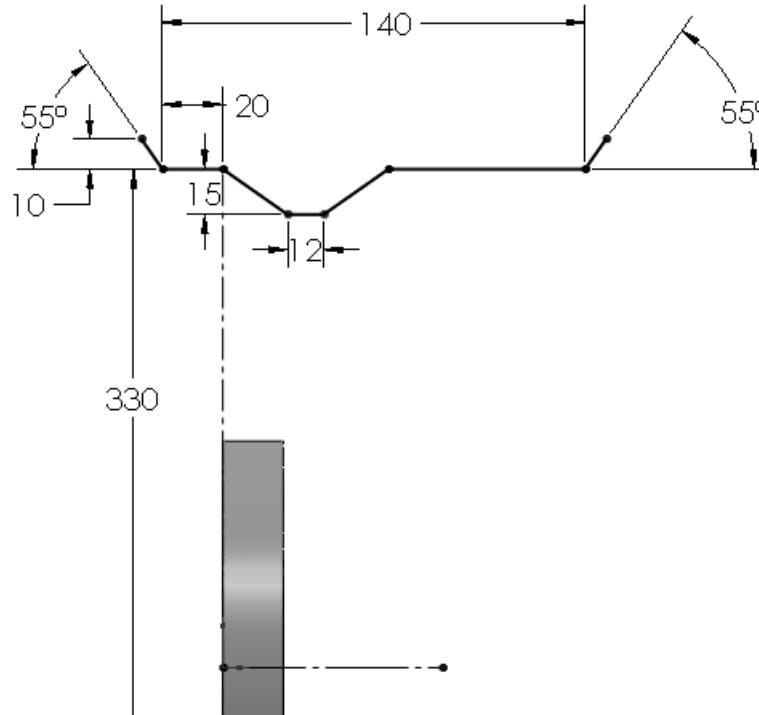


12. – Make a pattern of 5 holes to mount the wheel to the car.

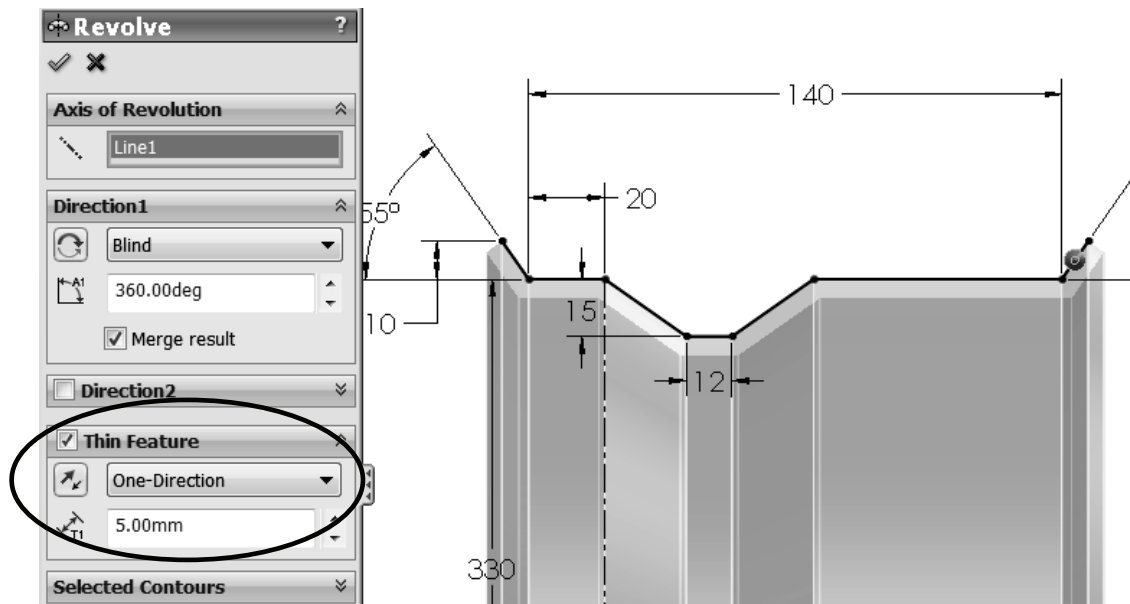


13. – Now we'll make the wheel's rim that matches the tire. Draw the following sketch also in the Right plane. (The 330 mm diameter dimension is doubled about the horizontal centerline as diameter.)

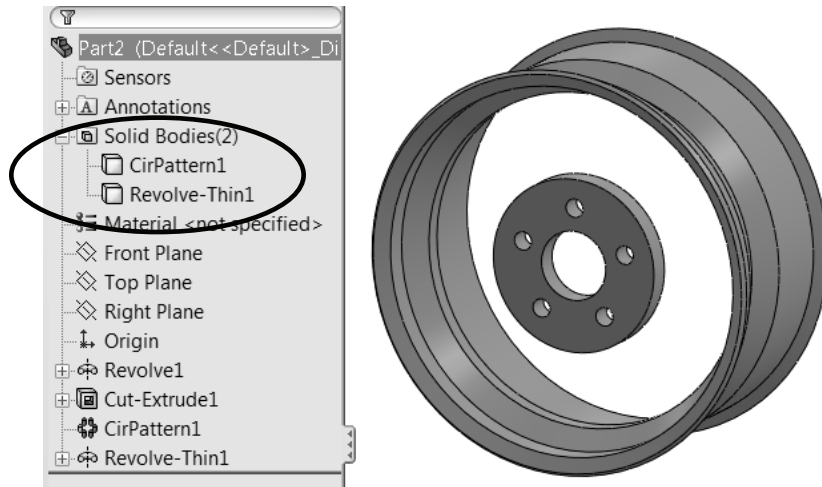
TIP: Make the two short lines on the sides equal, and the two lines connected to the 12mm horizontal line also equal.



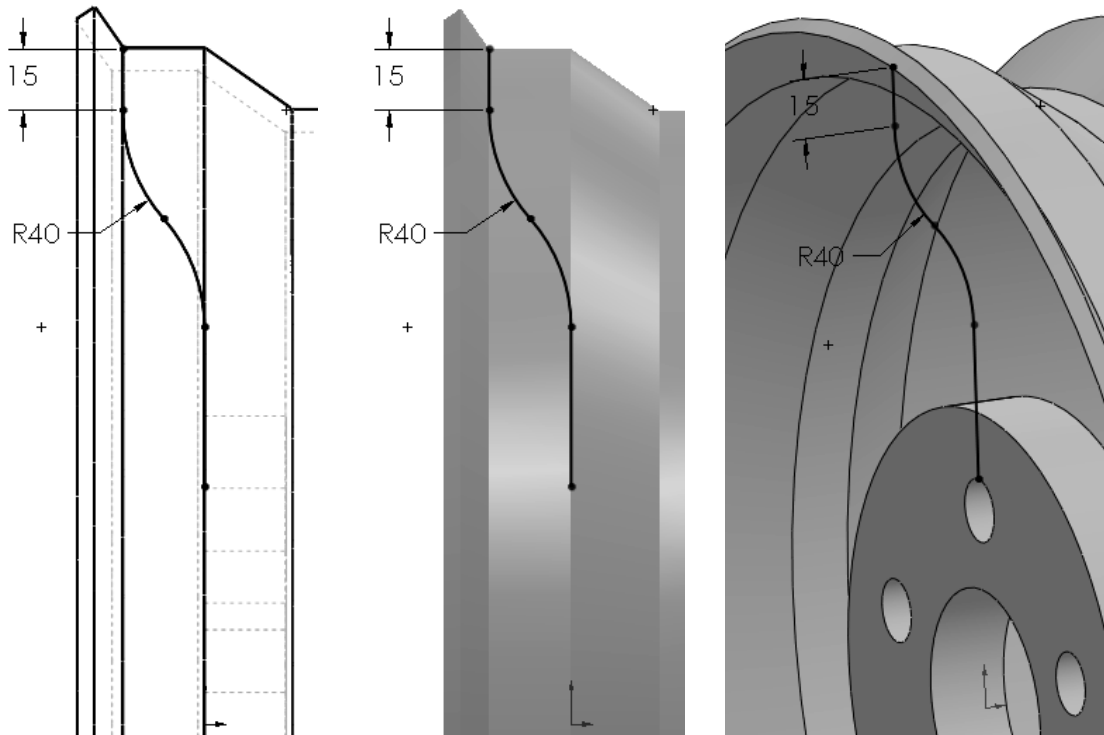
14. – After finishing the sketch, make a Revolved-Boss. Since this is an open sketch we will be warned about closing it if we don't want a thin feature. Select NO when asked. Make a thin revolved feature 5mm thick going inside.



