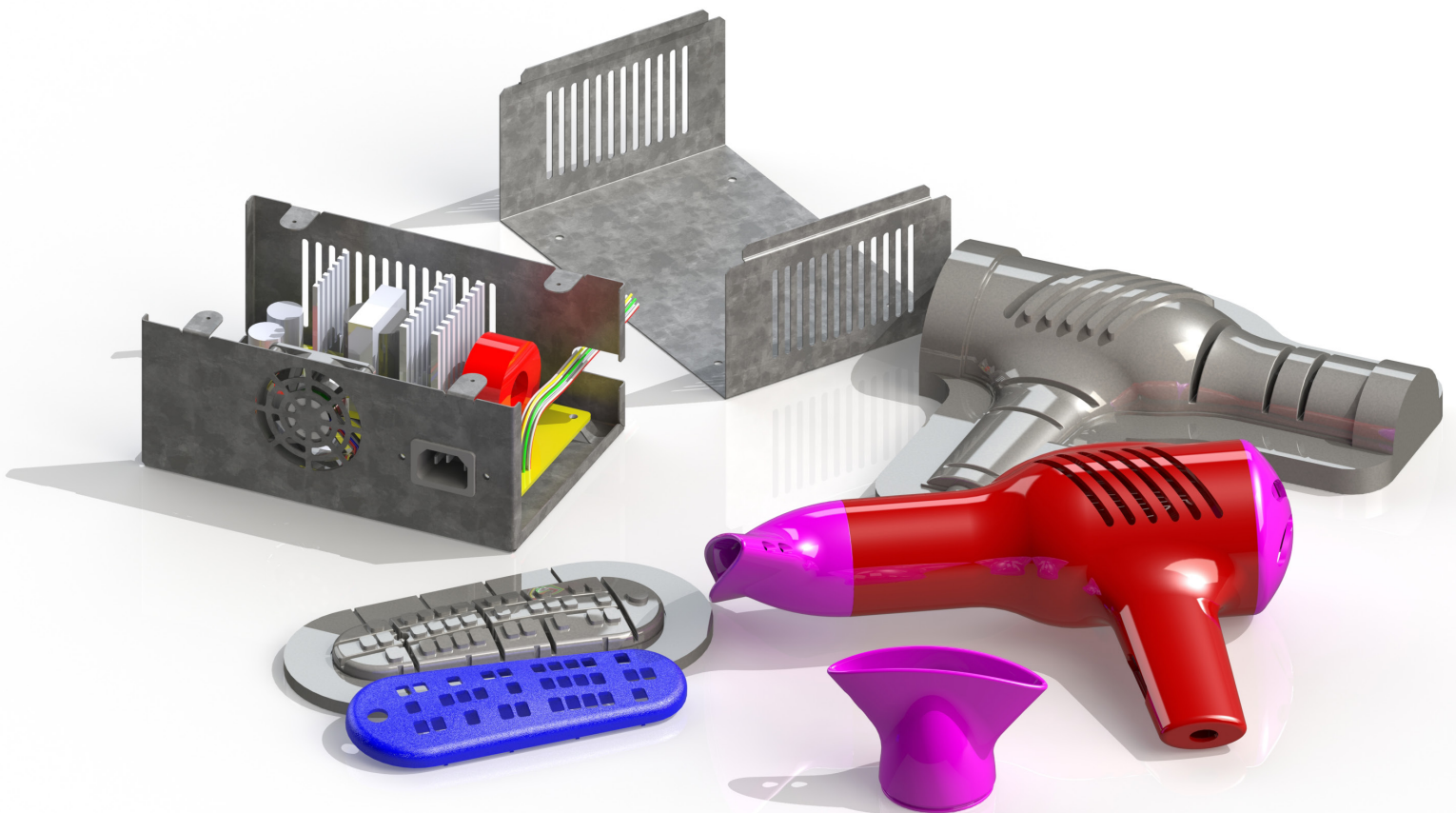


Beginner's Guide to SolidWorks® 2015 Level II

Sheet Metal, Top Down Design,
Weldments, Surfacing and Molds



Alejandro Reyes MSME, CSWP, CSWI



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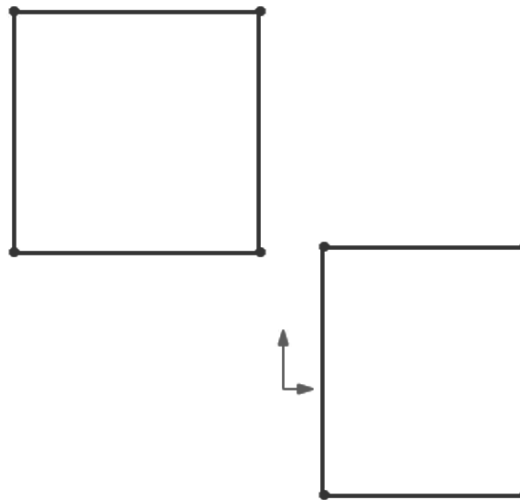
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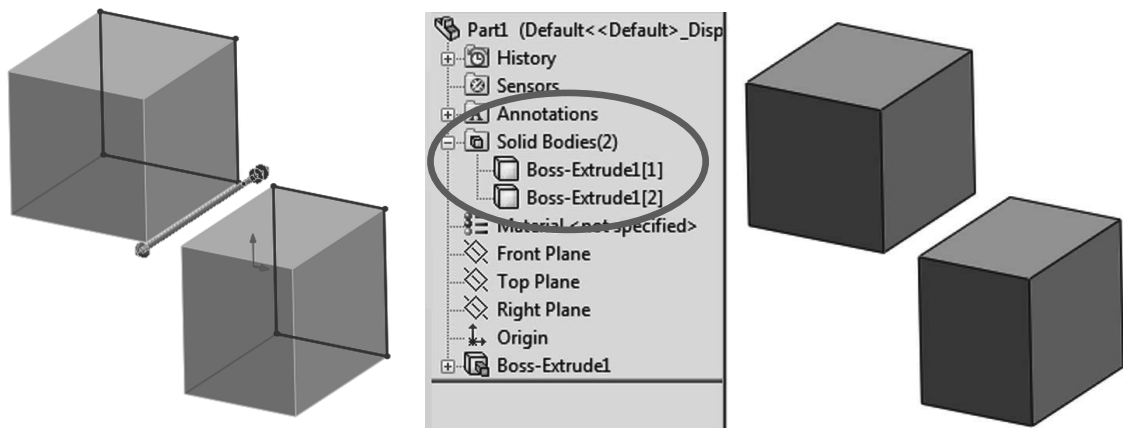
Multi Body Parts

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. A different way to create a multi body part is by splitting an existing part into multiple bodies (this technique will be shown later). To show how it works, make a new part, and add a sketch to the “*Front Plane*” as shown. Do not worry about dimensions in this example as we are just illustrating the concept of how multi bodies work.

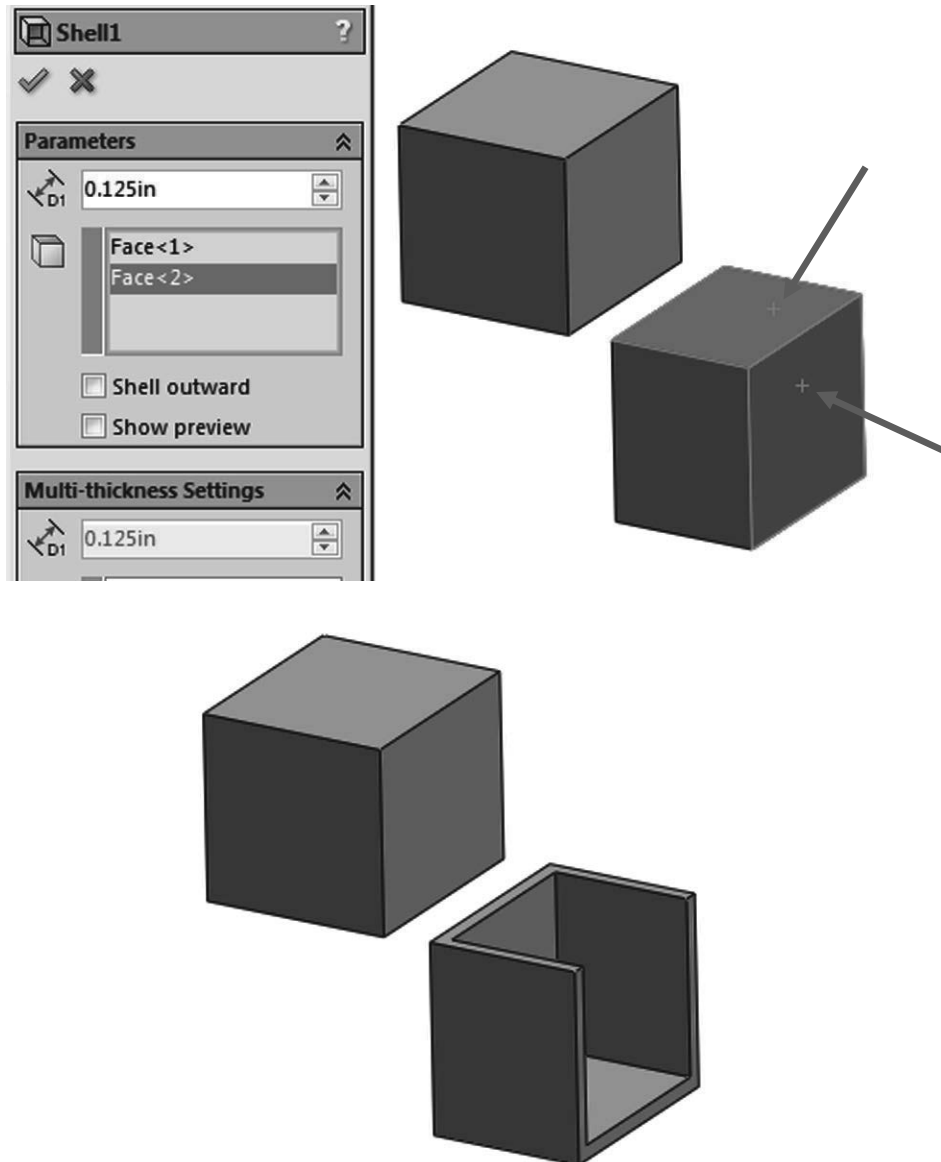


2. - Select the “**Boss Extrude**” command. Notice that we are not given a warning about having two separate bodies; it just works. Extrude any size that looks similar to the following image (dimensions are not important at this time) and click **OK** to finish. (Making two or more extruded/revolved/swept/lofted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

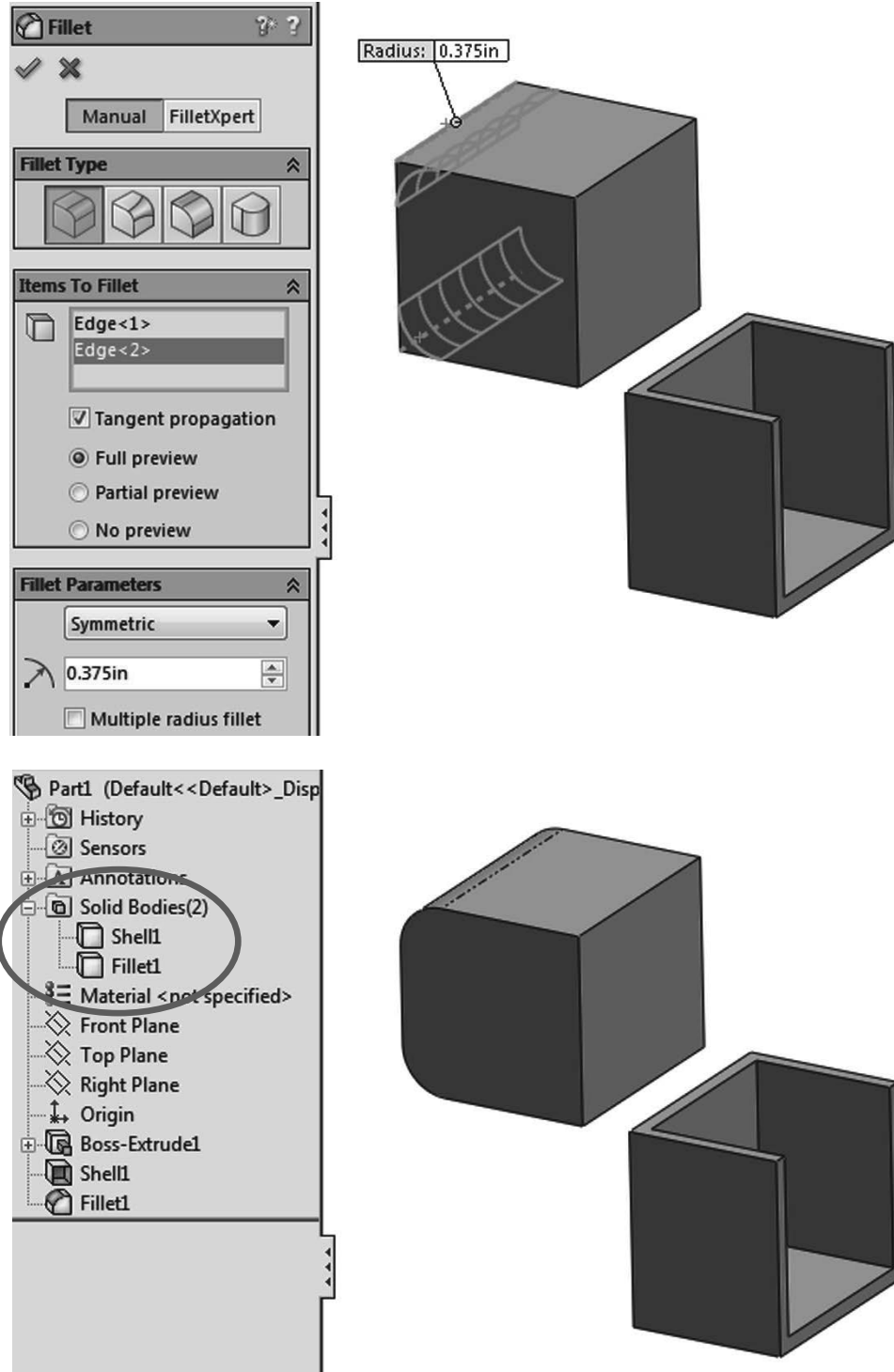


The first thing we notice is a new folder in the FeatureManager called “*Solid Bodies (2)*.” This folder is automatically added when SolidWorks detects multiple disjointed bodies in a part and lists the number of bodies found in the part (in this case 2). If we expand the folder, we can see the two bodies 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, in other words, a feature affecting only one body. Local operations are applied features like fillets, shells, chamfers, etc., which can only be applied to a single body at a time. Select the “Shell” command from the Features toolbar, and shell the bottom body removing the indicated faces.

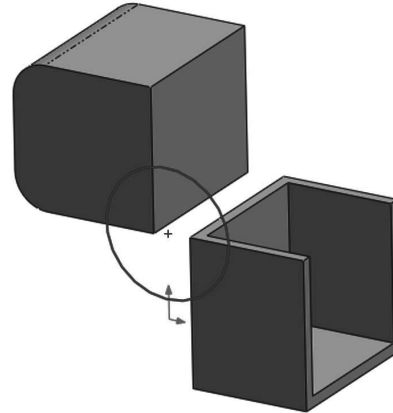


4. - Select the **Fillet** command and round two corners of the top body as shown.

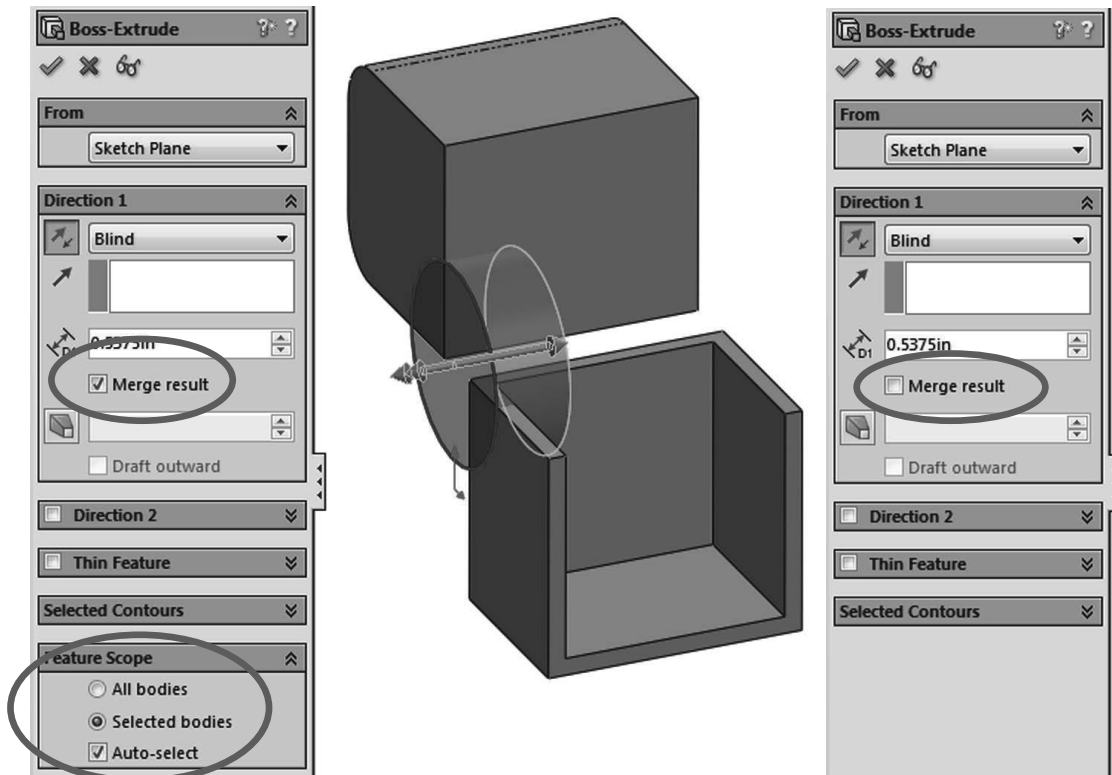


To help the reader understand the concept of local operations better, “local” means that we can add applied features (features that don't require a sketch like Shell, Fillet, Chamfer, Draft, etc.) to one body at a time. Notice the name of a body changes to the last feature applied to it.

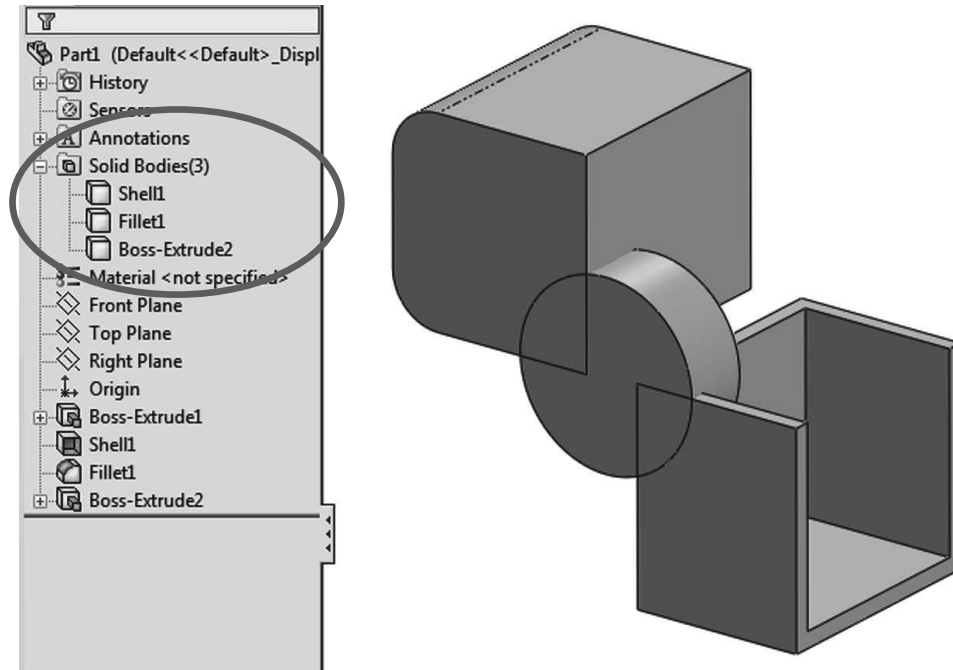
5. - If we add another boss extrude feature that intersects with existing bodies, by default it will automatically merge with them. At the time of making the extrusion we can optionally select which bodies to “merge” (or fuse) with to make a new single body. Add the following sketch to explore the merge option when extruding a new boss into existing bodies. Select the front face of a body and make a sketch that intersects both bodies as shown.



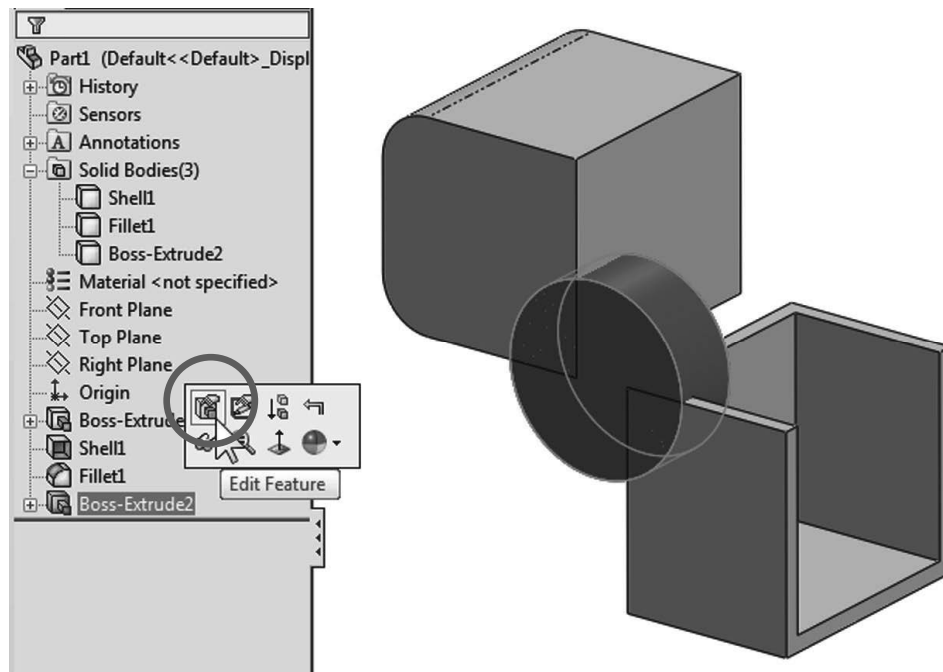
6. - Extrude the sketch *into* the existing bodies. (The same would happen if we extrude *away* from the bodies, because they are touching the new extrusion.) Notice the “Merge result” option in the Extrude command. It’s always been there (except when there are no solid bodies) and by default is always checked. When we have multiple bodies in a part we see an additional option at the bottom called “Feature Scope.” This is where we can select the existing bodies we want to merge the new extrusion with. The options are to merge the new extrusion with “All bodies” or “Selected bodies” automatically or manually. 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 solid body it intersects with or touches. If the “Merge result” option is un-checked, “Feature Scope” is automatically removed, and the new Boss-Extrude will not merge with any body. For this step uncheck the option “Merge Result” and click OK to finish.

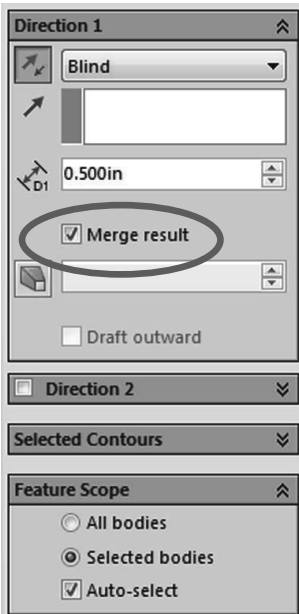
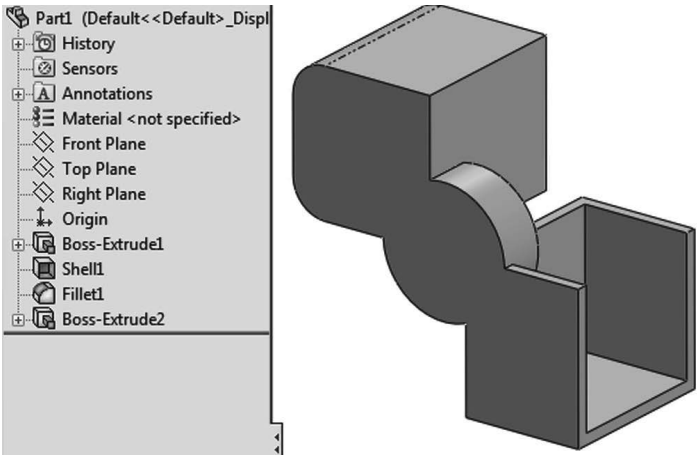
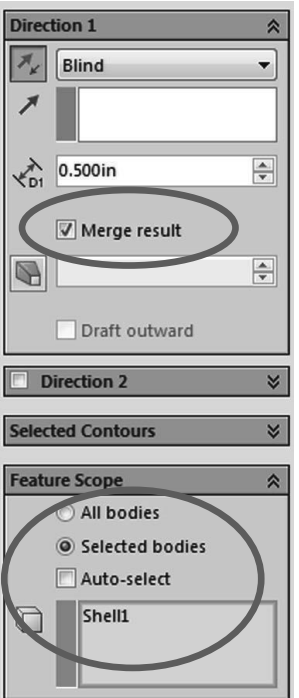
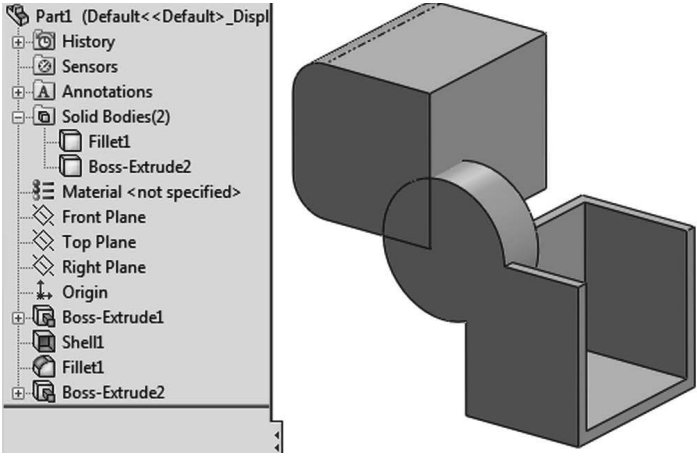


7. - After adding this extrusion we end up with three bodies in our part. See how the different bodies' edges intersect each other. If the "Merge result" option had been checked, we would not see these edges overlapping as they would have merged into a single body.

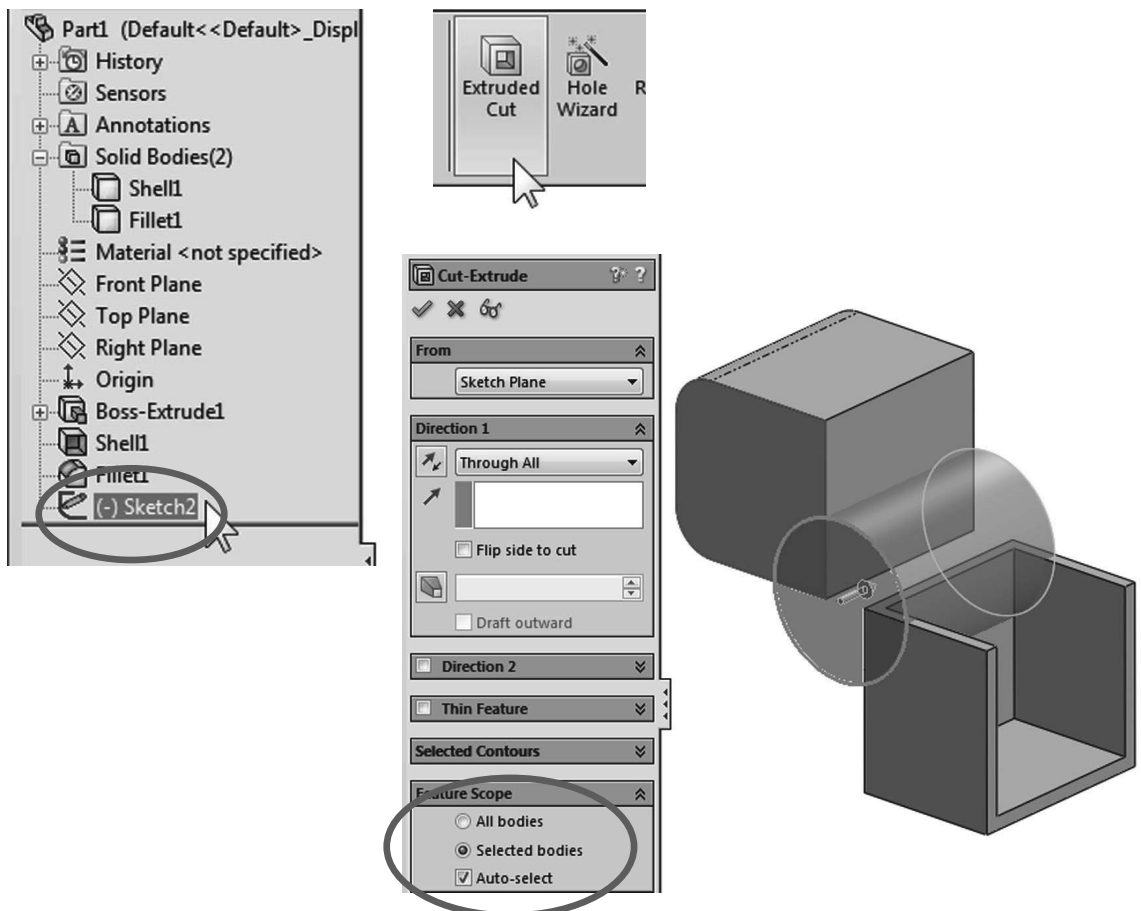
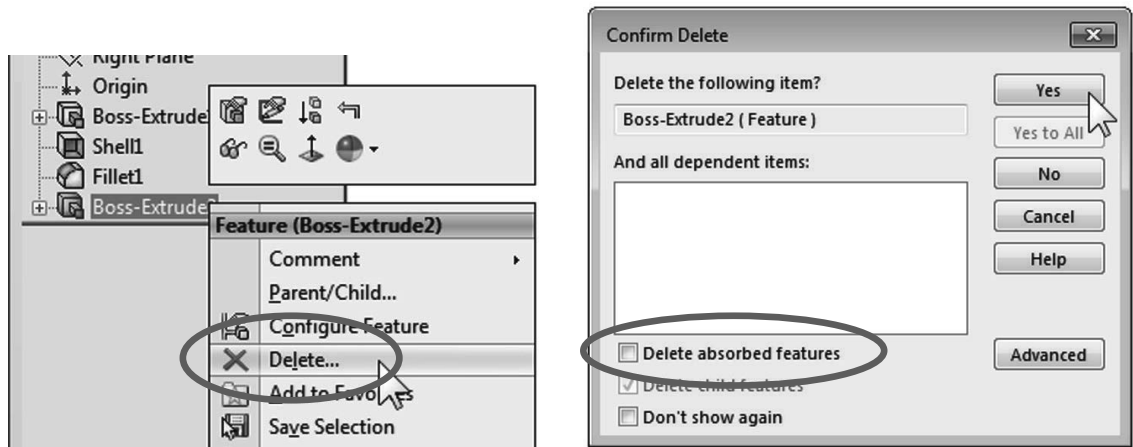


Edit the last extrusion to explore the effect of different "Merge Result" and "Feature Scope" combinations.

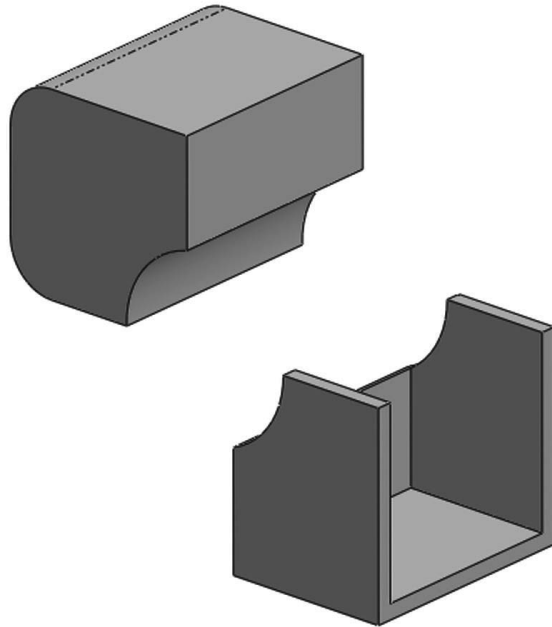


Combination:	Result:
<p>Merge Result: Checked Feature Scope: Auto-Select</p> 	<p>Single body. By touching both of the existing bodies “Auto-select” merges with them, fusing them into a single solid body. The “Solid Bodies” folder is no longer visible as there is only one solid body in the part.</p> 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

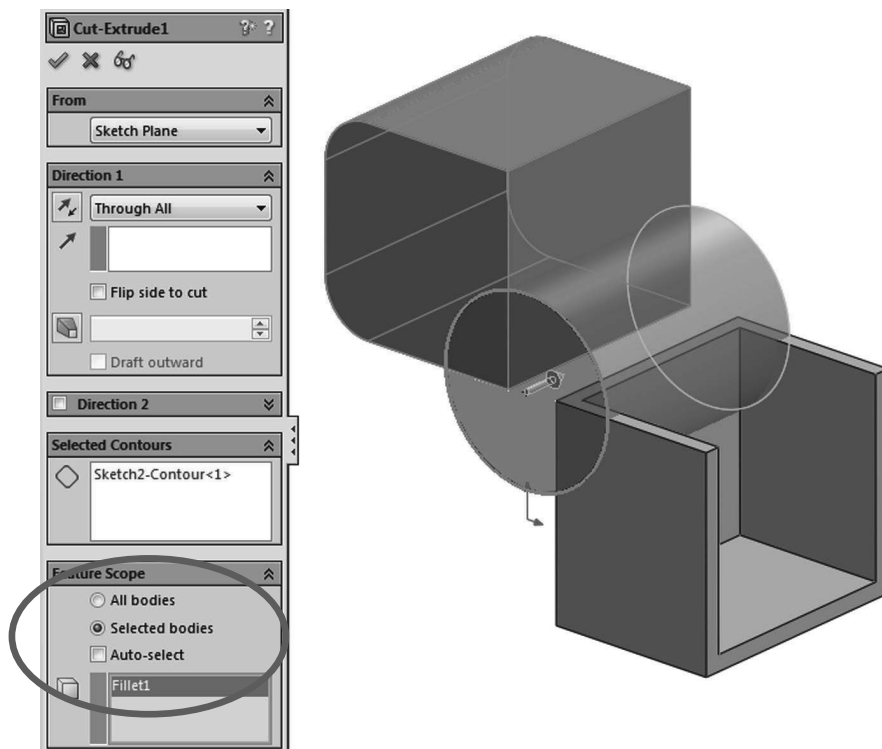
8. - The “Merge result” option works the same way with any feature that adds material to the part, including revolved boss, sweep, loft, etc. Now we’ll see how it works when we remove material. Delete the Boss-Extrude feature and keep the sketch. Select the sketch in the FeatureManager and make a Cut-Extrude “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.