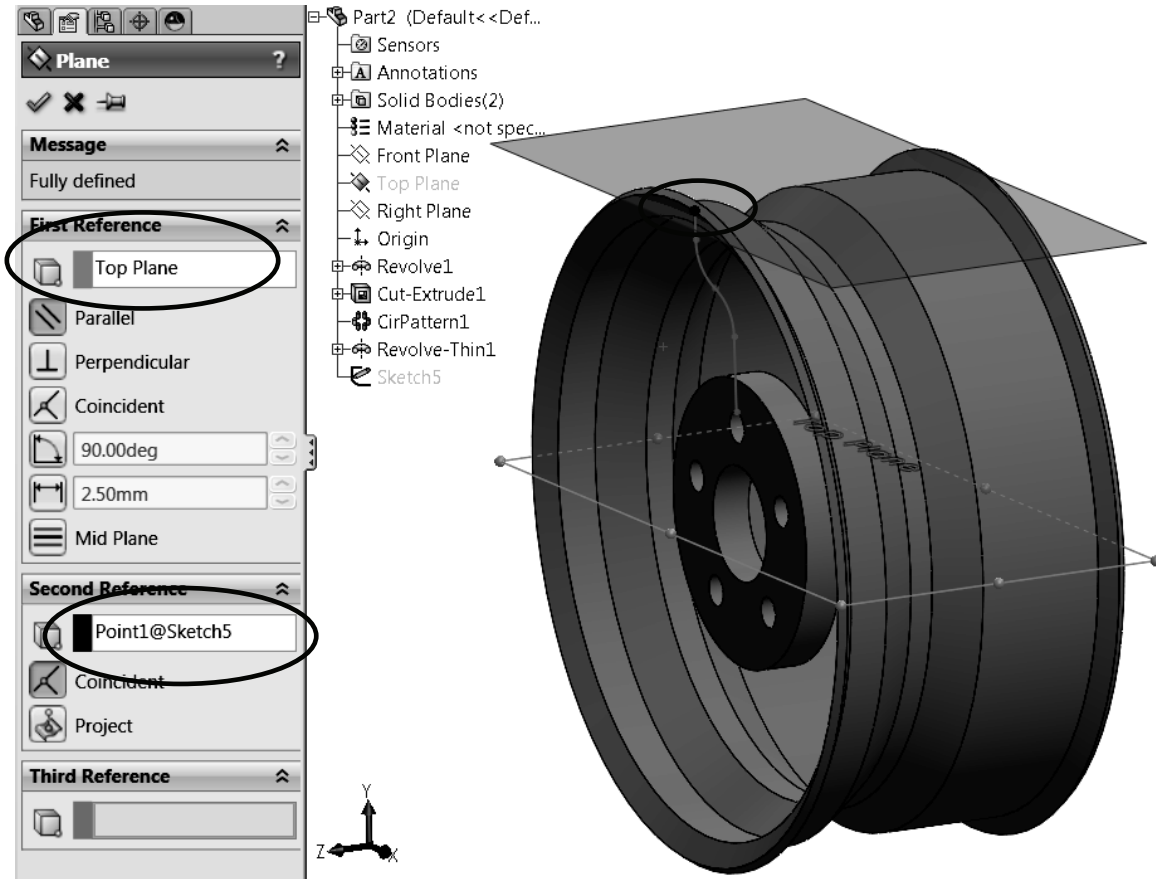
After finishing the revolved feature, we can see that now we have two bodies.



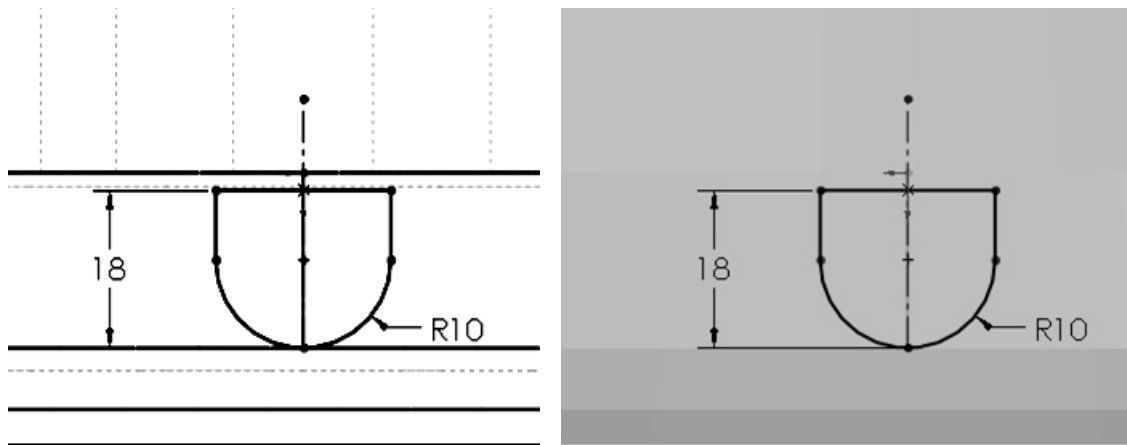
15. – Now we need to make the spoke in the wheel; this is the part where we can get creative. We'll assume our limitations are defined by the hub and the actual rim where the tire mounts. We'll design the first spoke and will not merge it to any other body; we'll have to make a couple of local operations on the spoke and show how to make a body pattern later on. 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 arcs tangent to both lines and 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.

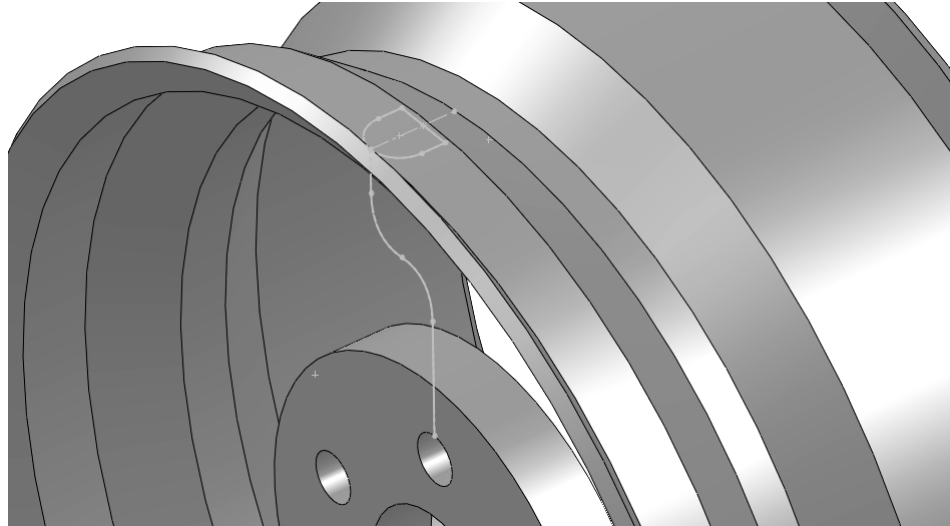


16. – Exit the sketch and make a new plane parallel to the Top plane coincident to the top endpoint of the previous sketch on which to draw the profile.

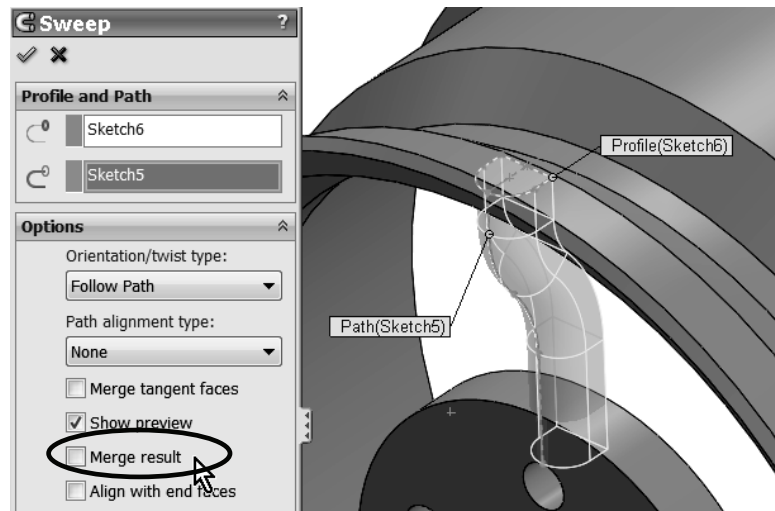


17. – Switch to a Top view, add a new sketch to the new plane and draw the following sketch. This sketch will be the Profile for the sweep. The arc is tangent to the edge where the path starts. Exit the sketch when done.

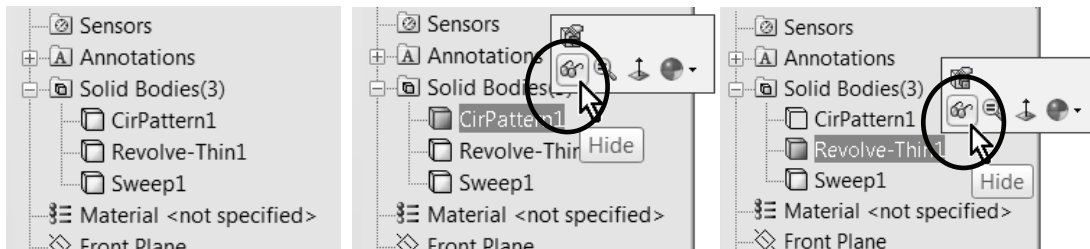




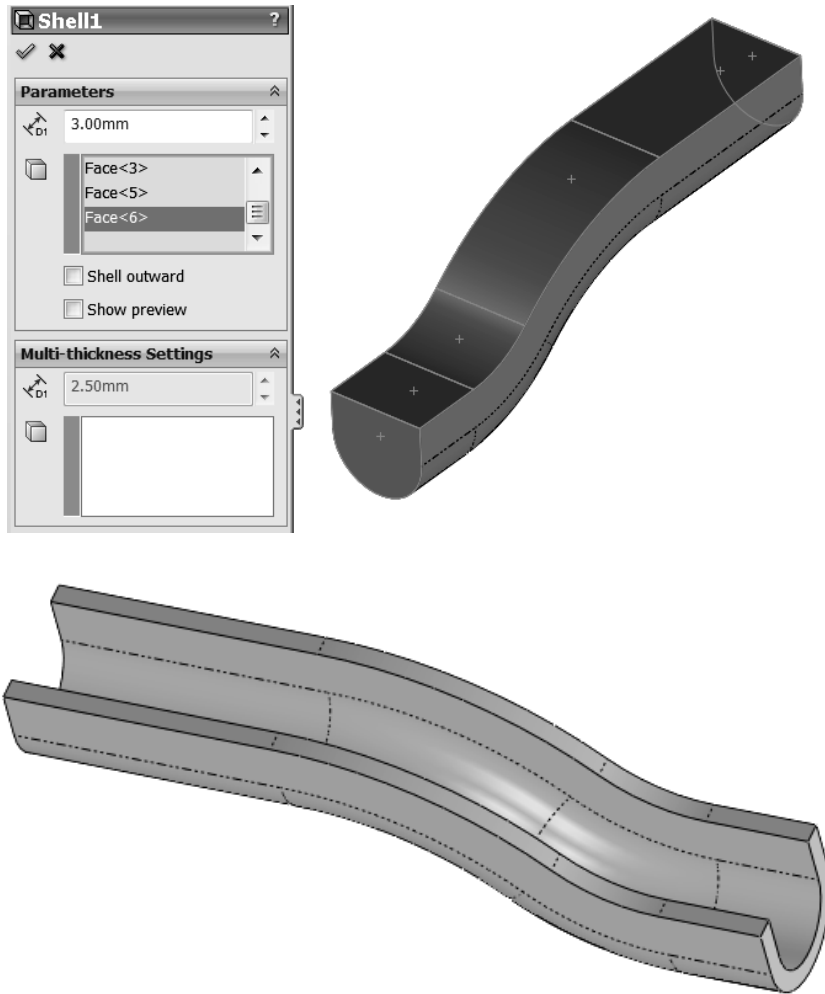
18. – Select the **Sweep** command; pick the second sketch for the profile and the first one as the path. In the Options, uncheck “**Merge Result**” as we still need to make a couple more operations before merging all bodies. Click **OK** to finish.



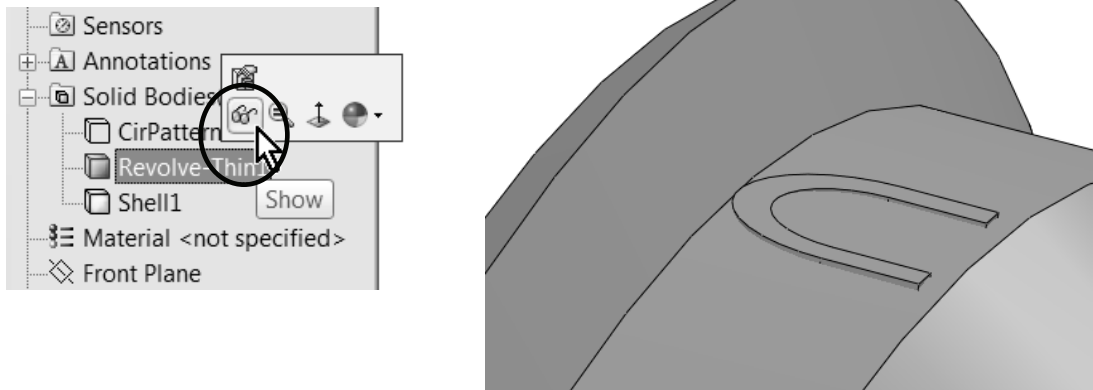
19. – Now we have three bodies. To continue, we’ll hide the first two and only leave the Sweep body visible. Select each of the other two bodies and hide them.



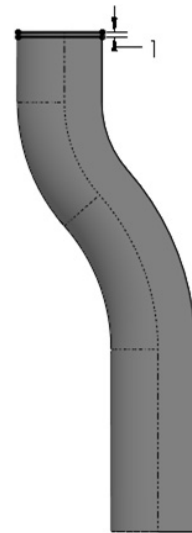
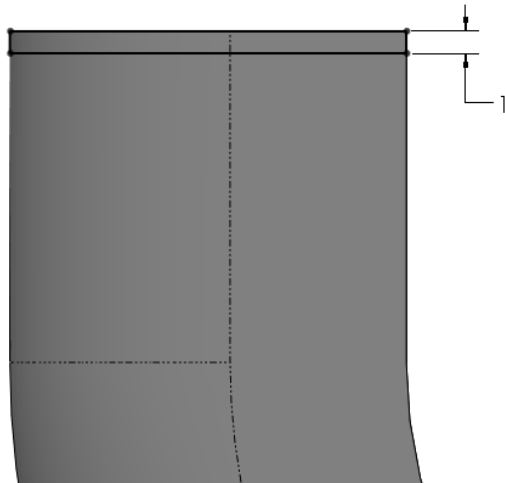
20. – After we are left with only the Sweep body, select the **Shell** command and remove the ends and back of the body. Make the shell 3mm thick.



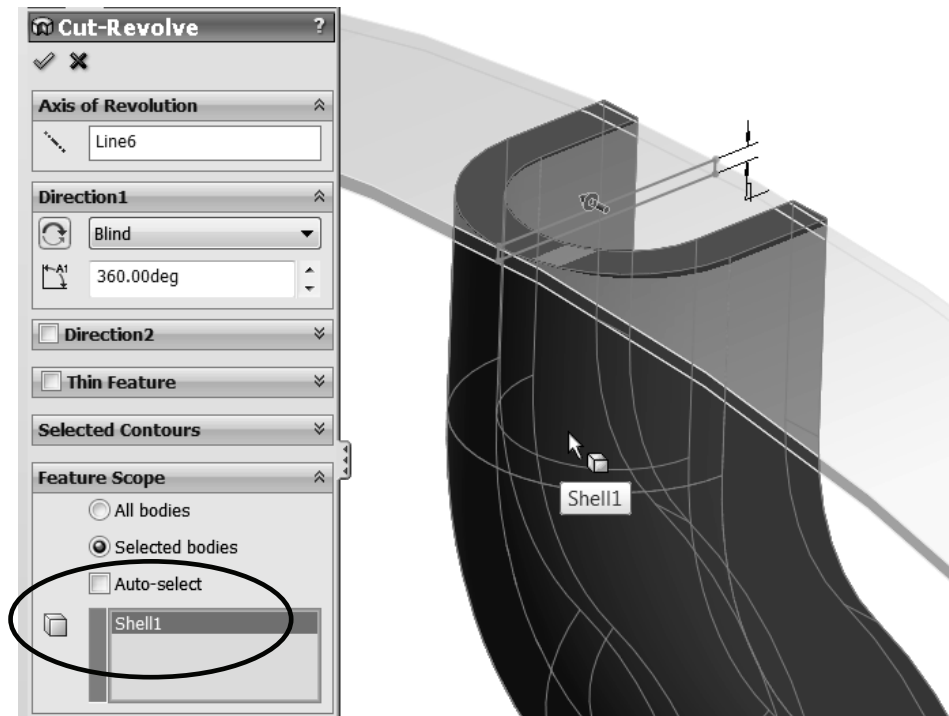
21. – We need to make a cylindrical cut to the top; otherwise we'll have a part of the Sweep coming through the top. Show the Revolve-Thin body to see it.