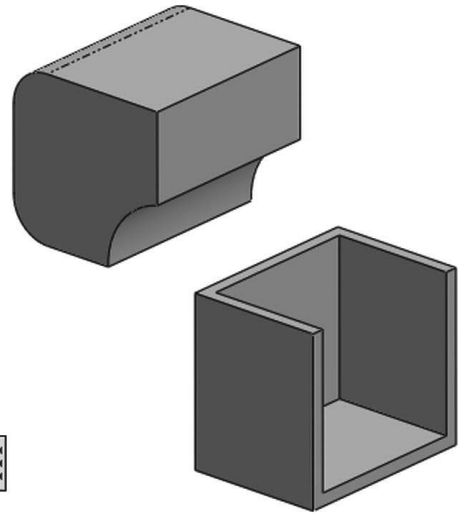
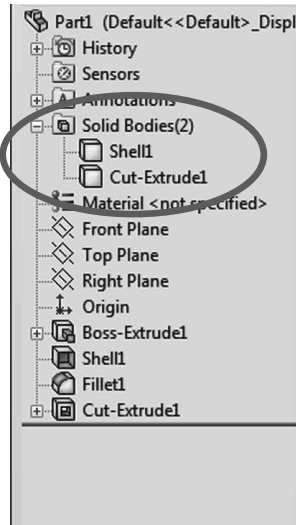
9. - What we end up with is the same two bodies we had before, but now they have a cut through them because the cut automatically selected them.



10. - Edit the “Cut-Extrude1” 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 when the cut overlaps the lower body.

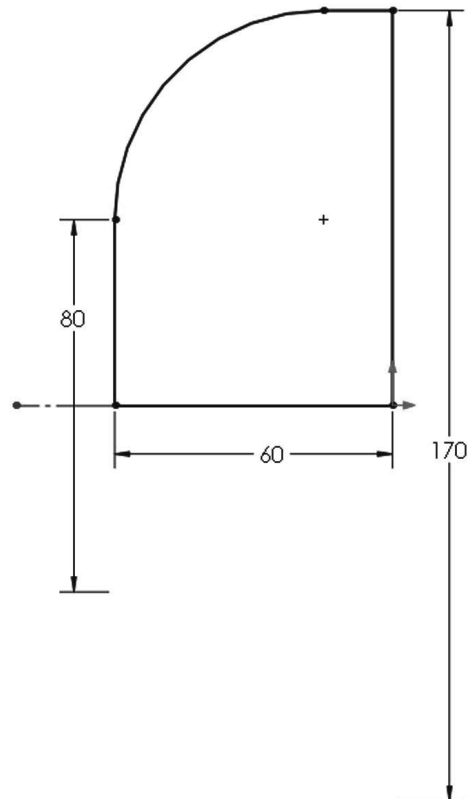


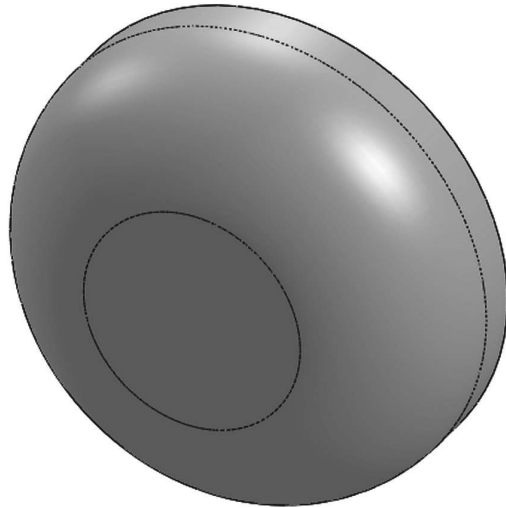
And just as with the features that add material, this technique works the same way with all the features that remove material including revolved cut, sweep cut, loft cut, etc.



11. - Multi bodies is a powerful technique to model parts that would otherwise be difficult to complete. A 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. In this example we’ll make a car’s wheel using multi-body techniques to complete it, including obtaining the common volume between two bodies as well as adding two different bodies. 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. Make a new part; the first feature for the wheel will be the hub or mounting pad. Draw the following sketch in the “*Right Plane*” of the part and make a 360° Boss-Revolve feature. 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).

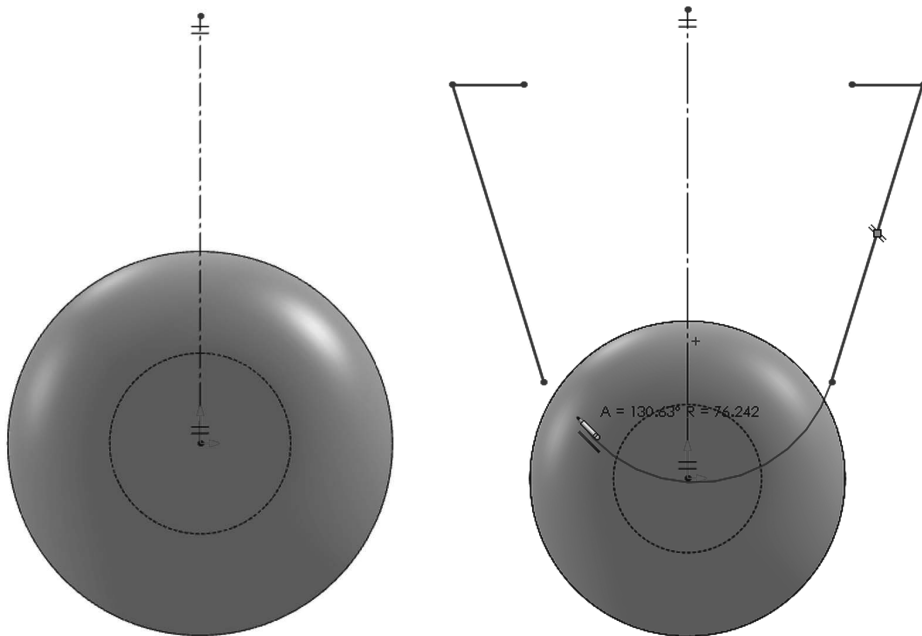
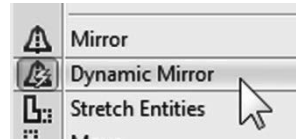




12. - Add a new sketch in the front flat face of the part, and add the following sketch. The arcs are tangent to the circular edges. Make the extrusion 20mm to the front (Direction 1) and “Through All” going to the back (Direction 2). Turn off the “Merge result” option to have two different bodies.

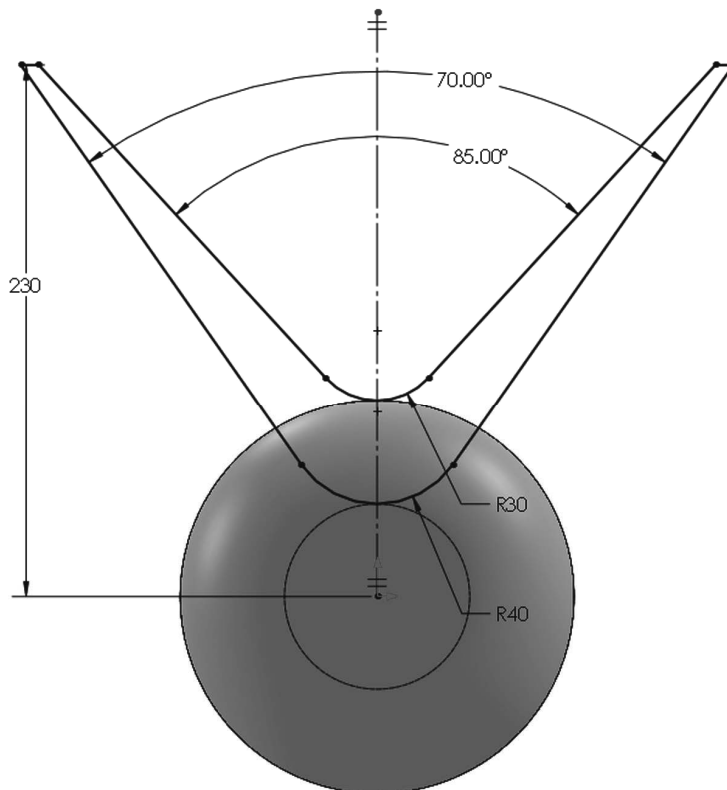
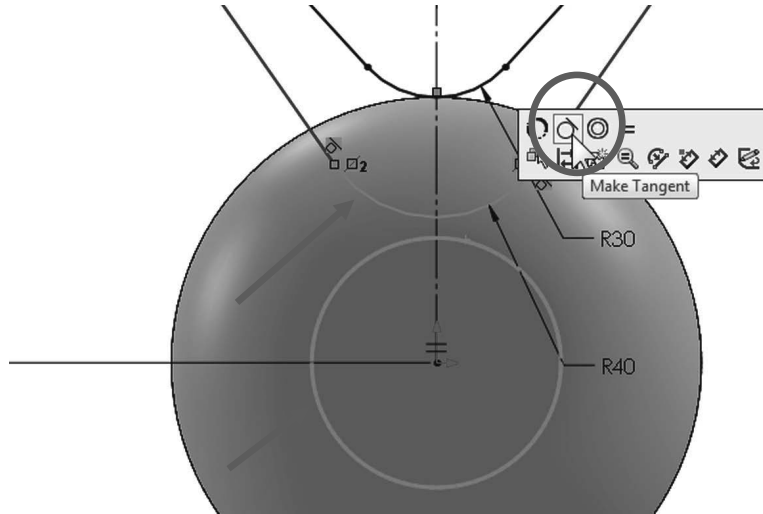
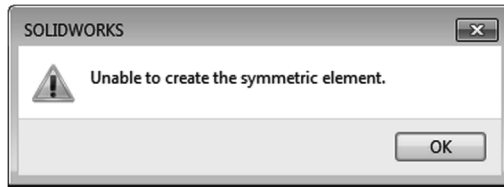


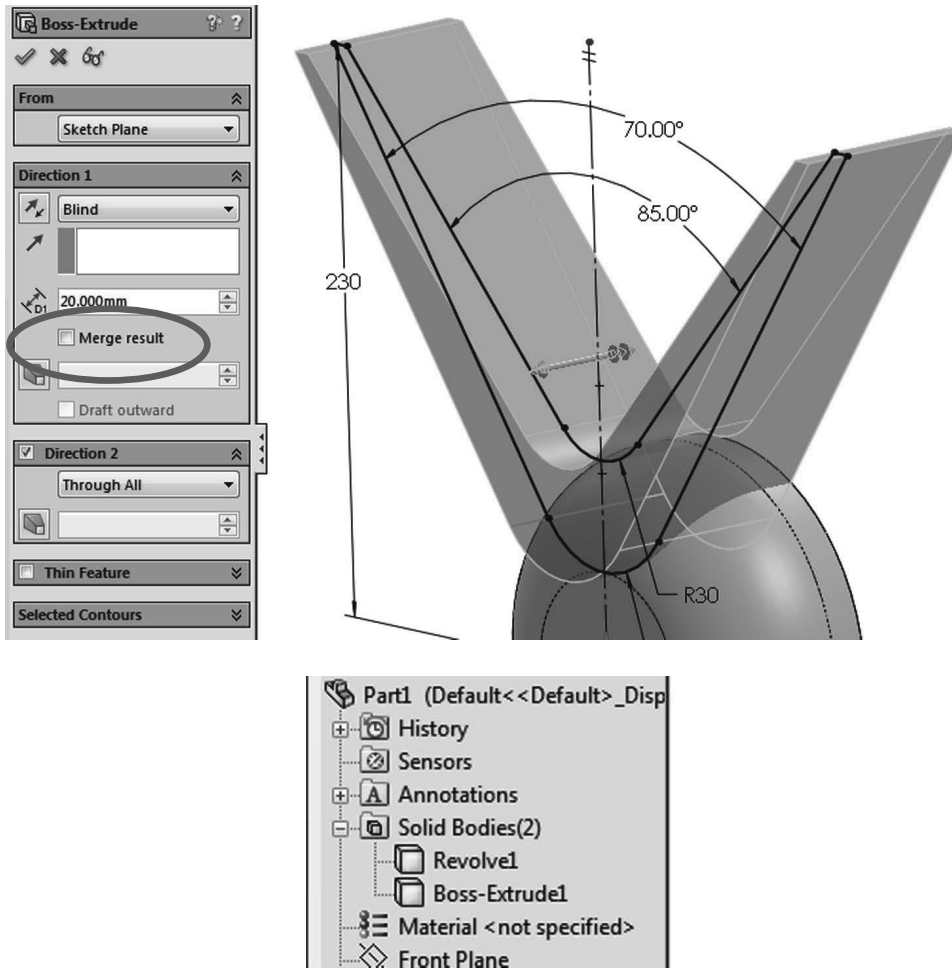
To make the sketch symmetric about the centerline, draw the centerline first, and while it's selected, turn on the “**Dynamic Mirror**” tool in the menu “**Tools, Sketch Tools, Dynamic Mirror**.” After we turn it on, the centerline will show an equal sign at both ends letting us know that whatever we draw on one side will be automatically drawn in the other side. At the same time, symmetric geometric relations will be automatically added. To turn it off, select it again in the menu.



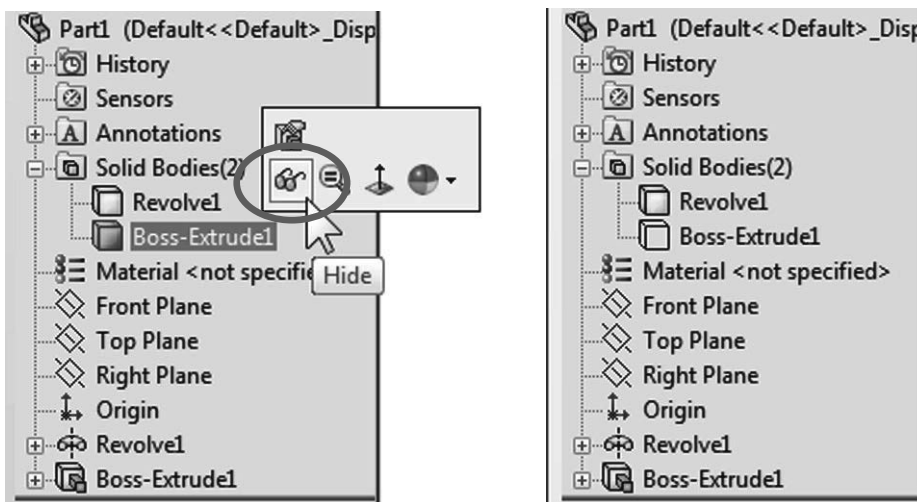


When using the “Dynamic Mirror” command, if we draw a line, arc, etc. crossing the centerline so that the mirror element would overlap the original, we get a warning letting us know and the mirror copy is not created.

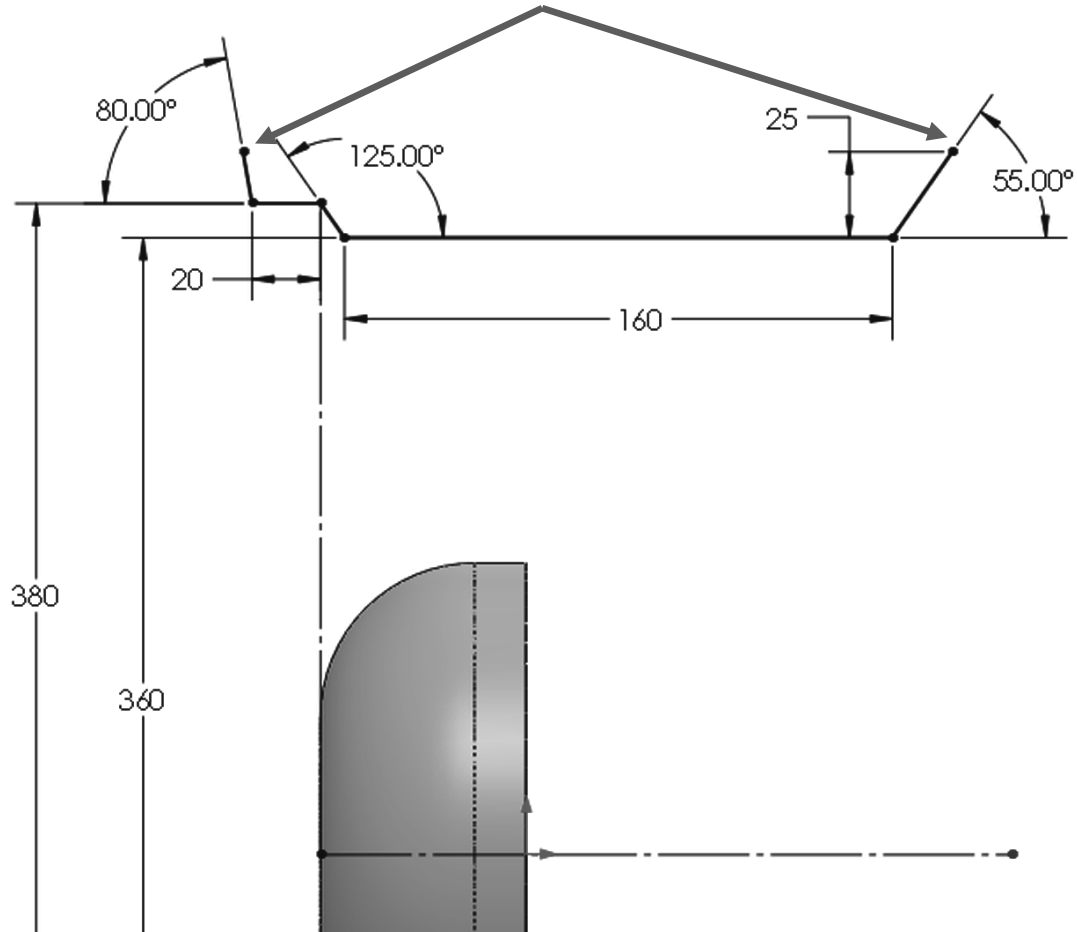




13. - In the next step we'll make the outside rim of the wheel. To better visualize this step, the second body will be hidden from view. Just like we can hide a part in an assembly, we can hide a solid body in a part. In the "Solid Bodies" folder, select the "Boss-Extrude1" body and select "Hide" from the pop-up toolbar.