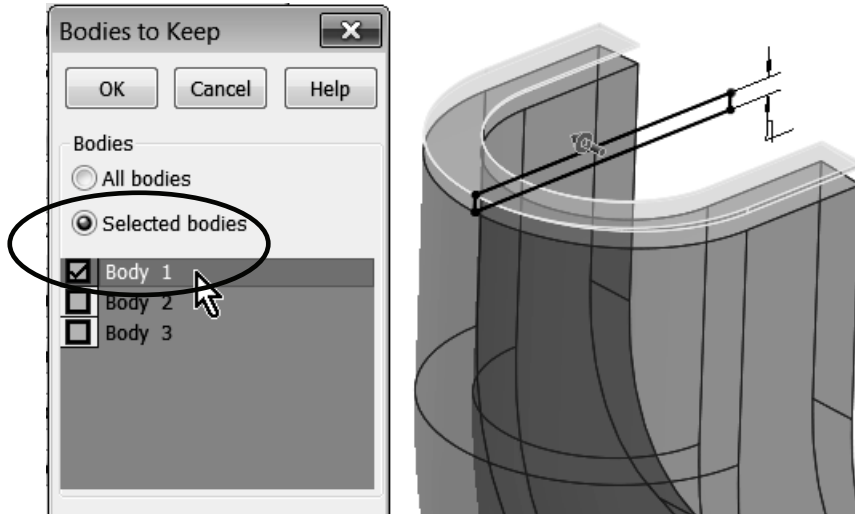
22. – Hide the Revolve-Thin feature again and switch to a Right view. Add a new sketch in the Right plane as shown. We are making the cut just under the top to make sure the Sweep body (now named Shell1) will not go through the top. Add a horizontal centerline at the origin to make the Revolved Cut about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



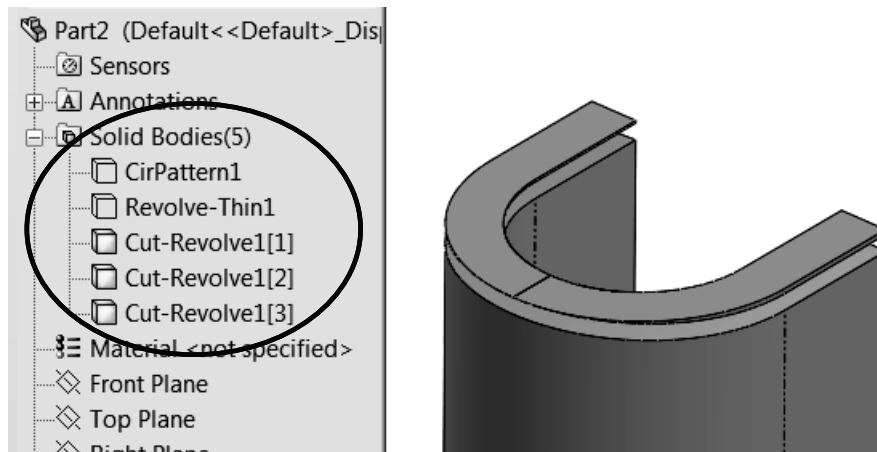
23. – Select the **Revolved Cut** command. What we want to do is to cut only the spoke body, and nothing else. Under “**Feature Scope**” uncheck “Auto-select” and select the spoke body only.

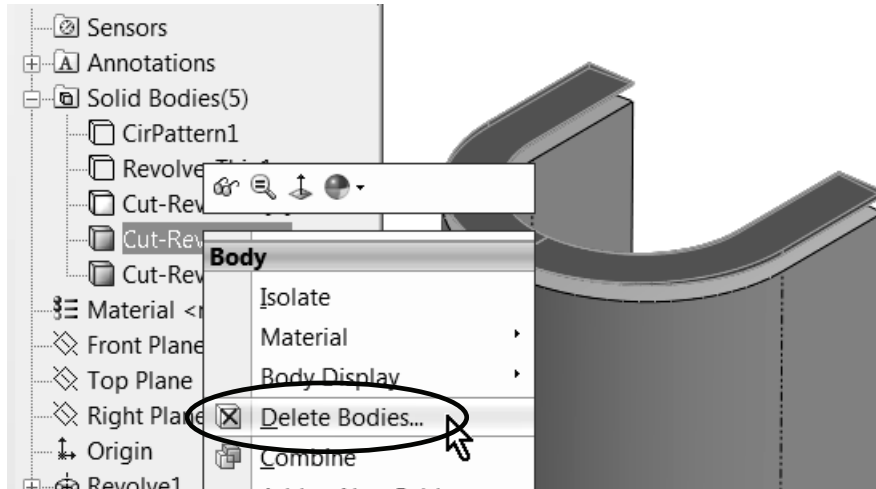


24. – After clicking **OK** we get a new dialog box asking us which bodies we want to keep, since the cut we are making divides the spoke in three pieces. Pick the **“Selected bodies”** option, and check the box for the spoke’s largest piece which we are interested in keeping. Click **OK** to complete. In this case we still have our three original bodies and discard the two smaller ones.



25. – If we had opted to keep all three bodies after the Revolved Cut, we would have five bodies in our part. To eliminate the two extra bodies we don’t want, we have to delete them. Select the two bodies, right mouse click and select **“Delete Bodies”**. Make sure the correct bodies are selected and click **OK** to finish.



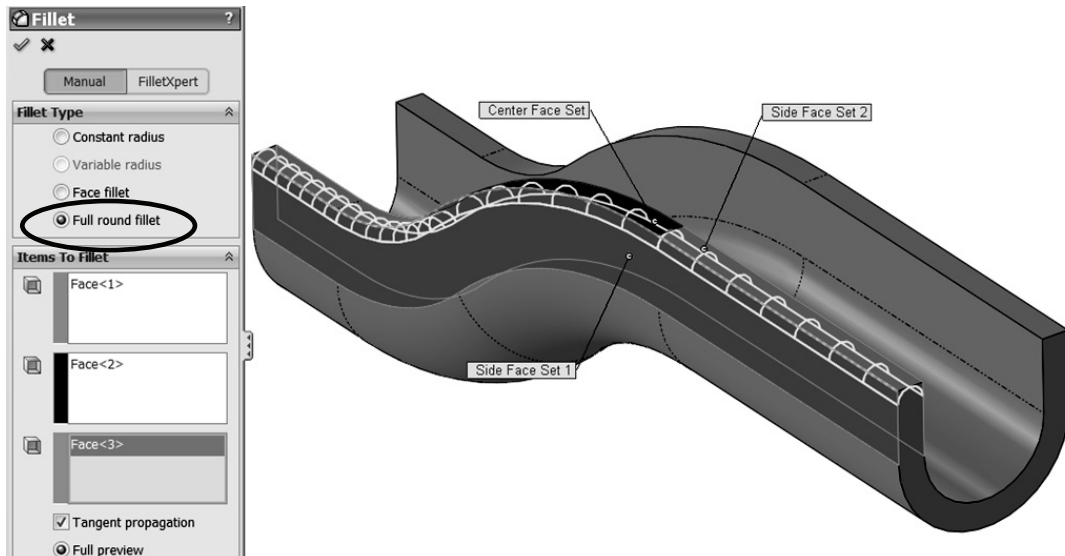


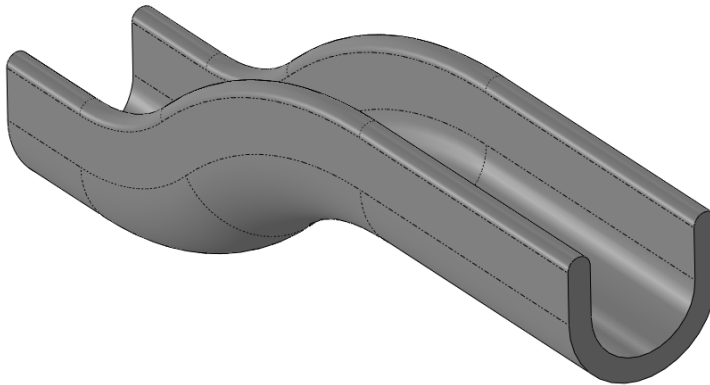
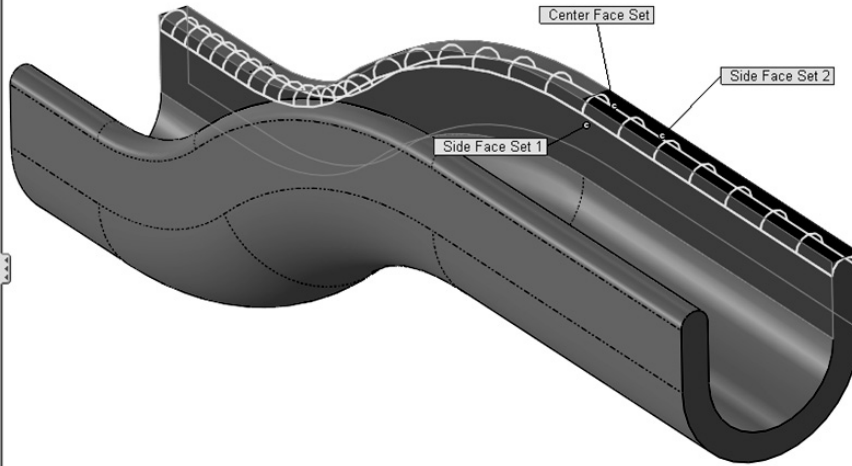
A new feature “**Body-Delete**” is added to the FeatureManager.



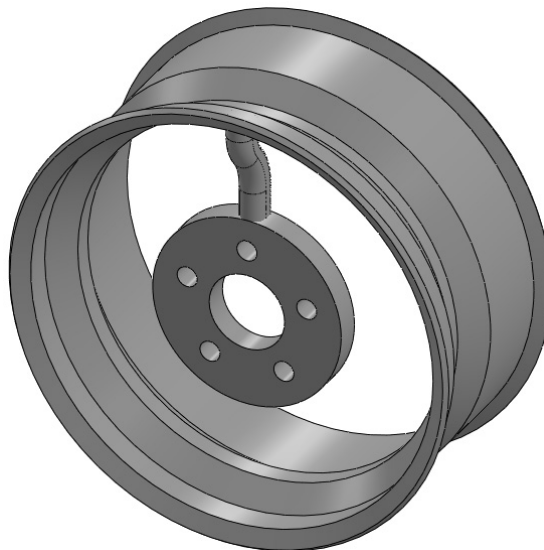
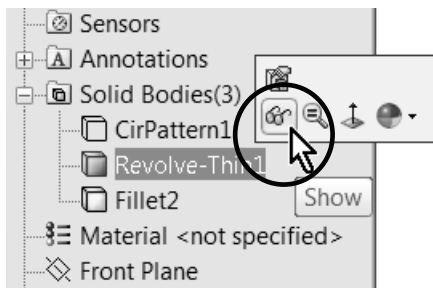
For our example we'll use the approach to select the bodies to keep at the time of making the Cut-Revolve instead of deleting the remaining bodies with an additional operation.

26. – Now add a Full Round Fillet to each of the sides 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.

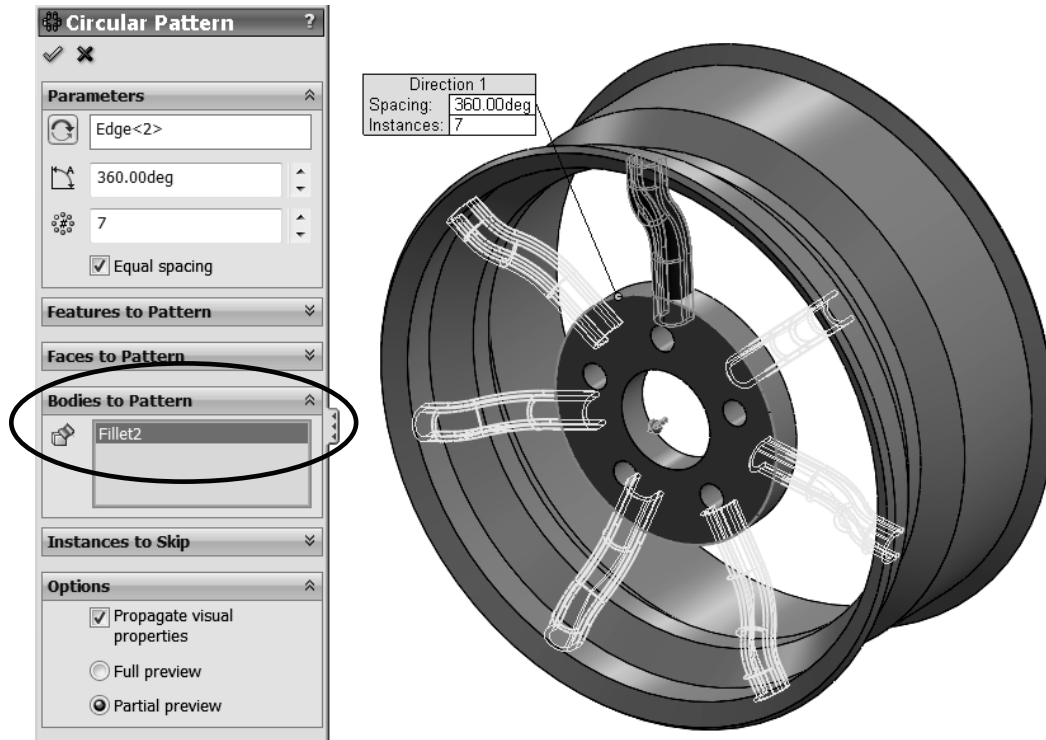




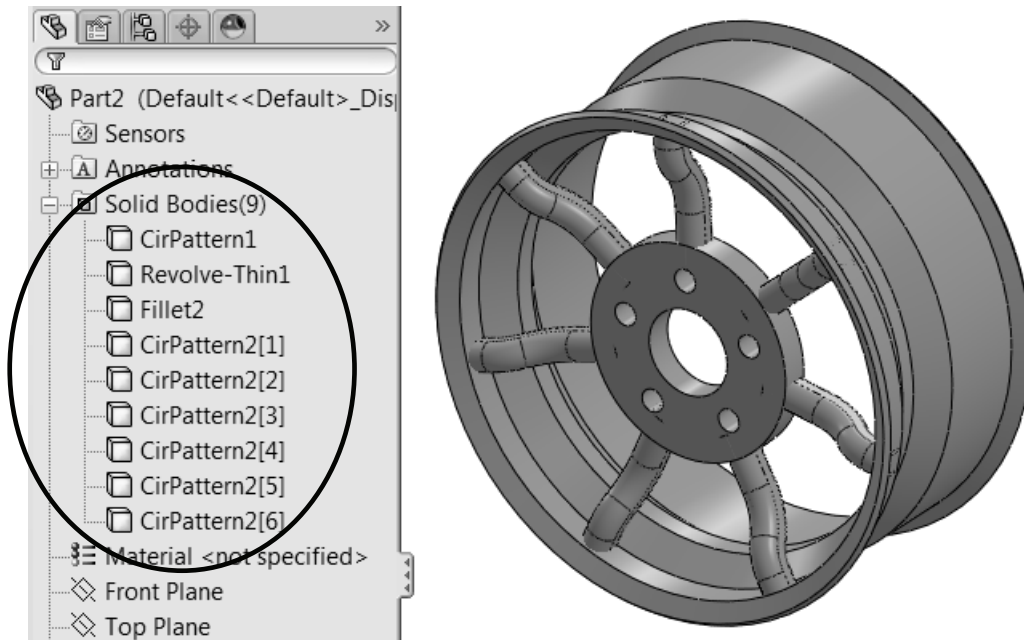
27. – We are now ready to pattern the spoke body. To show the hidden bodies, select them in the “**Solid Bodies**” folder and click in the **Show** command.



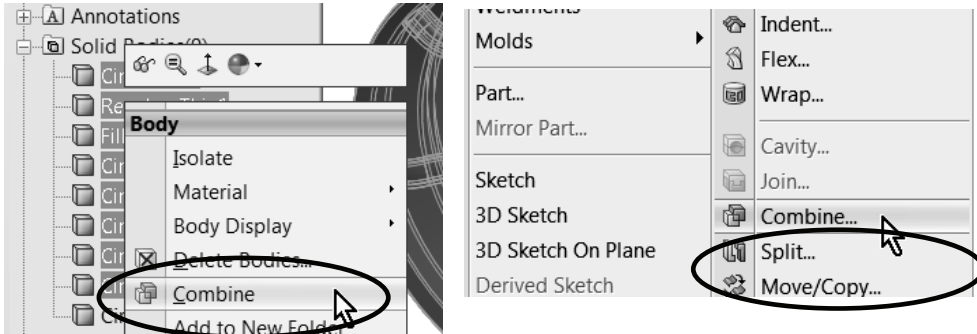
28. – Select the **Circular Pattern** command, but instead of making a pattern of features, we'll make a pattern of bodies. Select a circular edge to define the pattern's axis and expand the "**Bodies to Pattern**" selection box. In the graphics area, select the Spoke and make 7 copies equally spaced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



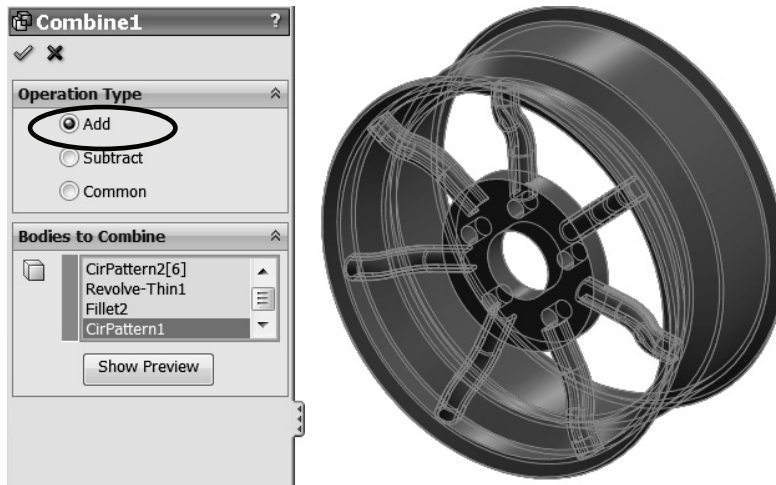
29. – Now we have nine solid bodies in our part.