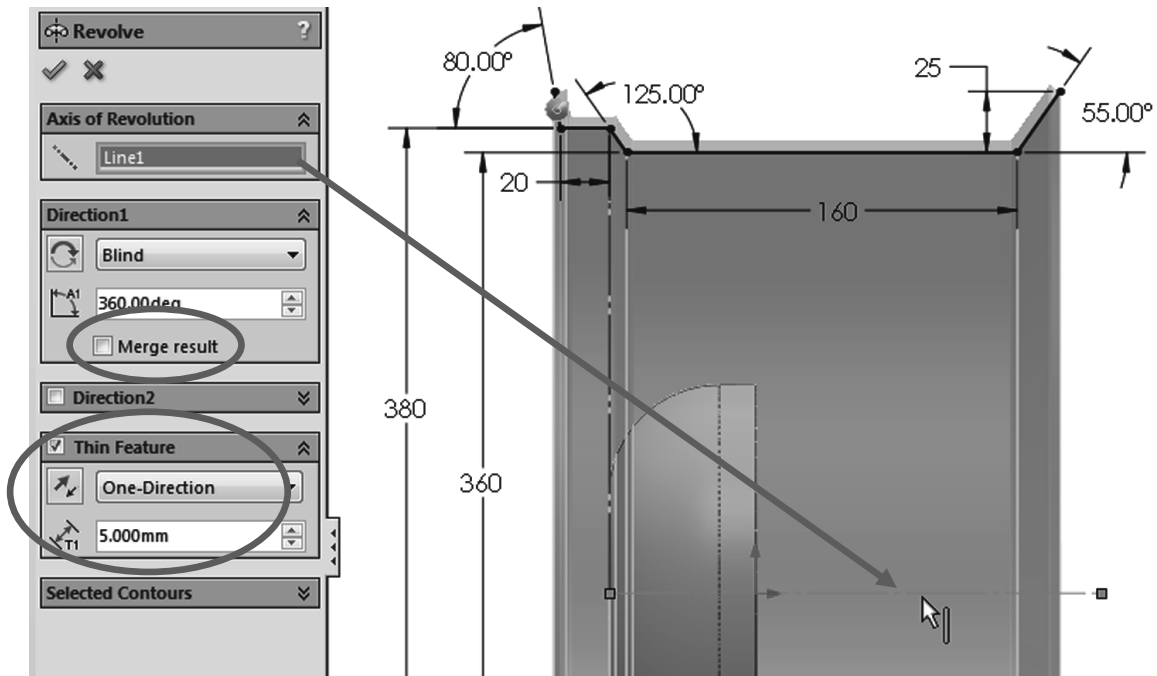


14. - Add the following sketch in the “*Right Plane*”; we’ll use it to create a thin revolved feature to build the wheel’s rim. The 360mm and 380mm dimensions are doubled about the horizontal centerline. The endpoints at the ends of the sketch are at the same height; add a Horizontal geometric relation between them.

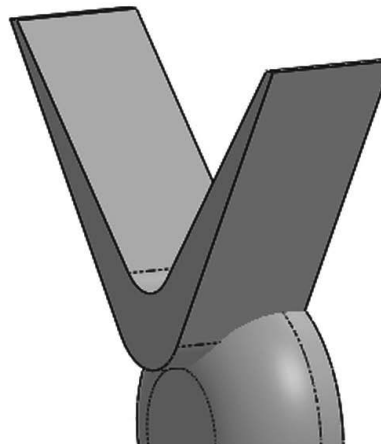
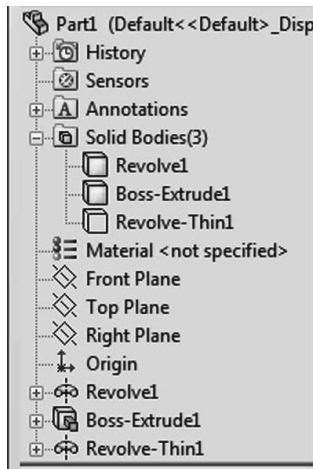
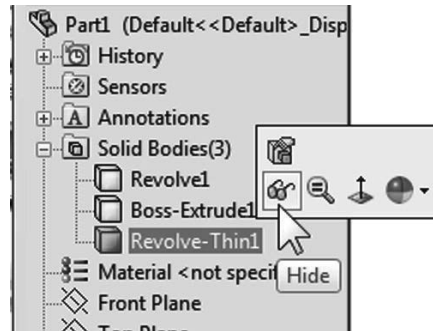
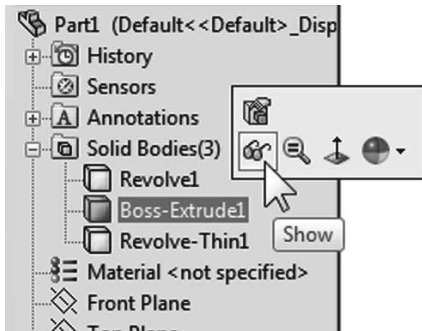


15. - Select the “**Revolved Boss/Base**” command and make a 360 deg. thin revolve feature. SolidWorks will ask if we want to close the sketch; select “No” to continue. Use the horizontal centerline as the “Axis of Revolution,” turn off the “Merge result” checkbox and make the “Thin Feature” 5mm thick, going away from the wheel.

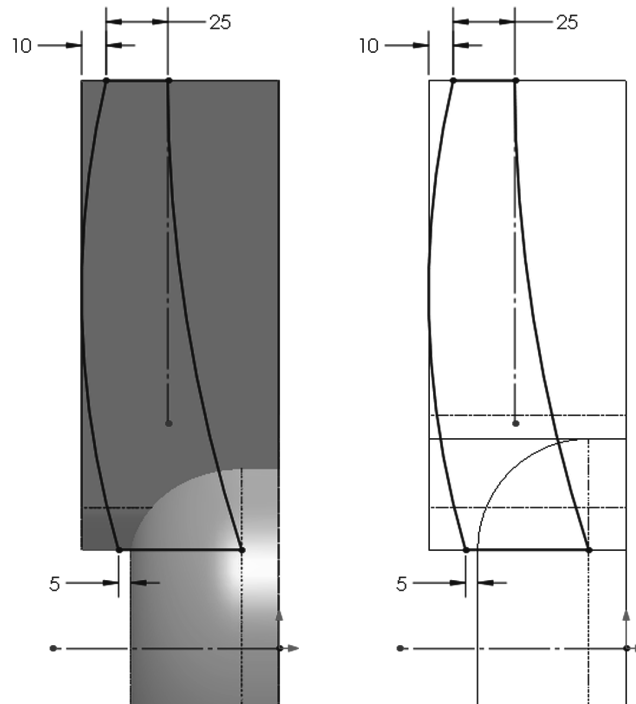
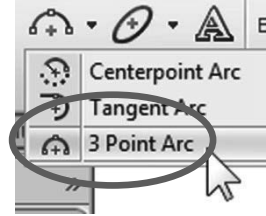




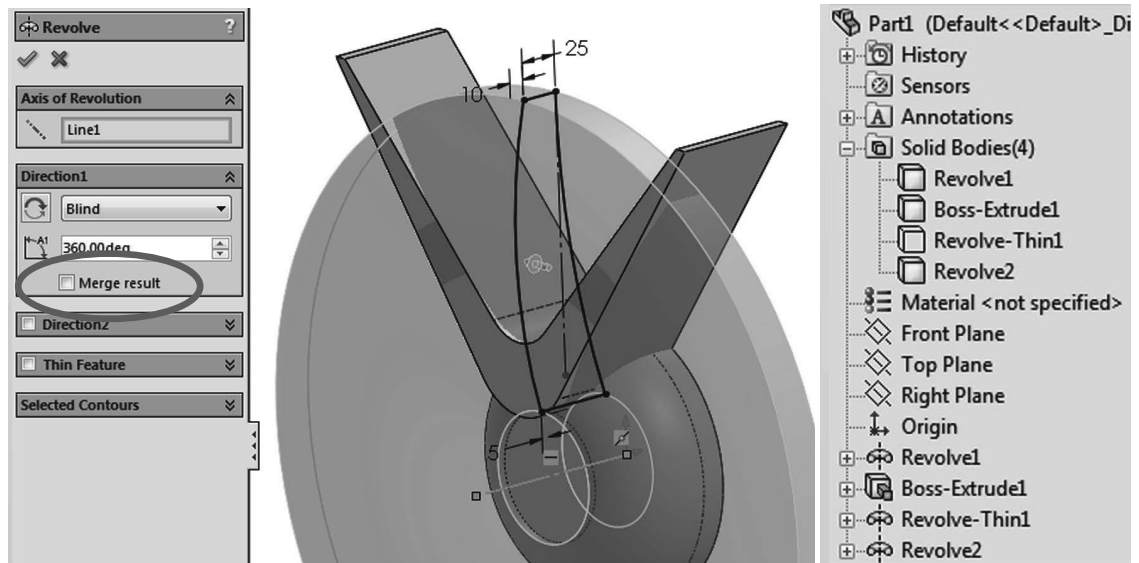
16. - Now we have three different solid bodies in our part. The next feature will be another revolved feature that will be used to build the wheel's spokes. Before making the new solid body, hide the "Revolve-Thin1" body and show the "Boss-Extrude1" body.



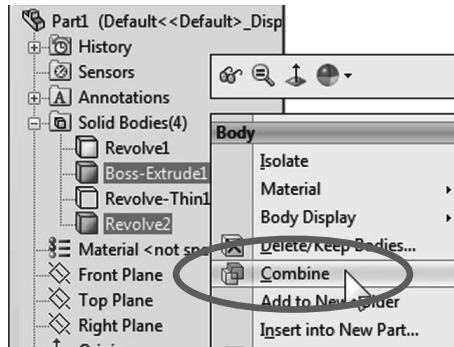
17. - After hiding the revolved thin feature, add a new sketch in the “Right Plane.” Note: The arc on the left is tangent to the solid body’s edge, and the arc on the right is tangent to a vertical centerline. Don’t forget the horizontal centerline at the origin to make the revolved feature about it. We can use a “**Three Point Arc**” for this sketch. The part is shown in both shaded and wireframe modes for clarity.



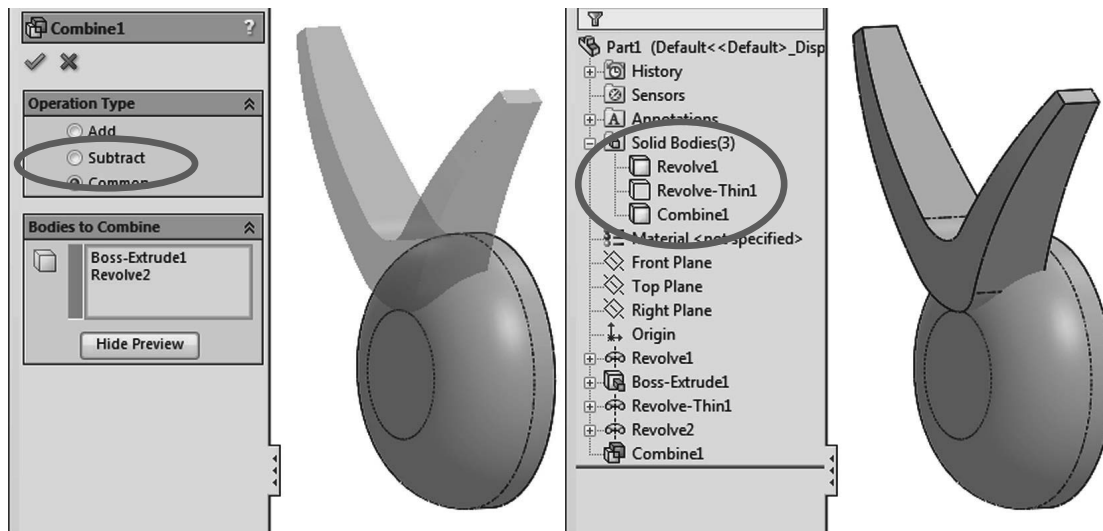
18. - Make a 360 deg revolved boss using the horizontal centerline and turn off the “Merge result” checkbox to create the fourth solid body.



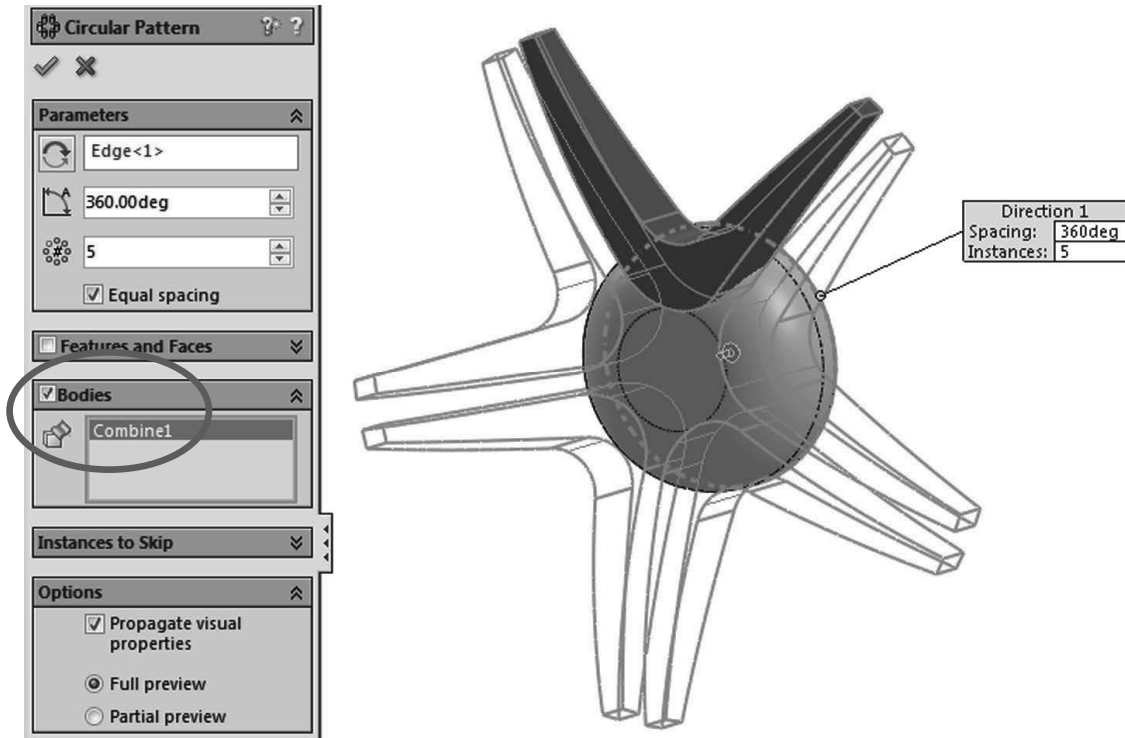
19. - In the next step we will obtain the common volume between two of the bodies to get the spokes of the wheel. Select the “*Boss-Extrude1*” and the “*Revolve2*” bodies in the “*Solid Bodies*” folder holding the “*Ctrl*” key while selecting, and from the right-mouse-button menu select the command “**Combine**” or from the menu “**Insert, Features, Combine.**”



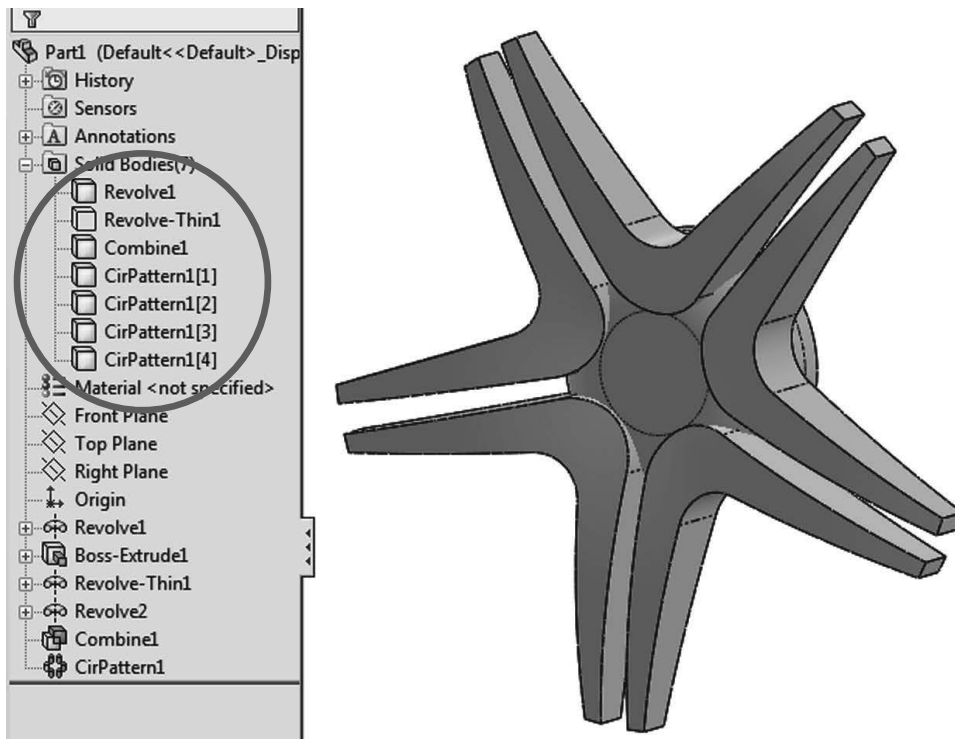
In the “*Operation Type*” options select “*Common*” and then press the “*Show Preview*” button (it will change to “*Hide Preview*”). The “**Combine**” command is used to Add bodies to form a new one, Subtract one or more bodies from another, or get the Common volume between two or more bodies, as is the case in this step. After clicking OK to calculate the common volume, the “*Bodies to Combine*” are consumed by the operation and replaced by the new body.