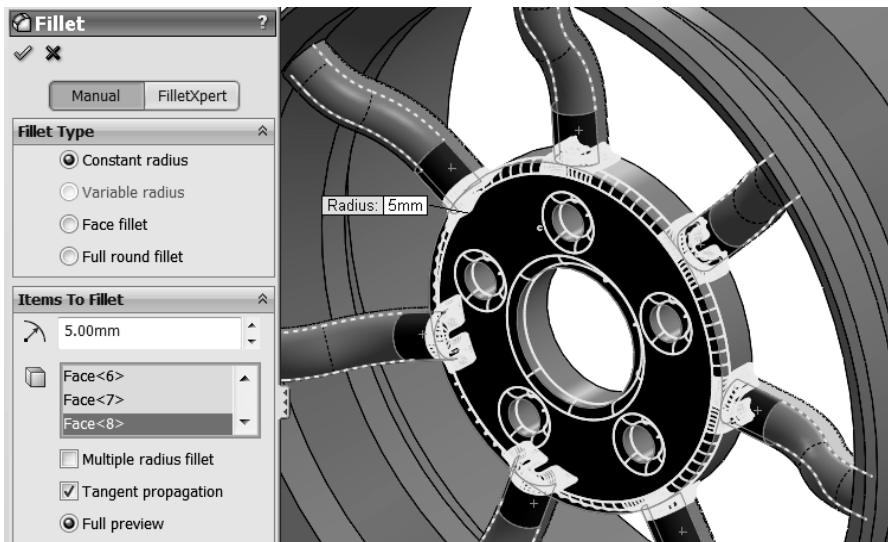
30. – We are ready to combine all solids into a single body. Select all bodies in the FeatureManager, right mouse click on them, and select “**Combine Bodies**” from the pop-up menu, or from the menu “**Insert, Features, Combine,**” and select all the bodies in the graphics area.



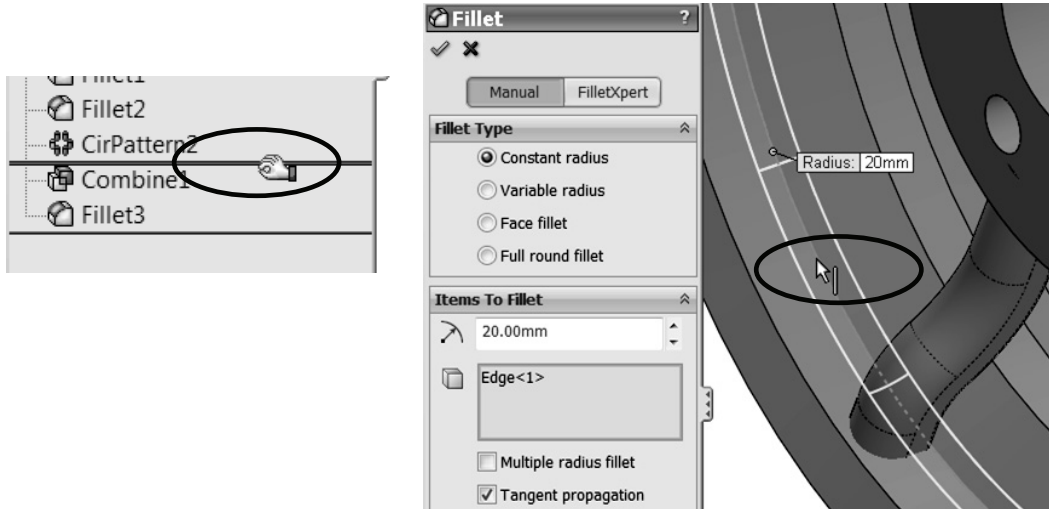
31. – Select the “**Add**” option and click **OK** to finish. All bodies are now combined into a single solid.



32. – Add a 5mm fillet to the faces indicated.



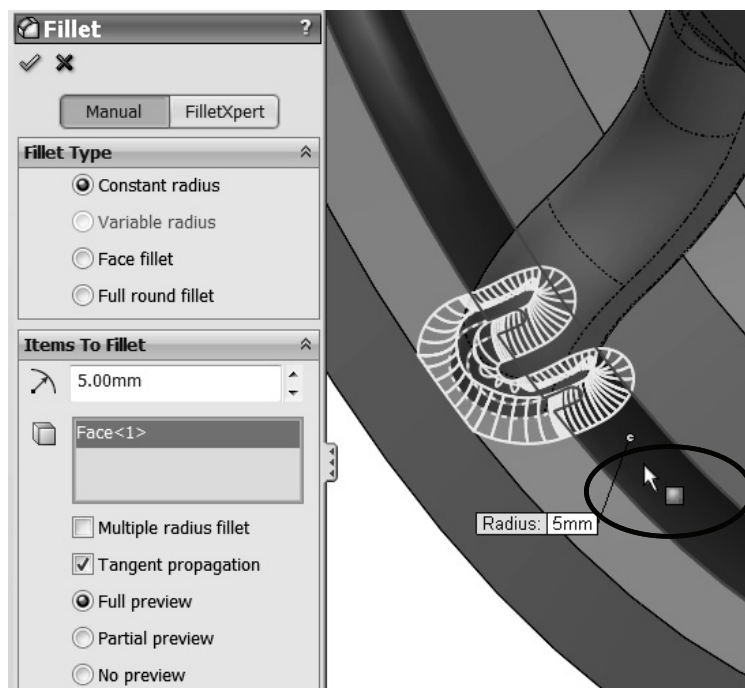
33. – To add a similar fillet to the other side of the Spokes, first we need to add a fillet to the Rim's body. If we try to add the fillet after the bodies are combined, the fillet will fail. Drag the Rollback bar from the bottom of the FeatureManager before the “Combine1” feature and add a 20mm fillet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



34. – Drag the Rollback bar down to the bottom when finished.



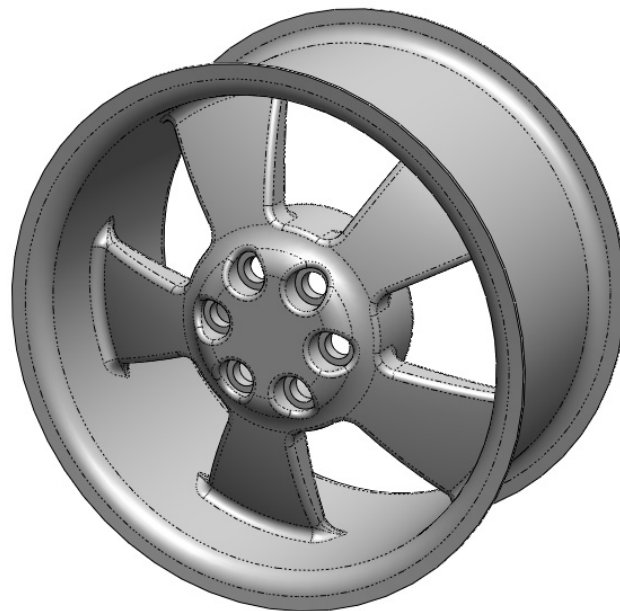
35. – Add a 5 mm fillet to the face of the previous fillet to propagate it to all Spokes at the same time.



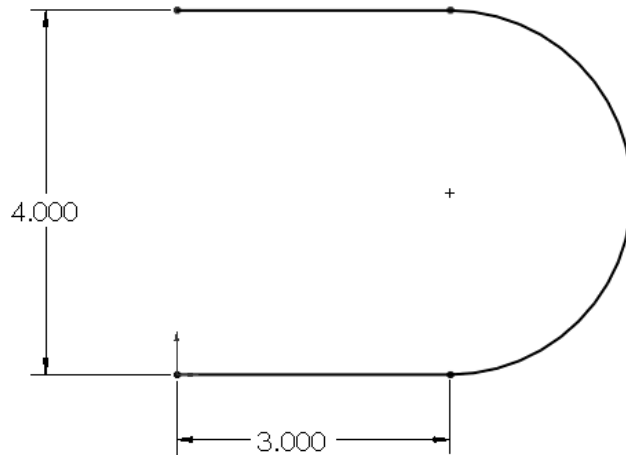
36. – Your finished wheel should now look like this:



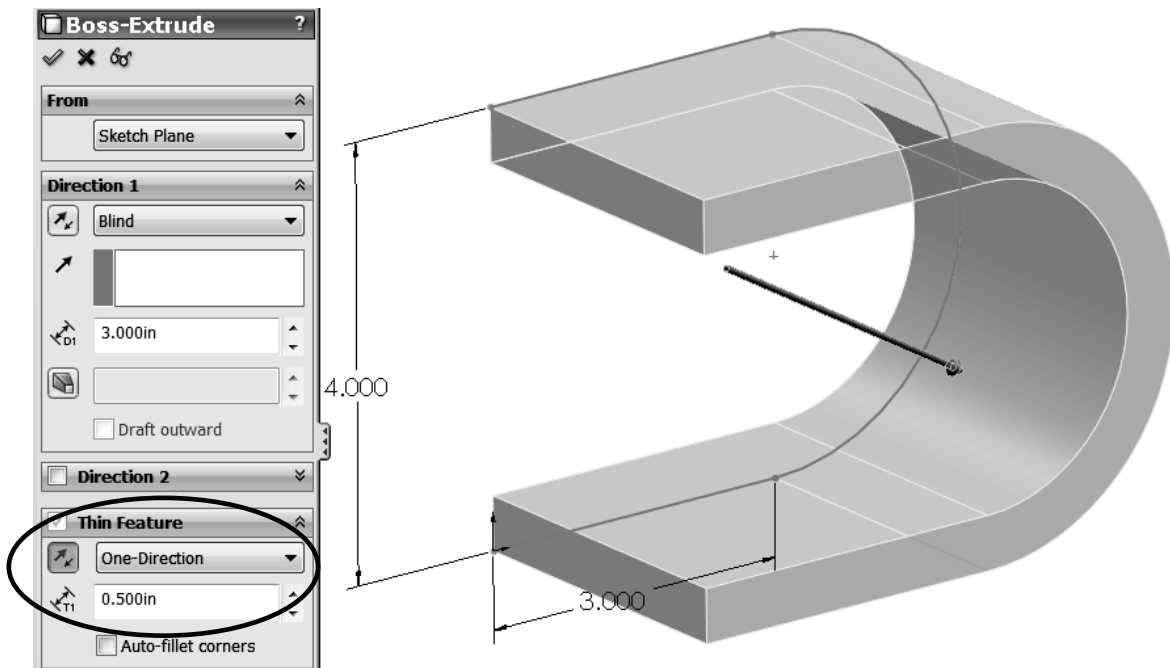
I know this is not a good-looking wheel, but it's a good example to show how to work with multi body parts and perform local operations. We'd love to see the designs from our readers; please email us a picture to areyes@mechanicad.com.



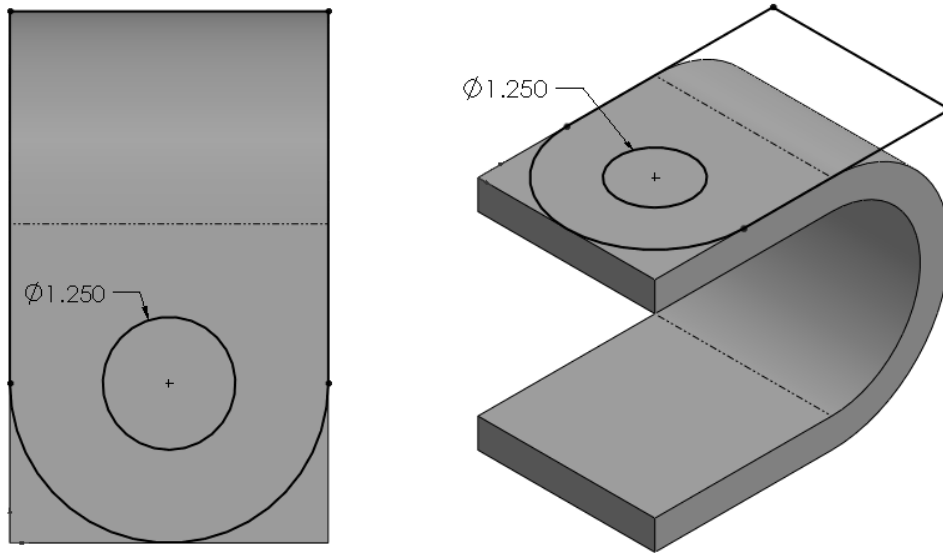
37. – Now we'll learn more about combining bodies. What we did with the wheel was to add bodies, in essence merging or fusing them. The other two operations we can do when combining bodies is to subtract one or more bodies from another, or get the common volume between them. To obtain the common volume 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.

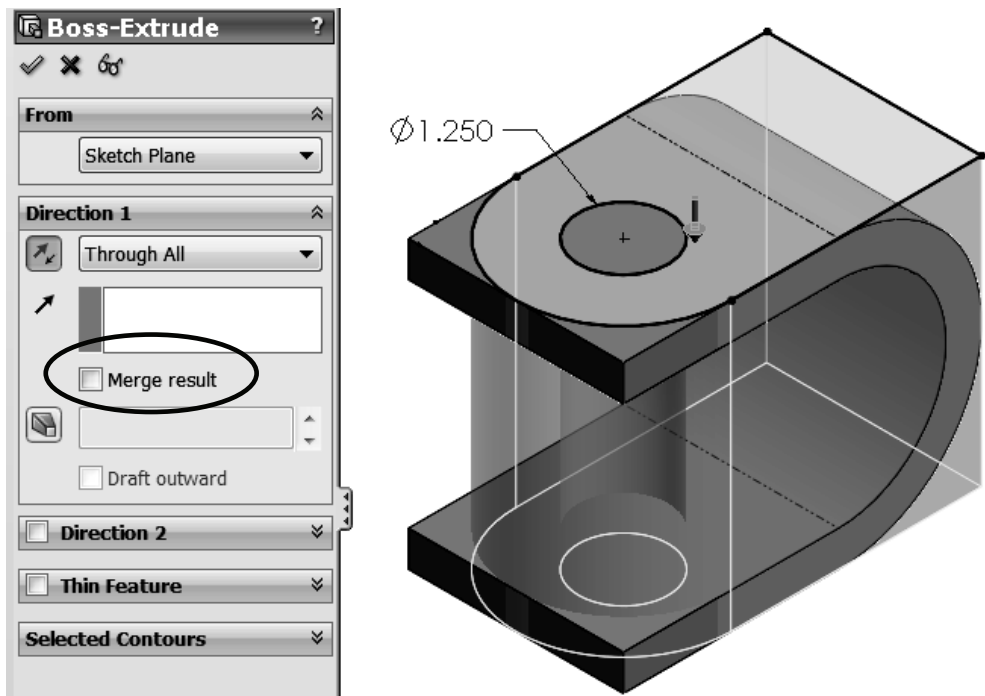


39. – For the next feature, add a sketch in the Top face of the first feature. Looking at it from a Top view should look like this:

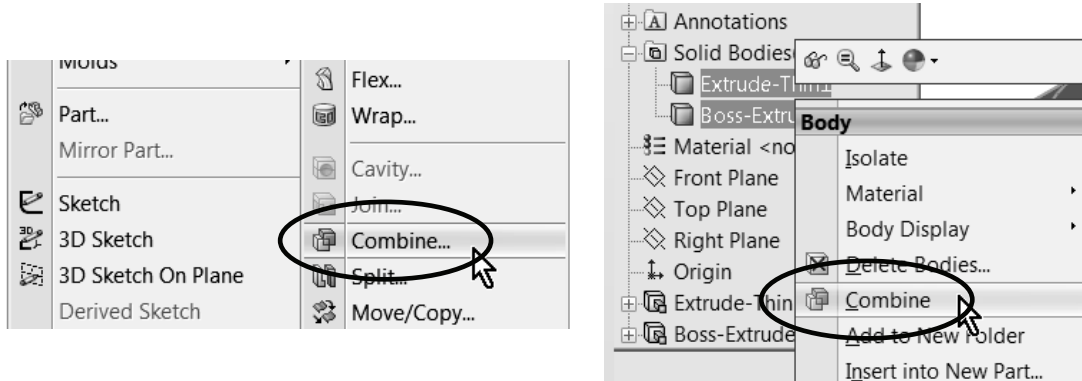


Notice the only dimension we need to add is the hole's diameter; everything else is defined with geometric relations.

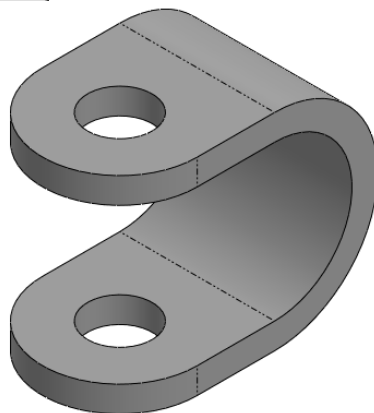
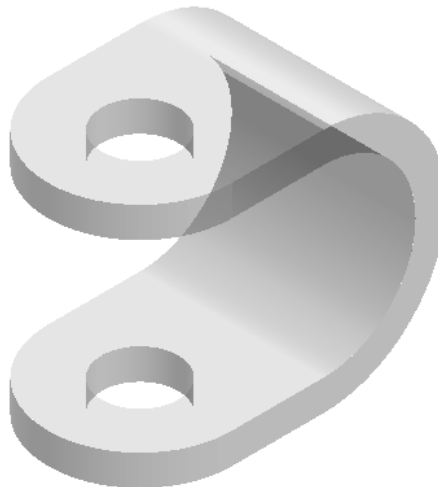
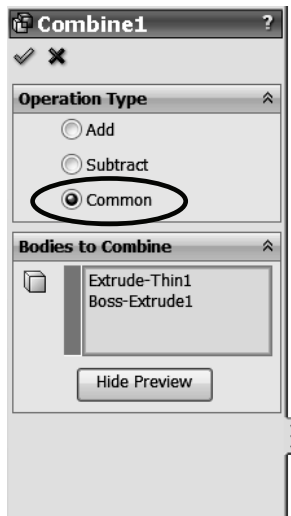
40. – Extrude the second sketch downwards with the “**Through All**” end condition, and be sure to uncheck the “**Merge result**” checkbox. We want to have two separate bodies.