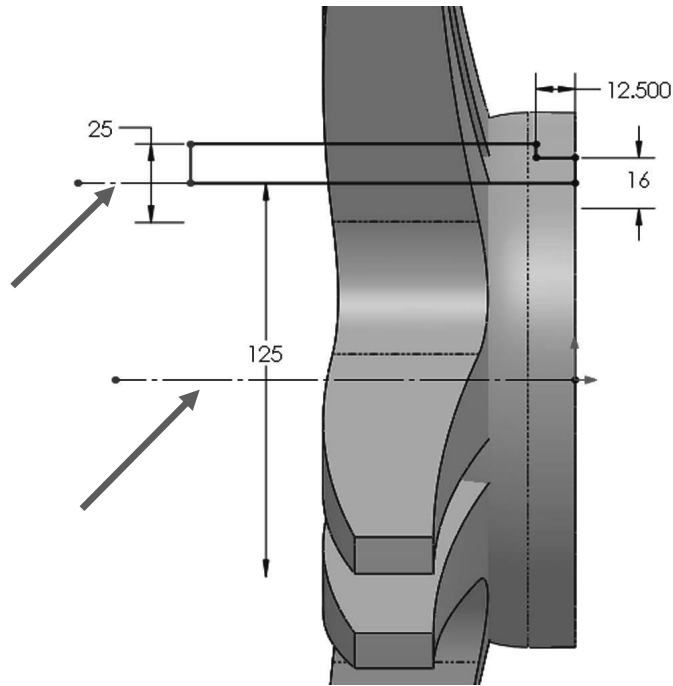
20. - Now that we have the body we are interested in to create the spokes, we need to make a circular pattern using the combined body as a seed feature. Select the “*Circular Pattern*” command using any circular edge for the “*Pattern Axis.*” For this step, instead of selecting the “*Combine1*” feature to make our pattern, select the “*Bodies to Pattern*” selection box and add the combined body to make the pattern. Change the number of instances to 5 equally spaced in 360 degrees. Click OK to continue.



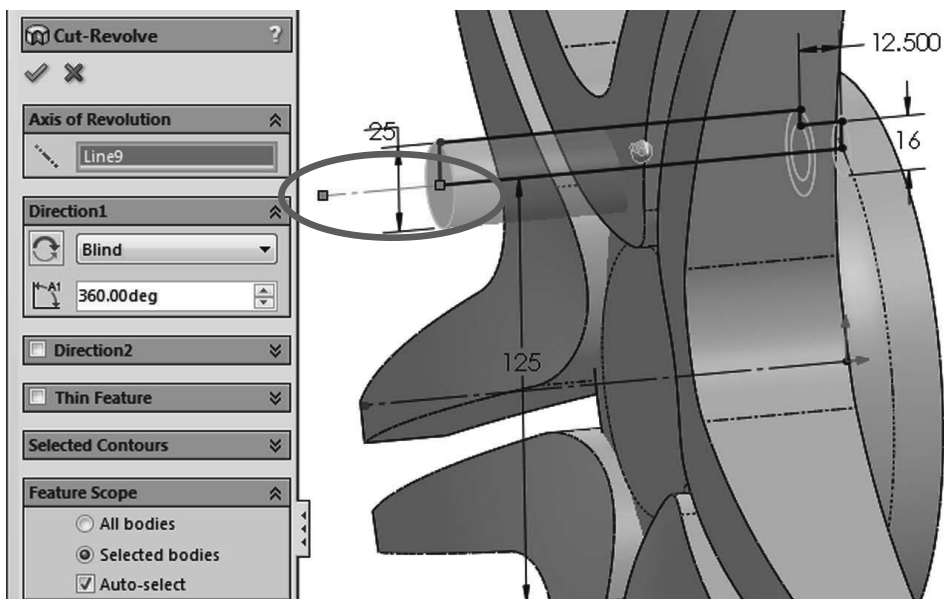
21. - After adding the circular pattern we have 7 different bodies in our wheel.



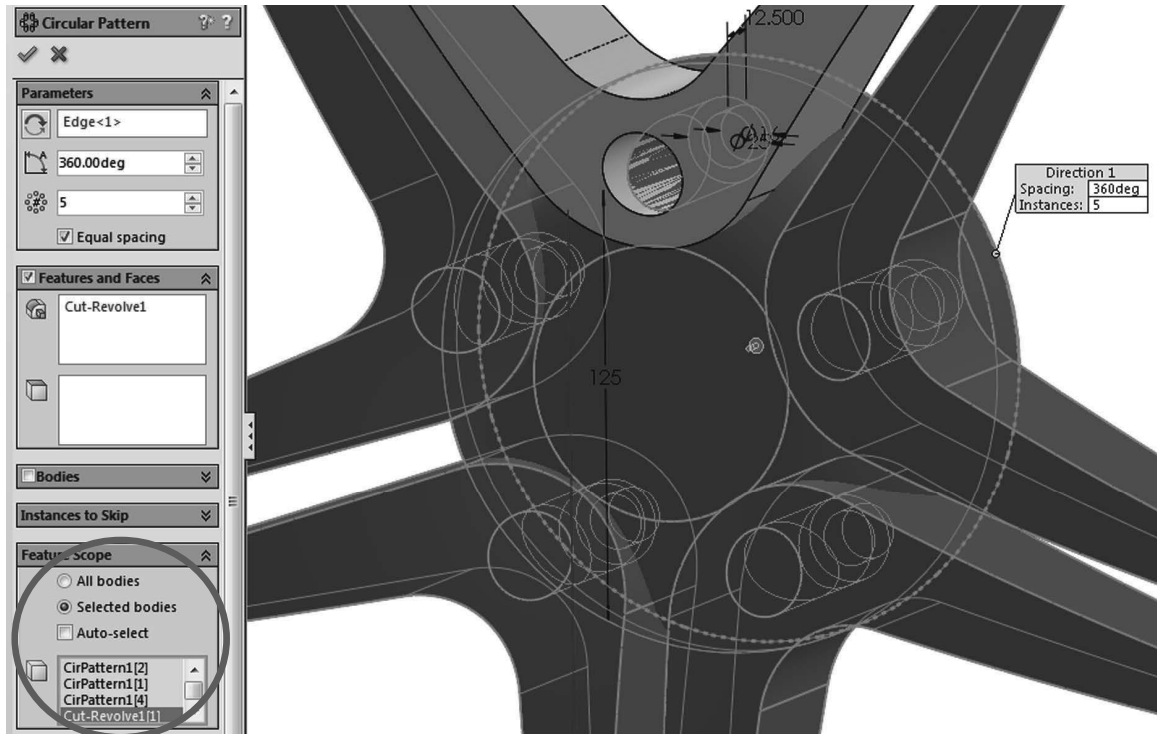
22. - The next step could be done after we combine the bodies into a single body, but we chose to add it at this time to show more multi-body functionality. In this step we'll add the holes for the lug nuts. With the following sketch we'll make the holes for the lug nuts. Add the sketch in the "Right Plane." Note the 28mm and 16mm dimensions are doubled about the top centerline; the 125mm is doubled about the centerline in the middle of the part. Make the sketch big enough to extend well past the left side of the part.



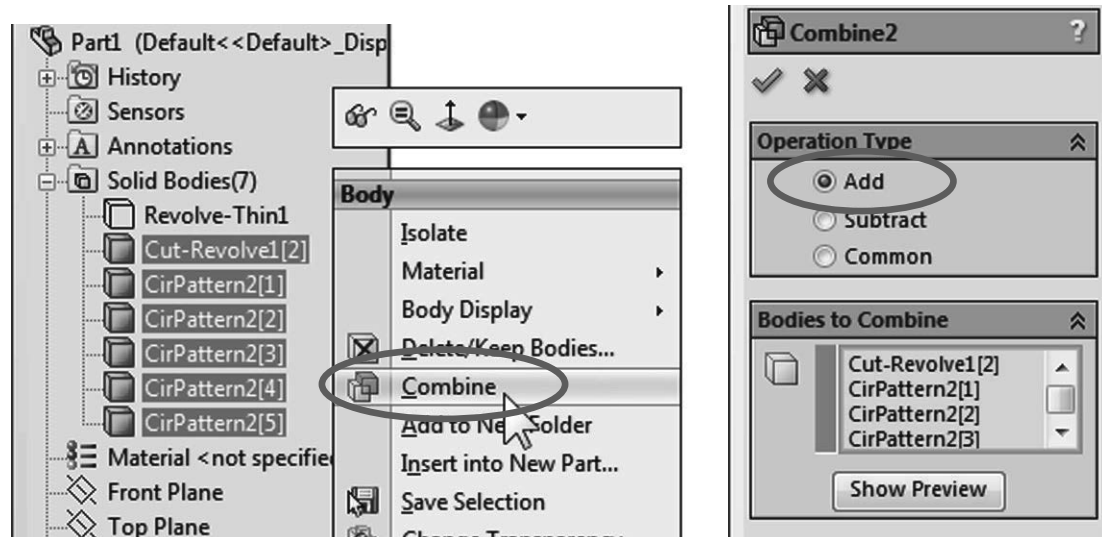
23. - Make a "Revolved Cut" using the top centerline as the axis of revolution, and leave the "Feature Scope" option to "Selected Bodies, Auto-select."



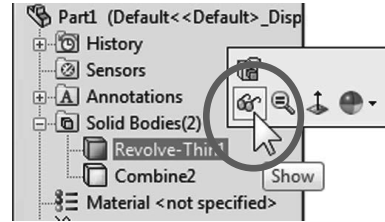
24. - Make a 5 copy circular pattern of the hole we just finished. In the “Feature Scope” use the “Selected Bodies” option, uncheck the “Auto-select” option and select the remaining four bodies made with the circular pattern and the center body (the first revolved feature we did). Click OK to finish the pattern.



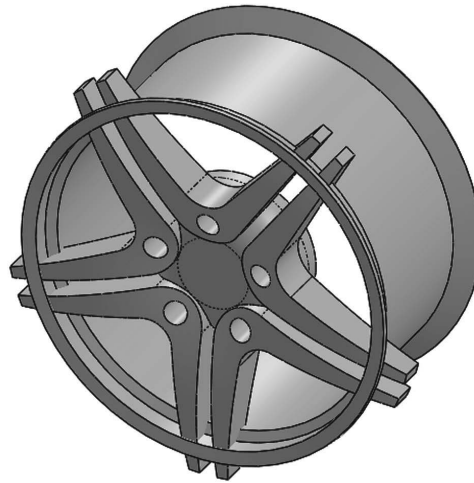
25. - Now that the holes for the bolts are complete we can merge the spokes and the hub into a single solid body. In the “Solid Bodies” folder select the patterned bodies and the hub as indicated, right-mouse-click in any of them and select “Combine” from the pop-up menu, or use the menu “Insert, Features, Combine.” Select “Add” in the “Operation Type” and click OK to finish.



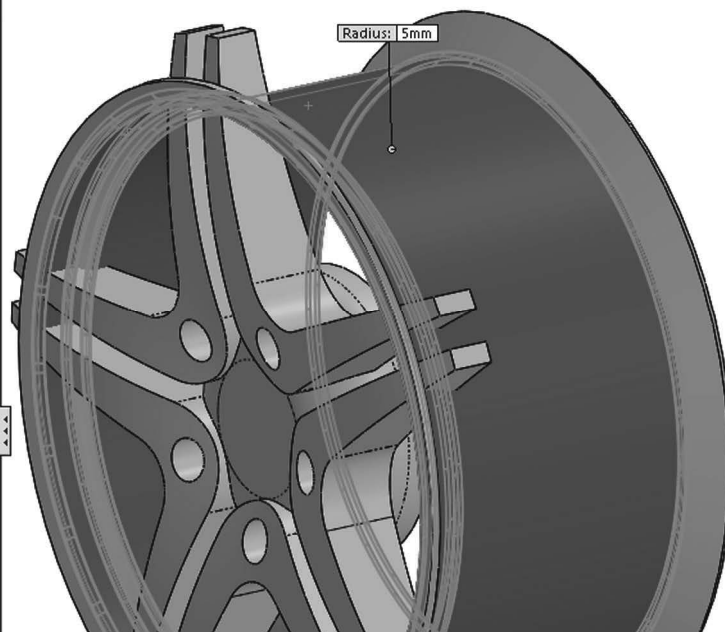
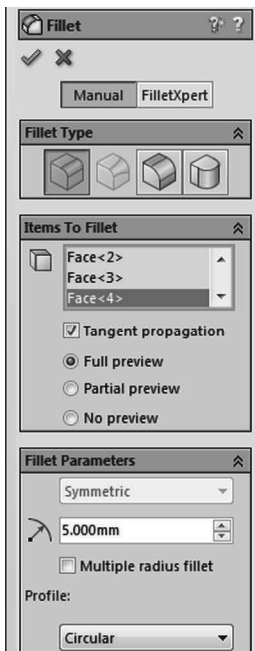
26. - After adding the bodies in the previous step we have two bodies in the part. One is the hub with the spokes and the other (hidden up to this point) is the wheel itself. In the “Solid Bodies” folder show the hidden body. Select it and click in “Show” from the pop-up menu.



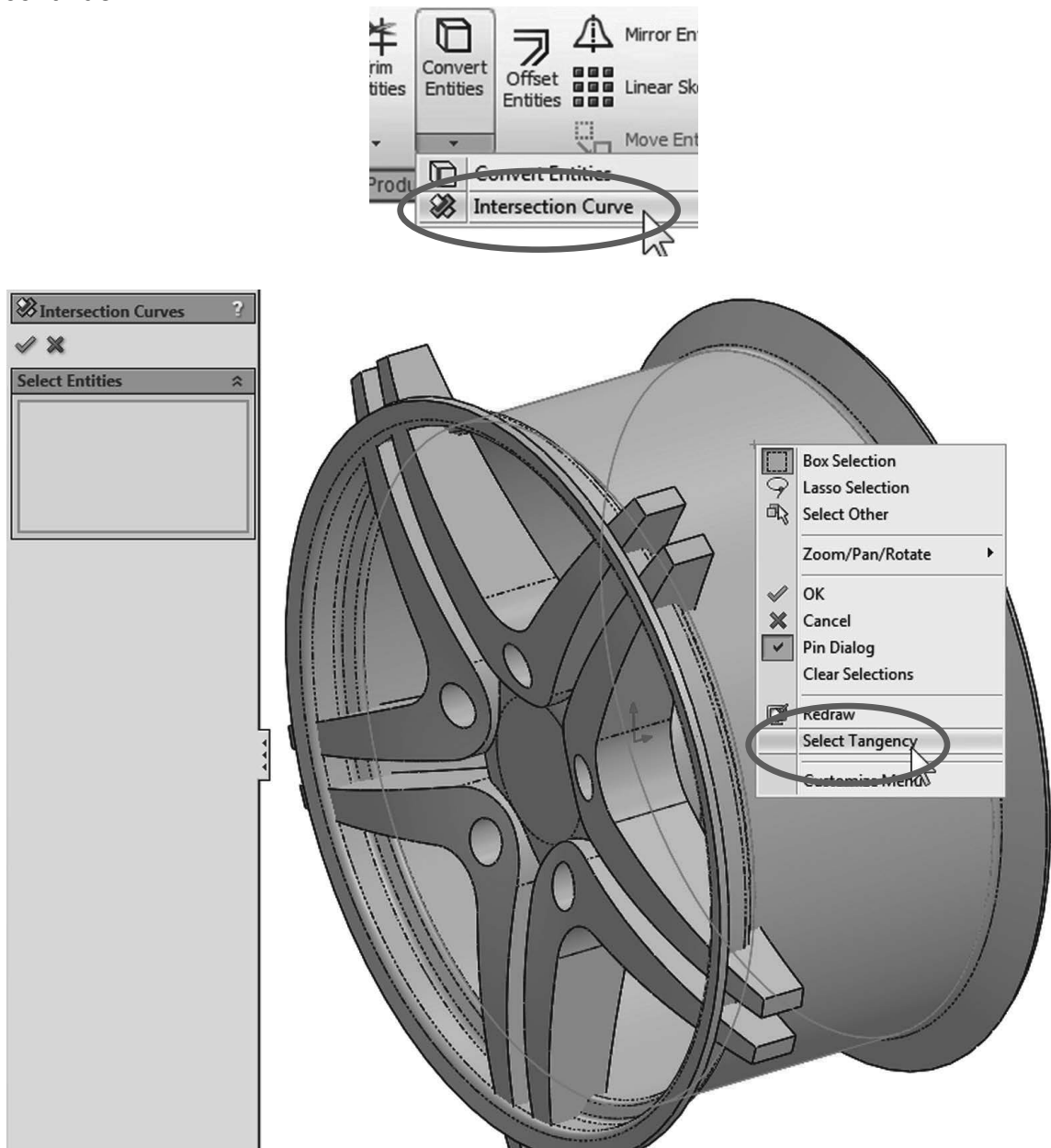
To hide or show a solid body we can select any feature that modified it in the FeatureManager and use the same “Hide/Show” command from the pop-up toolbar, not just in the “Solid Bodies” folder.



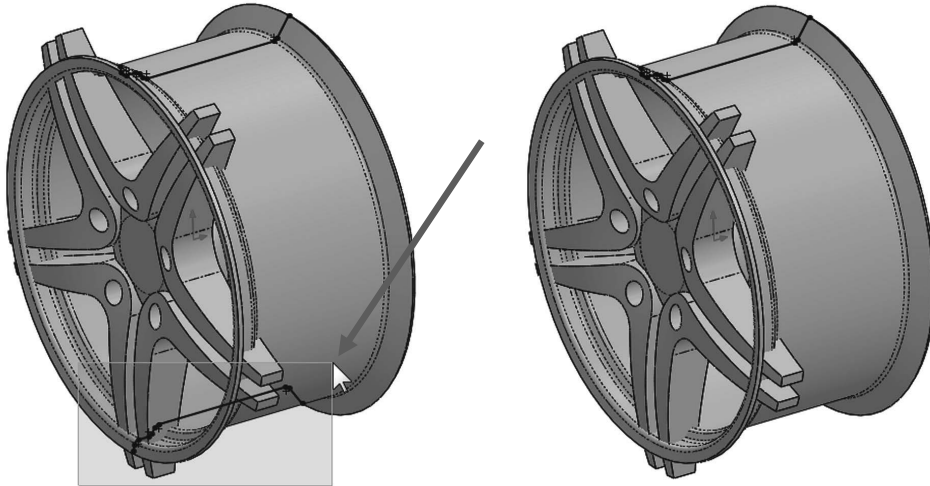
27. - To make the rim better looking add a fillet to round off the sharp edges of the “Revolve-Thin1” feature. Add a 5mm to all of the inside and outside sharp edges. All the edges can be rounded by selecting four faces.



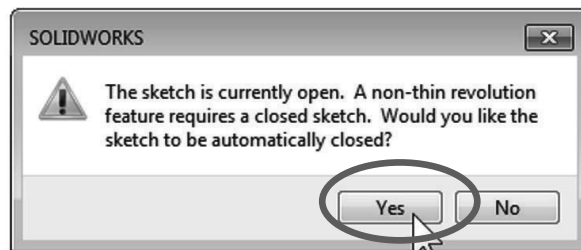
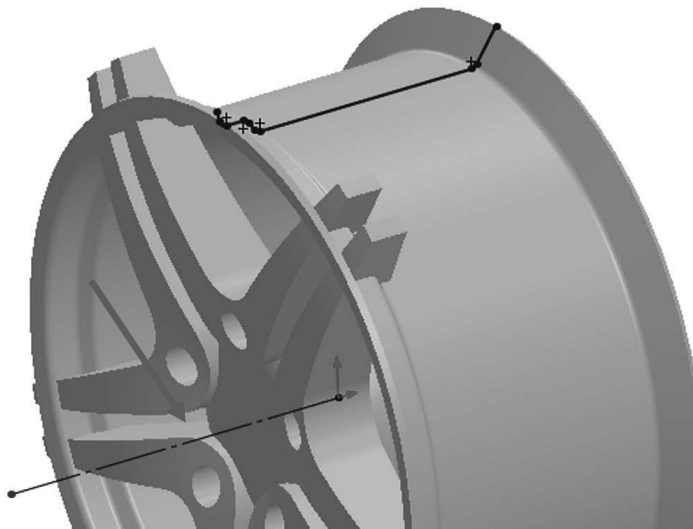
28. - The next step is to trim the excess material from the spokes that protrude beyond the rim. We'll use a "Revolved Cut" to trim it. Using the "**Intersection Curve**" command we can create a sketch that matches the profile of the rim to cut the spokes. Add a new sketch in the "*Right Plane*" and select the "**Intersection Curve**" command from the "**Convert Entities**" drop down icon, or the menu "**Tools, Sketch Tools, Intersection Curve.**" Select all the faces outside of the rim (or inside, in this case selecting either side will give the same result) using a right-mouse-click in any face, and select "**Select Tangency**" from the pop-up menu. This way the selection will propagate through the faces until no tangency is found. (The fillet we added in the previous step created the tangent faces to propagate.) After selecting the faces click OK to close the command and continue.



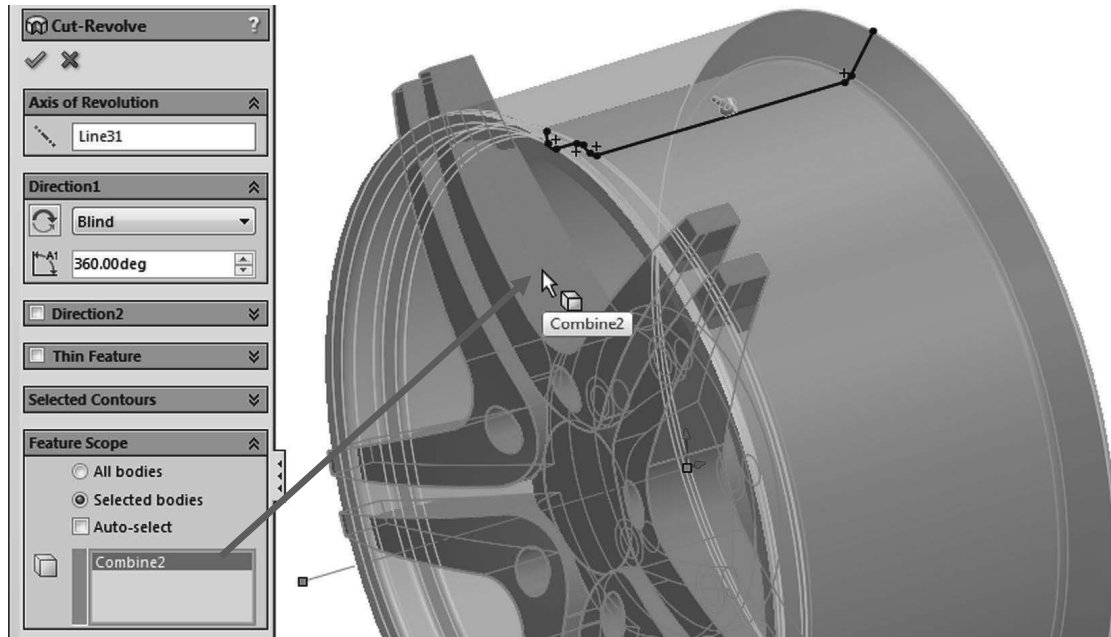
29. - After completing the intersection curve, sketch entities are created where the selected faces cross the sketch plane. Since the faces cross the plane in two different places (top and bottom) we get two profiles. Window-select the bottom profile since we only need the top one for this feature.



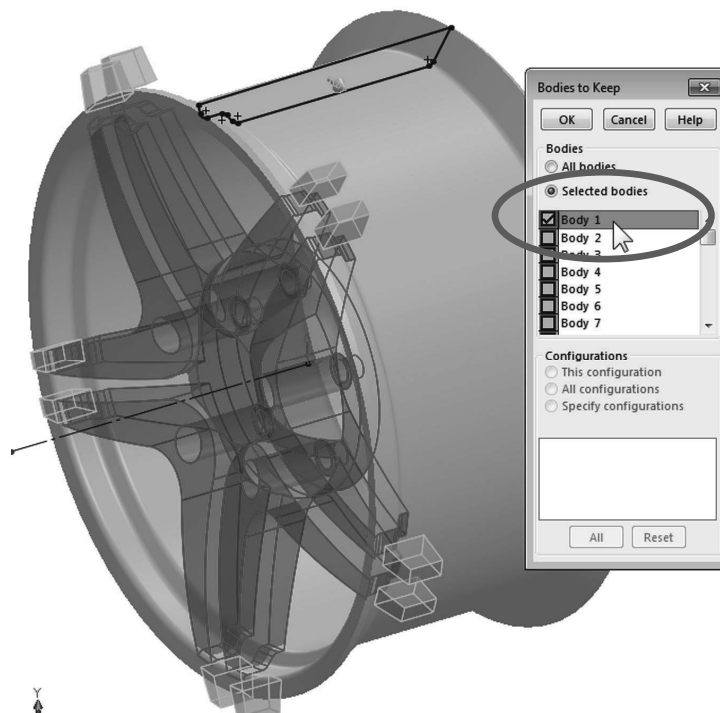
30. - Add a horizontal centerline at the origin and make a “**Revolved Cut.**” We’ll get the message alerting us that the sketch is open and asking if we want to close the sketch. In this case we want to create a closed profile and select “Yes” when asked. The model is displayed using “Shaded” (No edges) for clarity.



31. - In “Feature Scope,” turn off “Auto-select,” and since we only want to cut the spoke body (“Combine2”), add it to the selection list and click OK to continue.



32. - After pressing OK to make the revolved cut, the profile we used generates more bodies by trimming the tips of the spokes, and now we are asked which bodies we want to keep. In the pop-up selection box use the “Selected Bodies” option, and pick the body at the center of the spokes to keep it. You will be able to see the preview before selecting it. Click OK to finish.

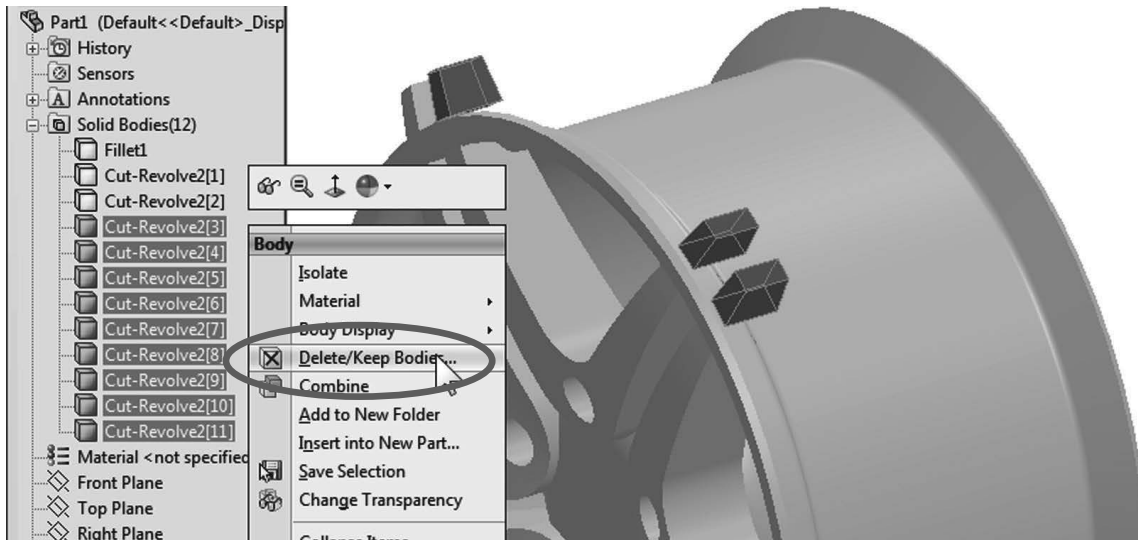




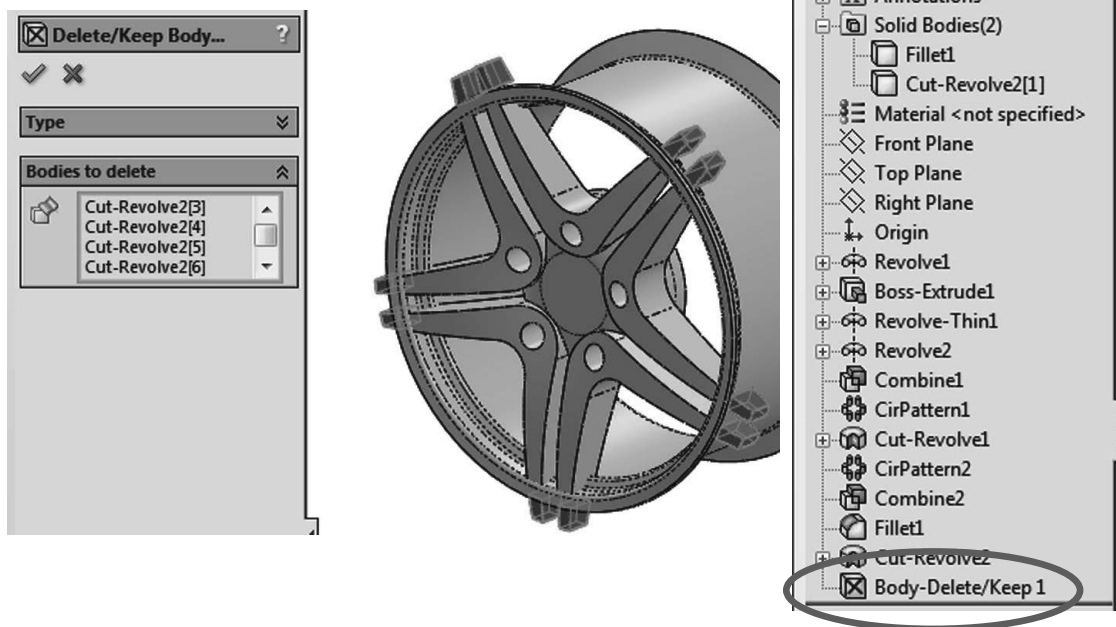
When a cut operation splits a part into multiple bodies, the largest body is usually the first one listed in the “Bodies to Keep” list.



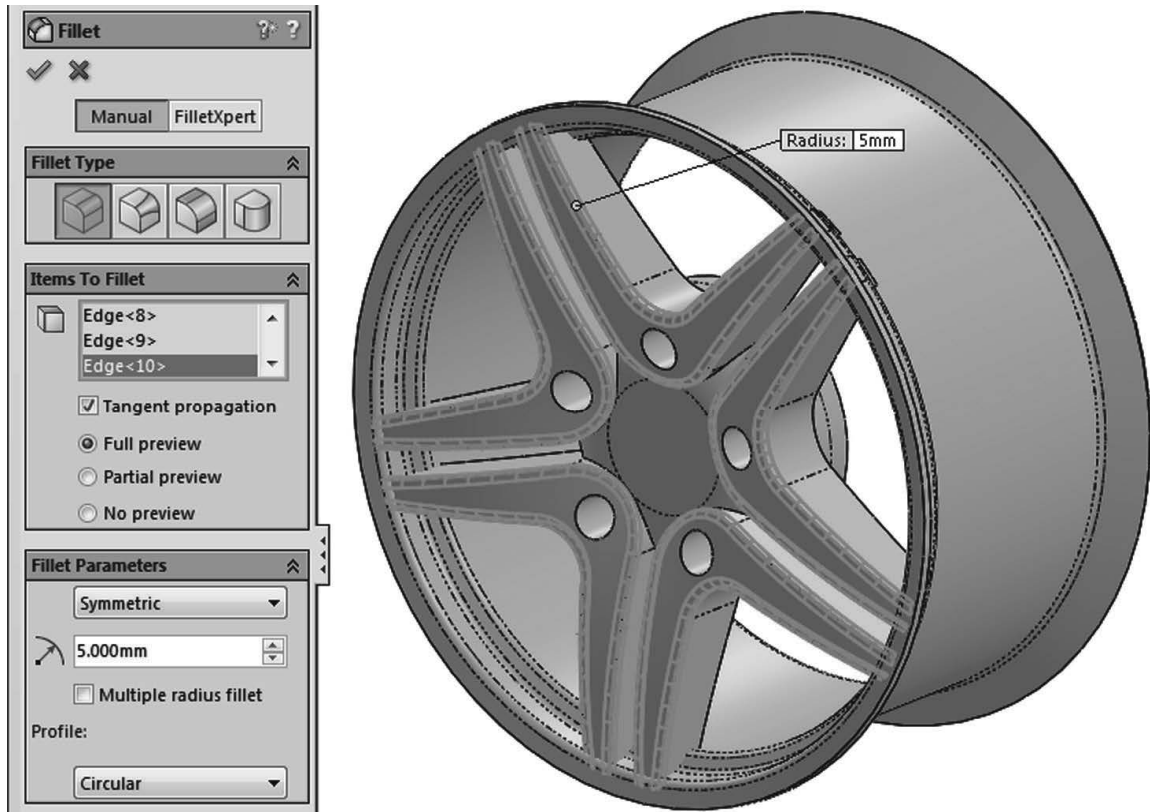
If we had kept all the bodies, we'd have a large number of bodies in the “Solid Bodies” folder. In that case we can delete the unwanted bodies by right-mouse-clicking on them and selecting “Delete Bodies.”