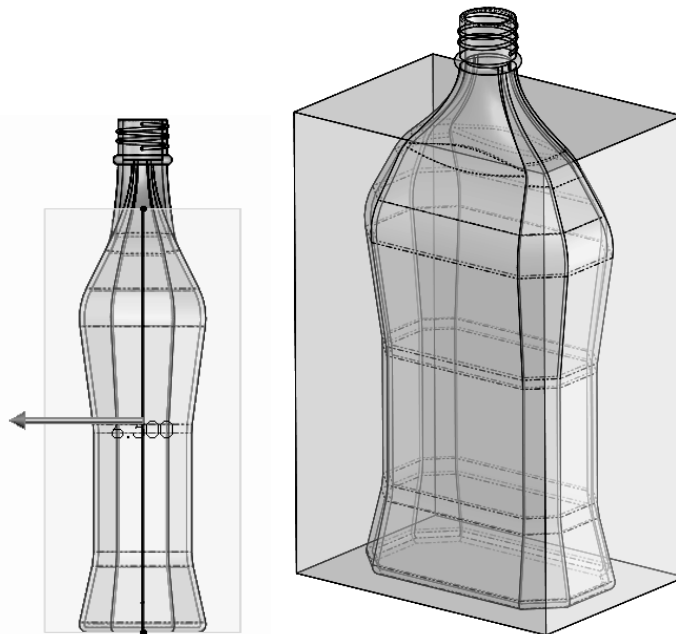
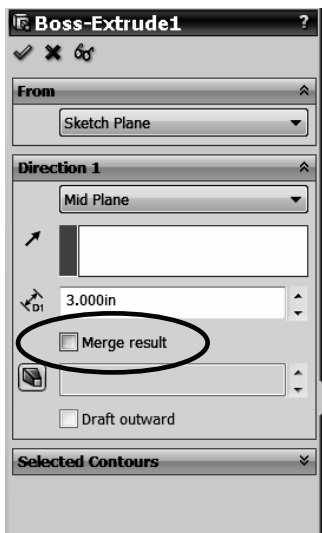
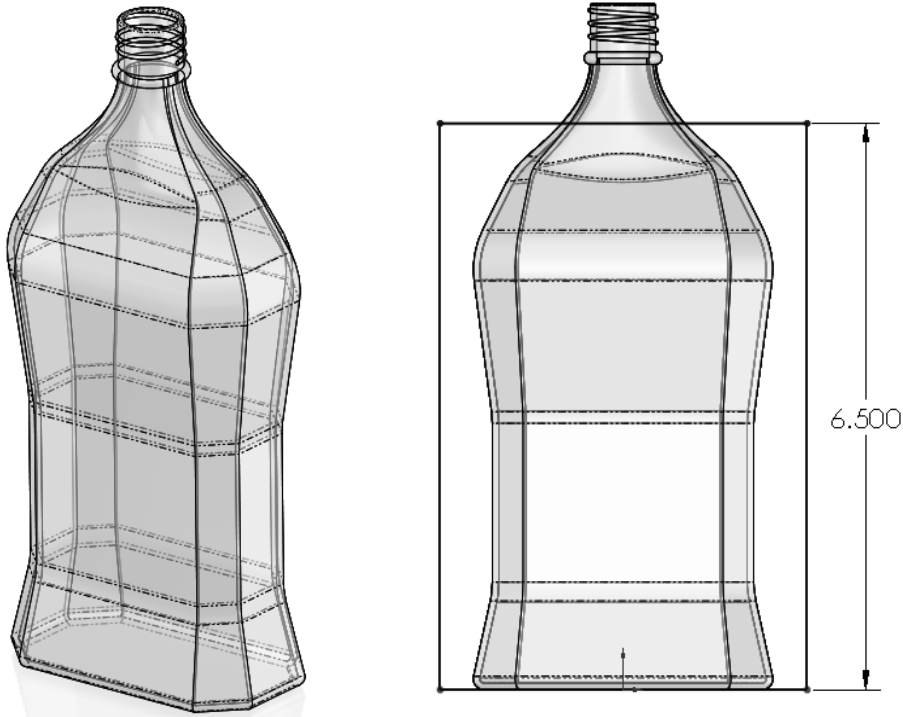
41. – Now that we have two separate bodies, we can combine them to get the common volume. Select the menu “**Insert, Features, Combine;**” or from the “**Solid Bodies**” folder in the FeatureManager select BOTH bodies, right mouse click in either one and select “**Combine**” from the pop-up menu.



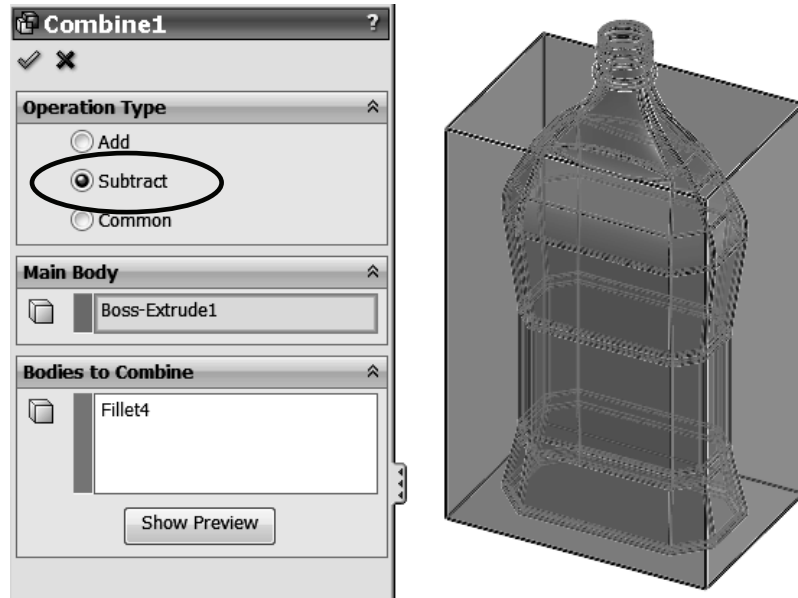
42. – In the Combine operation select the “**Common**” option under “**Operation Type**”. Click on the “**Preview**” button to see what the resulting body will look like and click **OK** to finish.



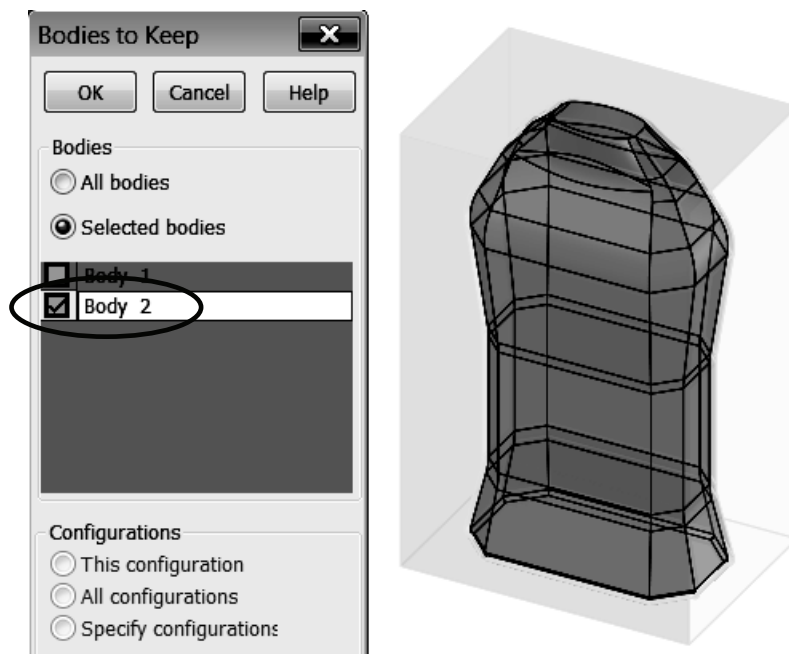
43. – The next operation to cover is the difference between bodies. One common use for a body difference is to obtain a mold's core and/or cavity (which we'll cover later in the book), but here we'll show how to obtain the volume capacity of an irregularly shaped bottle. Download the part *Bottle.sldprt* from our website (www.mechanicad.com/download.html) and open it. 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 with only the resulting 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.