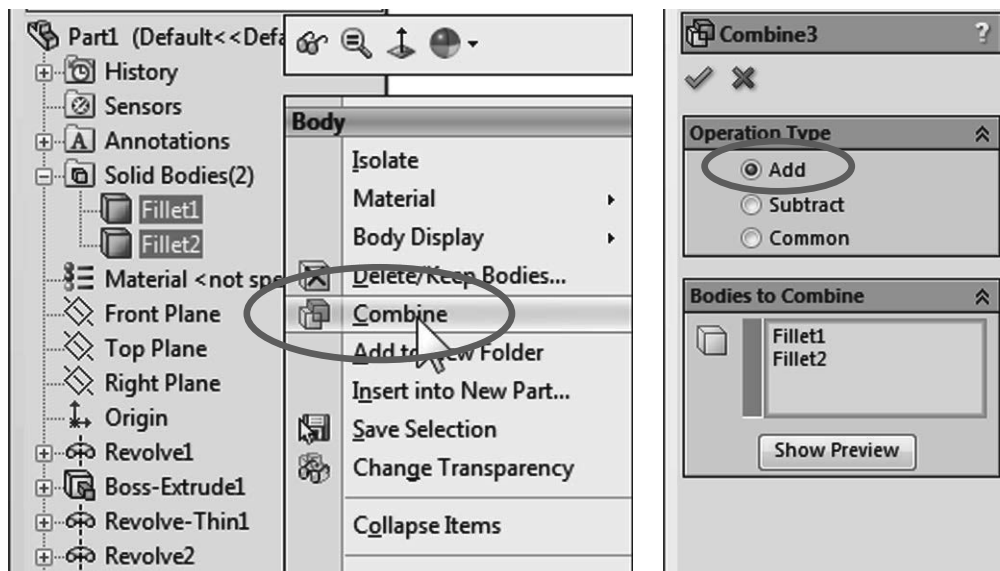
We are asked to confirm which bodies we want to delete. After we click OK to delete them, a new feature is added to the FeatureManager called “Body-Delete.”



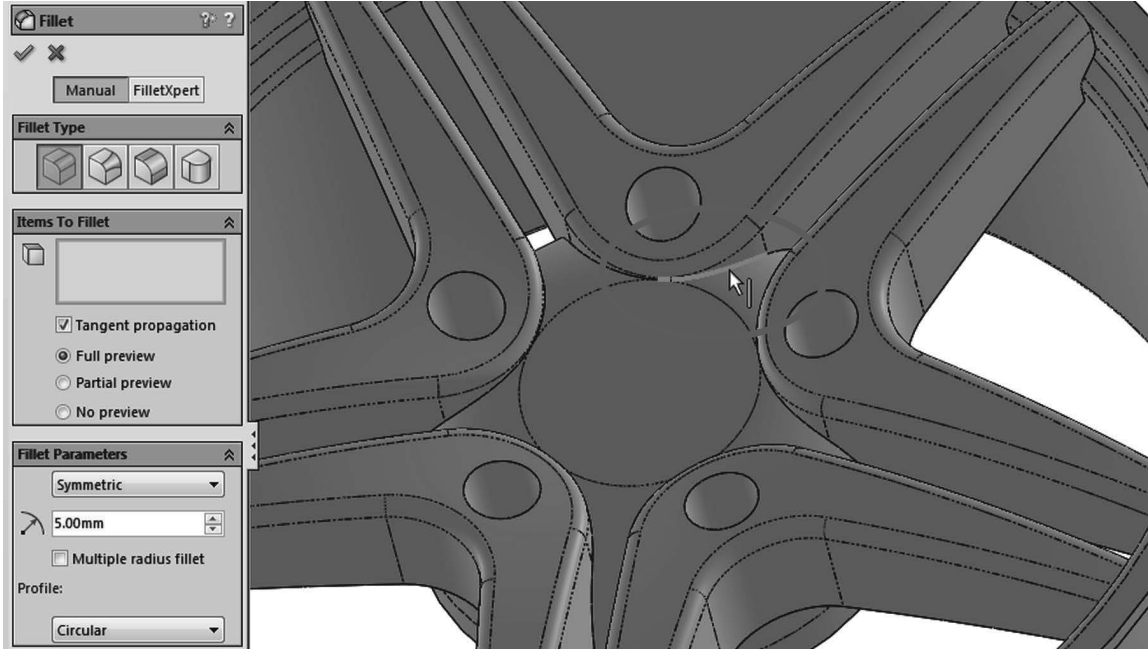
33. - Add a 5mm fillet to the spoke arms selecting the radial edges of the spokes and click OK to finish.



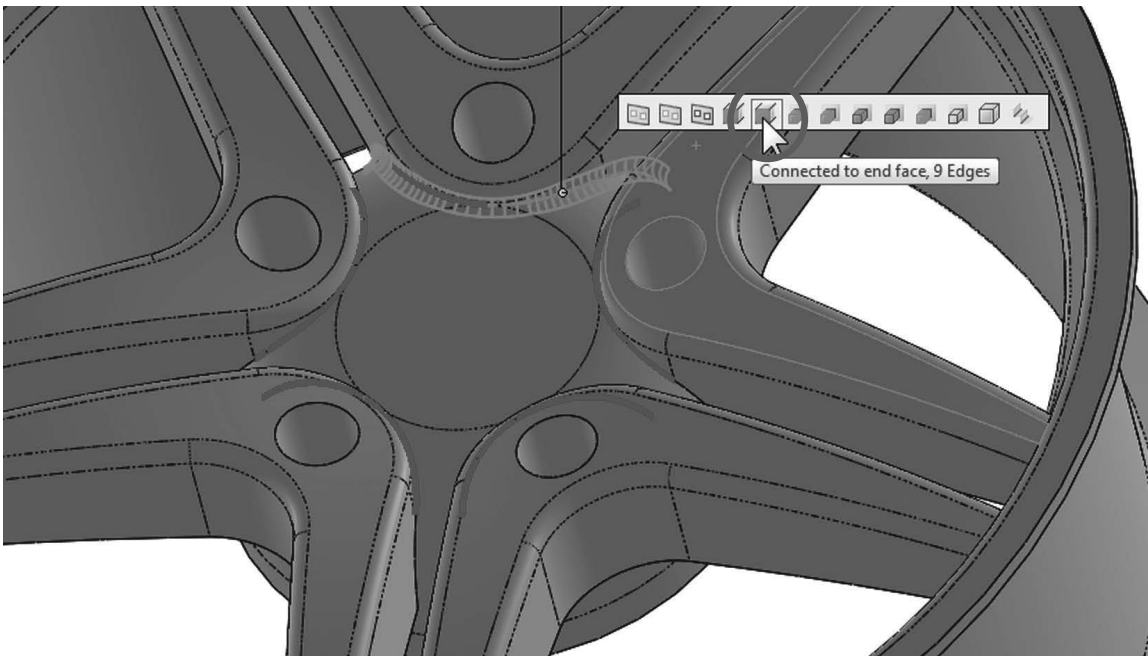
34. - Select the remaining two bodies and use the “**Combine**” command to add them together. In the “Operation Type” select “Add,” make sure we have both bodies selected, and click OK to finish. After combining the last two bodies, the “Solid Bodies” folder goes away and we have a single body left.



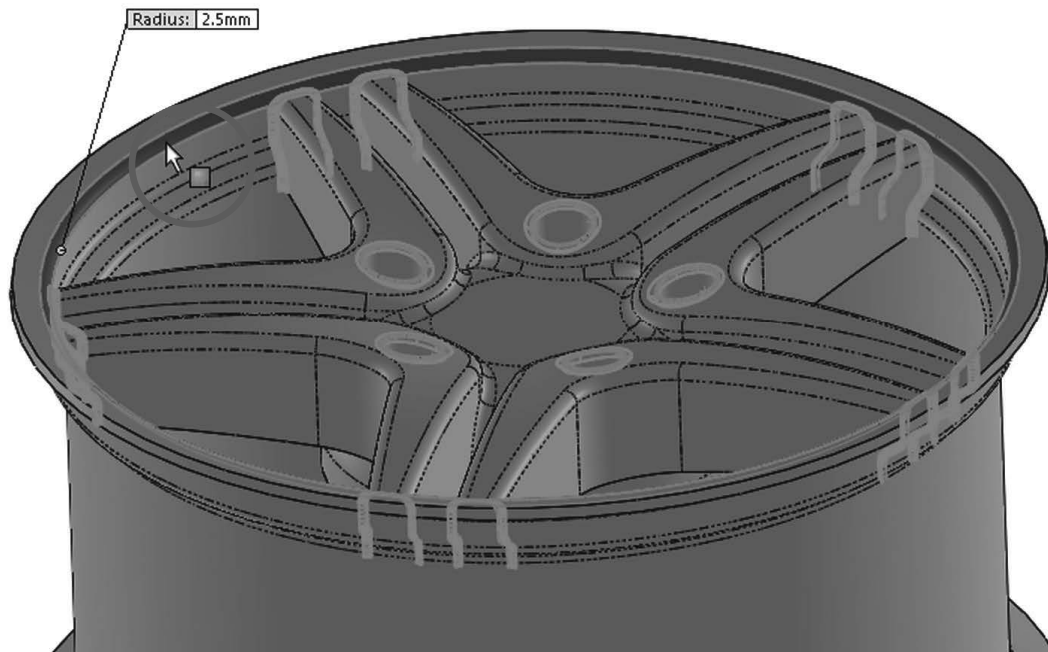
35. - Add a 5mm fillet to the edges in the center. After selecting the edge indicated, we see the expanded selection toolbar. Click in the “Connected to start face” icon to automatically select all the edges needed. Click OK to finish the fillet.



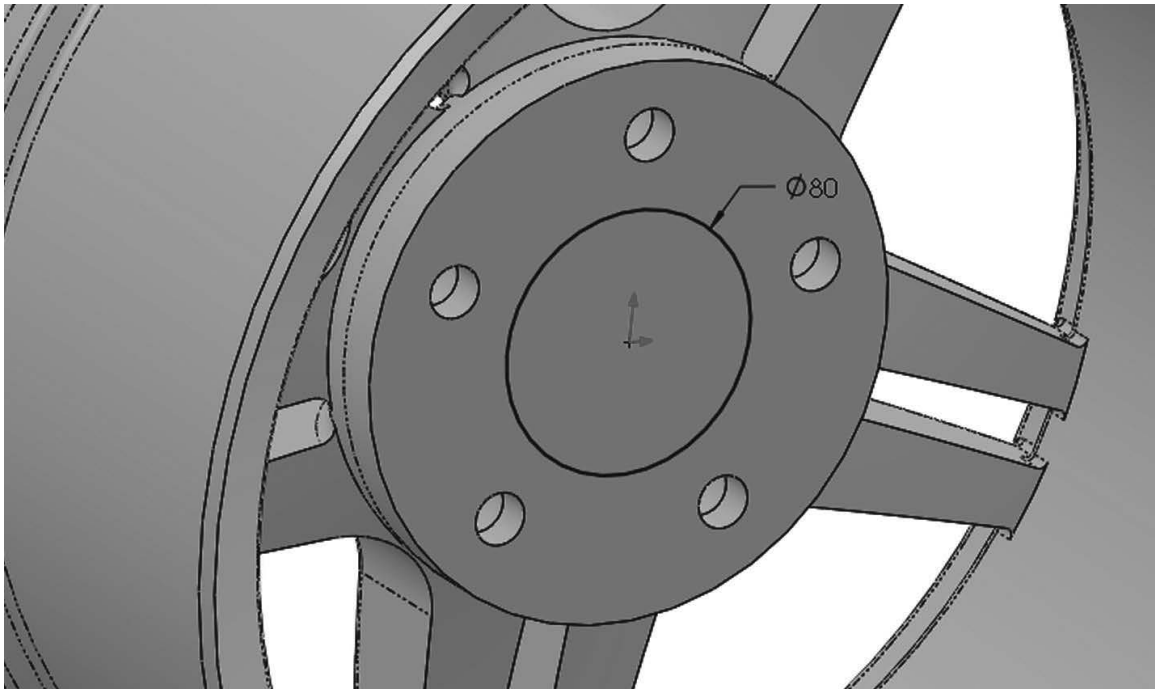
When we move the mouse over each of the icons in the selection toolbar we can see the edges that would be selected if we clicked on it. This is an easy way to make the selection of multiple edges easier.



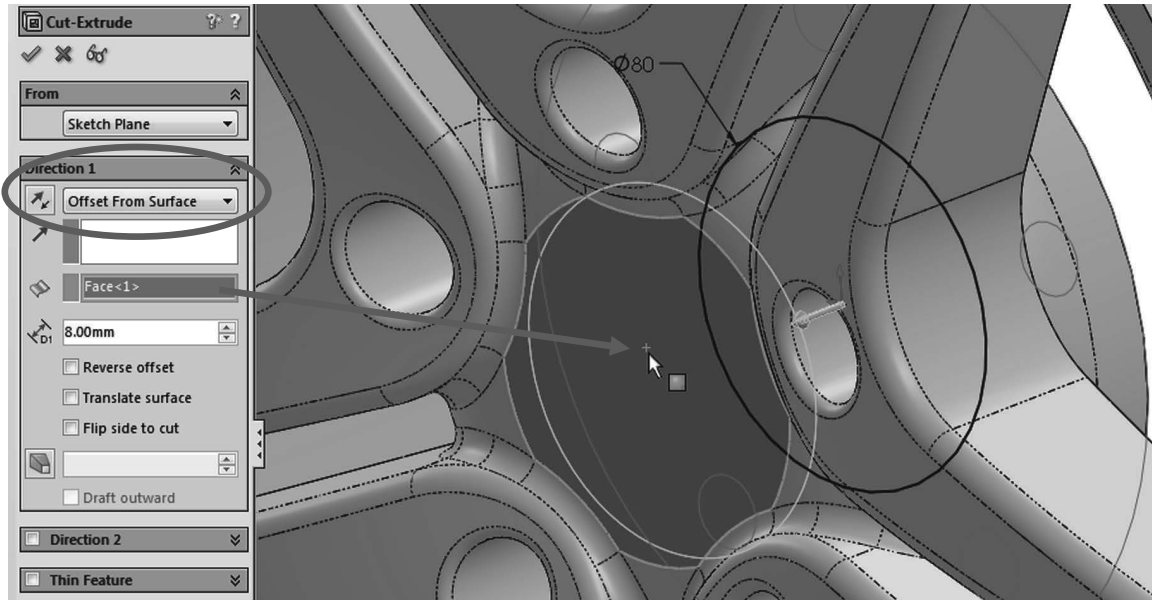
36. - Add a new 2.5mm fillet to the bolt holes and the connections between the spokes and the rim. Use the selection toolbar if needed.



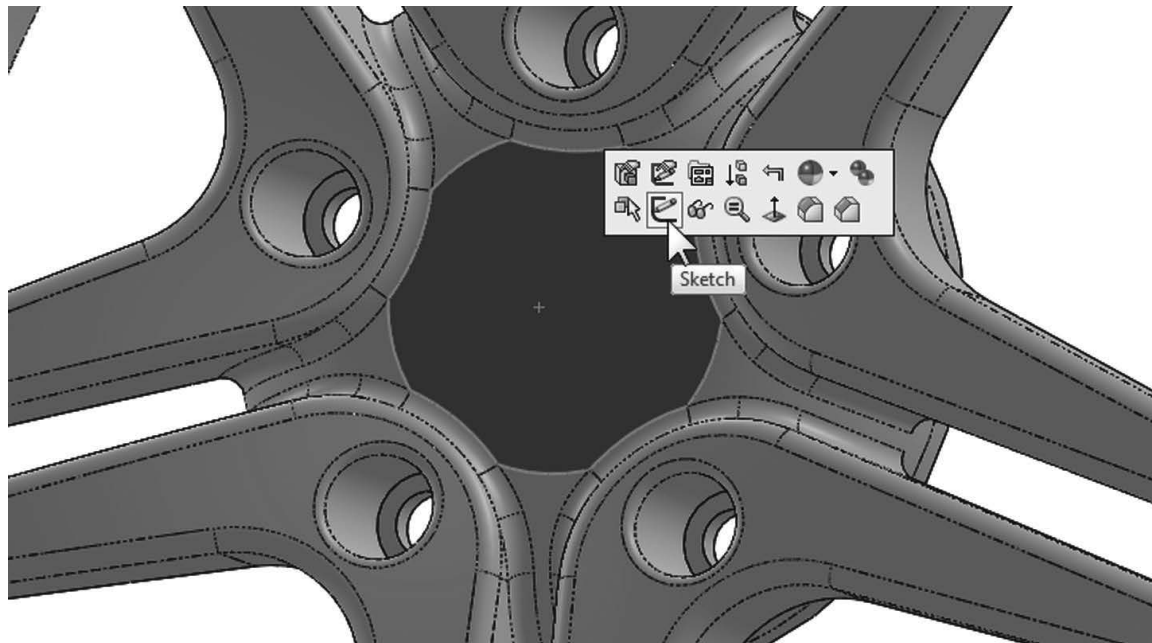
37. - To make the wheel lighter, add an 80 mm diameter hole in the rear of the hub. We want to make the cut deep enough to leave 8mm thick at the other side. To accomplish this we'll use an end condition called "Offset from Surface."



Select the **“Extruded Cut”** command. Use the **“Offset from Surface”** end condition and select the face on the other side of the cut. Enter 8mm in the distance box. In essence, the cut made will be 8mm away from the selected face. The **“Reverse offset”** option allows us to measure the distance to one side of the selected face or the other. Rotate the view to select the face if needed and click OK to finish.



38. - As a finishing touch we'll add a logo to our design. In SolidWorks we can add a picture or image to our models for appearance or presentations. From the files included locate the image file 'logo.png' for use in this step.

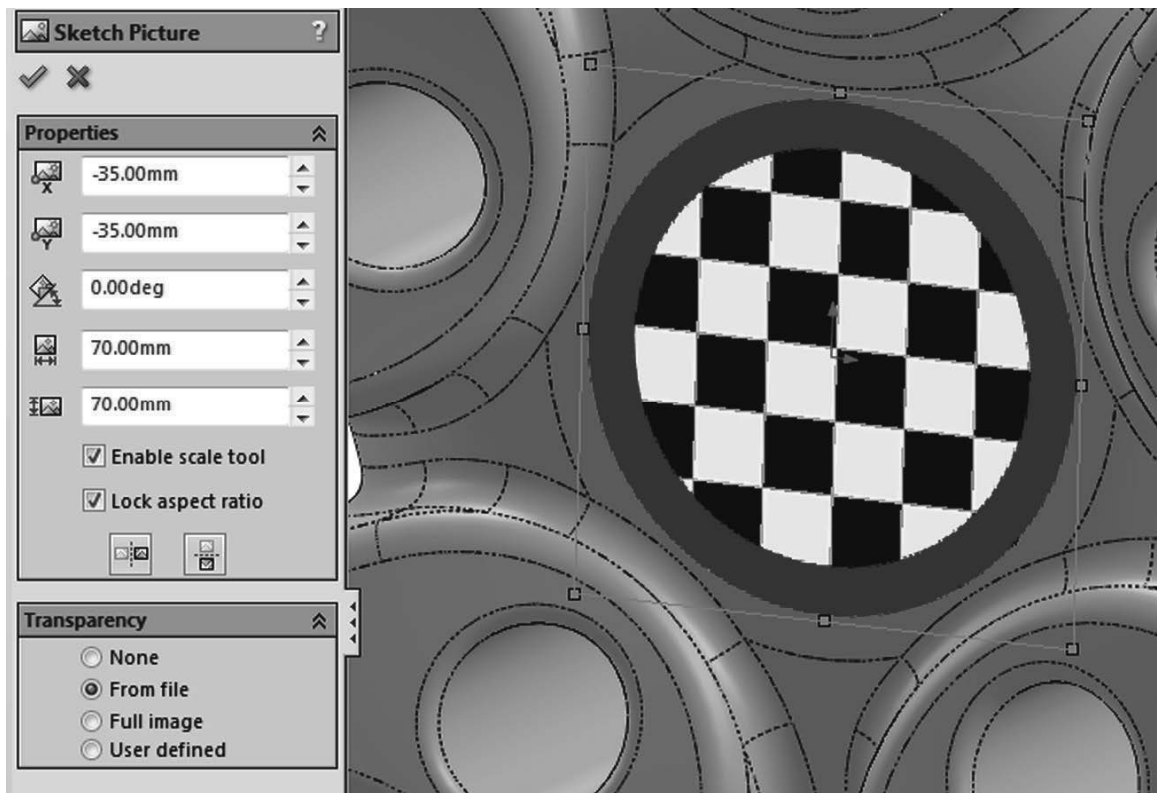


Add a new sketch in the front face of the wheel, where the logo will be located. Go to the menu “**Tools, Sketch Tools, Sketch Picture.**”

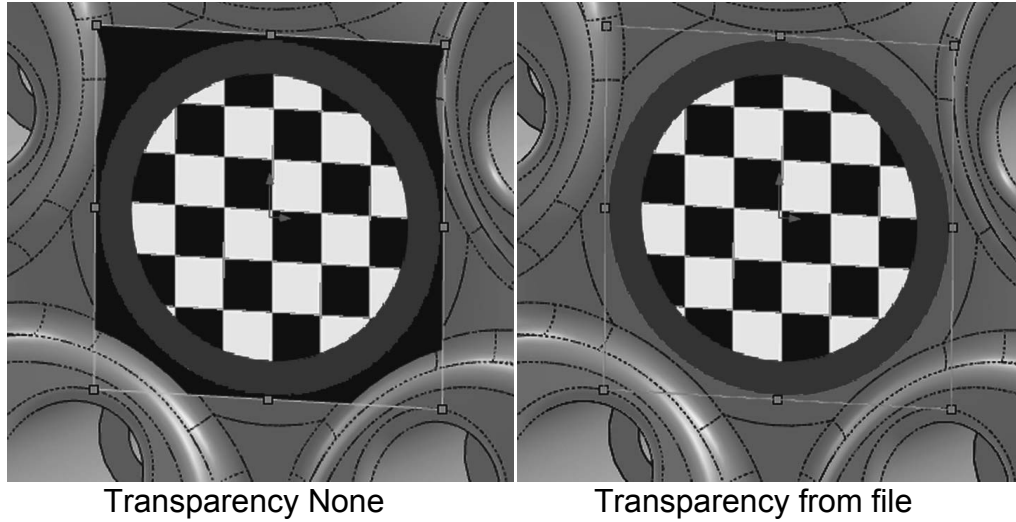
Browse to the location where the logo.png image was saved and select it. The image is added to the sketch and we are ready to adjust its properties. We can click and drag the corners and the image itself to locate and resize, or we can enter the image values in the “Properties” box. For this image, change the size to 70mm wide by 70mm height, and to locate it, move its center 35mm to the left (-35mm) and 35mm down (-35mm). For this image select the properties shown for size and location.

Another property we can set is the image transparency. We can make the entire image transparent (Full image), use the transparency saved in the image file (From file), or select a color from the image to make it transparent (User defined). Since the image provided has transparency saved with it*, select the “Transparency” option “From file” and note the image updating in the screen.

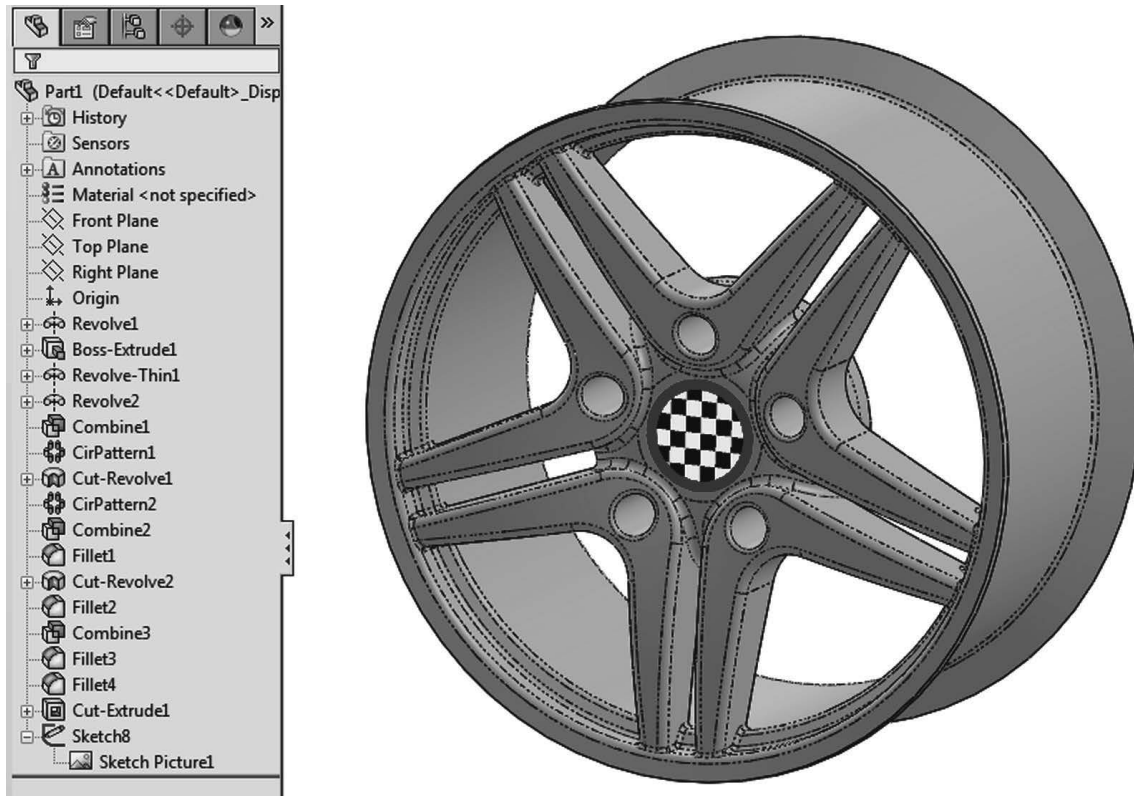
** File types that support transparency are GIF, PNG and TIFF. Transparency can be defined with standard image editing software.*



After the picture is added to the sketch it can be edited by double clicking on it in the FeatureManager.



39. - Set the transparency to “From file,” click OK to finish the image properties and exit the sketch. In the FeatureManager the image is absorbed by the sketch, and if needed, it can be hidden by hiding the sketch.

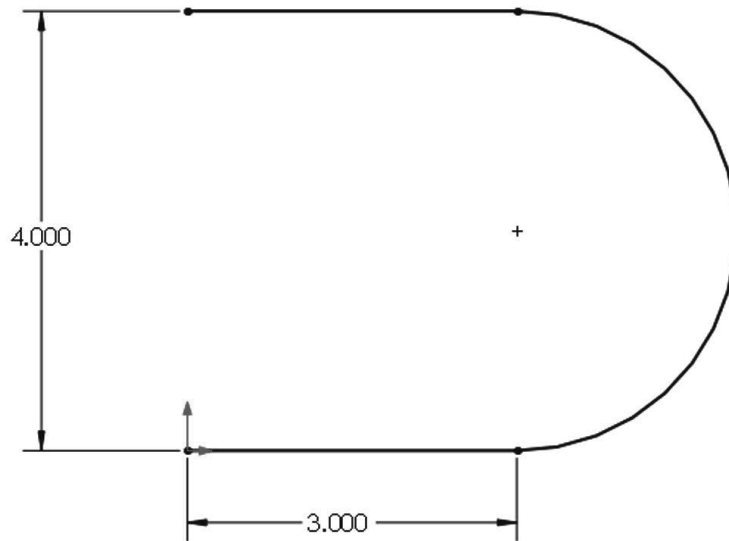


Feel free to add your favorite automobile logo. Due to copyright restrictions we cannot use registered logos in this book, but we can confirm that there are some that look VERY good!

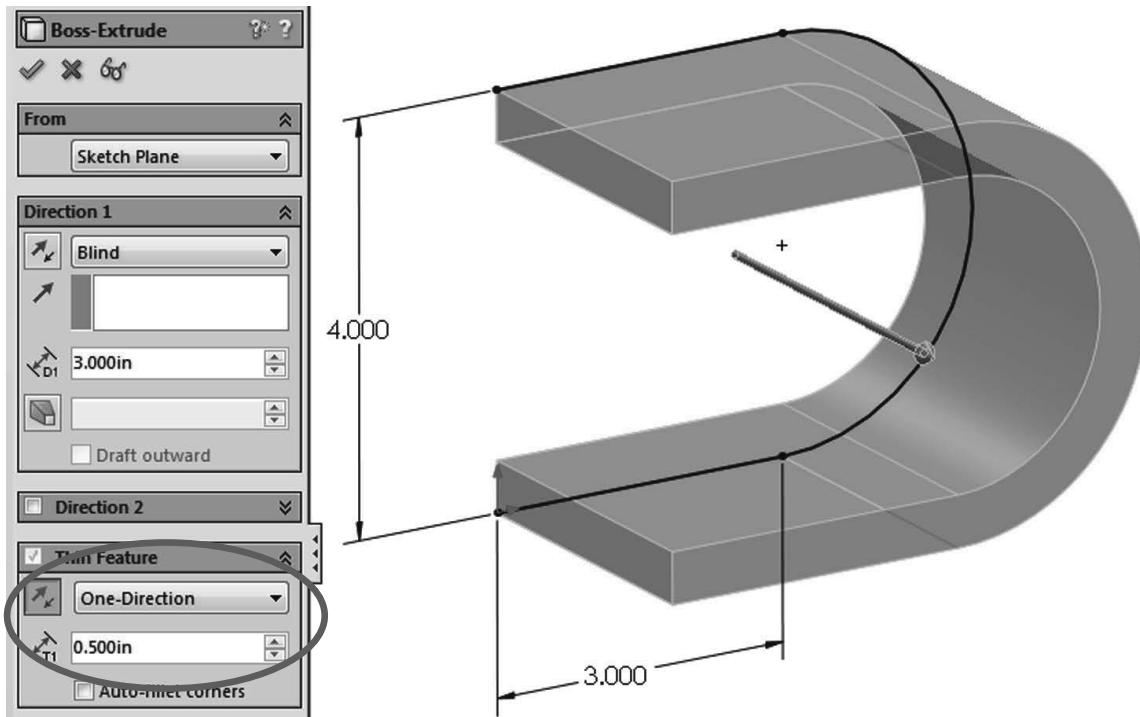
This is a good example to show how to work with multi body parts and perform local operations. Save the part and close it.



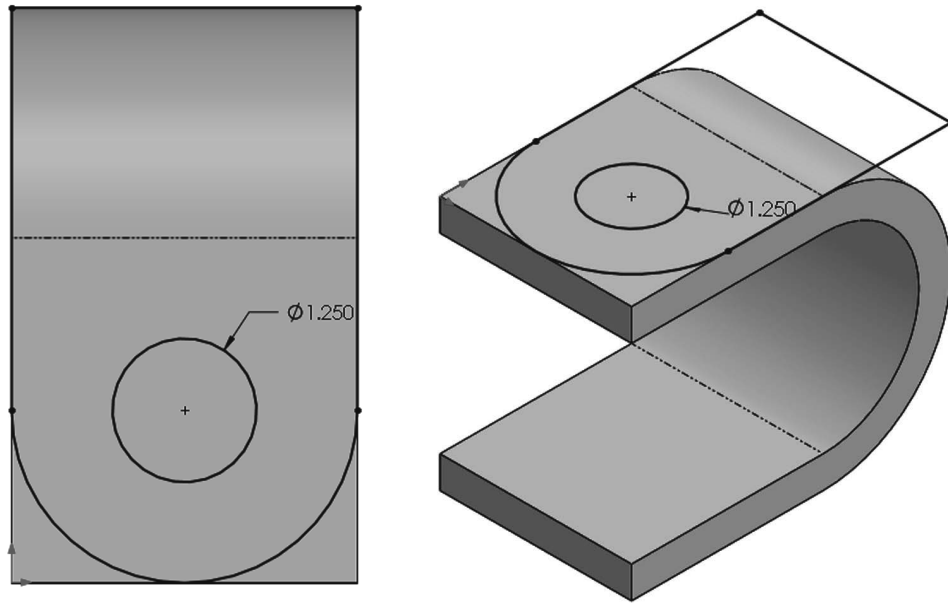
40. - The next area to cover in the multi bodies topic will be to learn more about combining bodies. The other two operations we can do when combining bodies is to subtract one or more bodies from another, or get the common volume between them. As we saw in the previous example, to obtain the common volume between bodies we need two bodies that intersect. Make a new part and draw the following sketch in the "Right Plane."



41. - Extrude the sketch as a Thin Feature. The extrusion's depth will be 3" and the thickness of the part 0.5" inside.

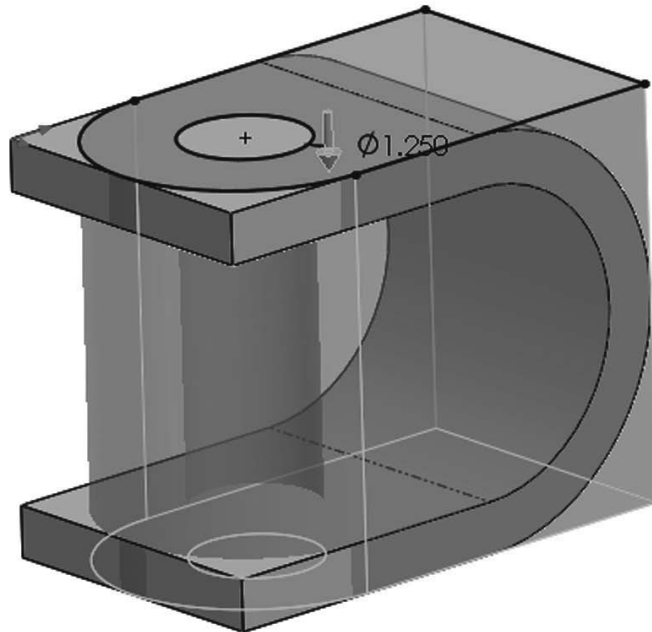
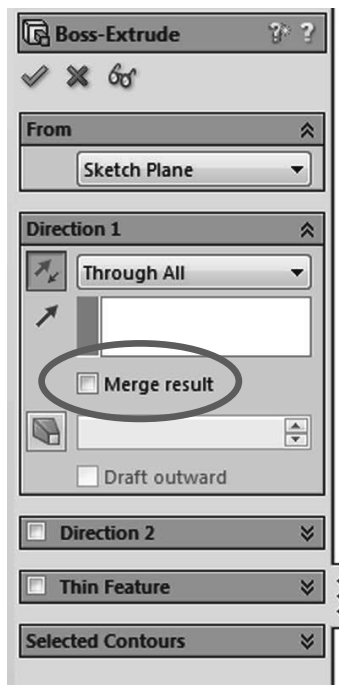


42. - For the next feature, add a sketch in the top face of the first feature. Looking at it from a top view should look like this:

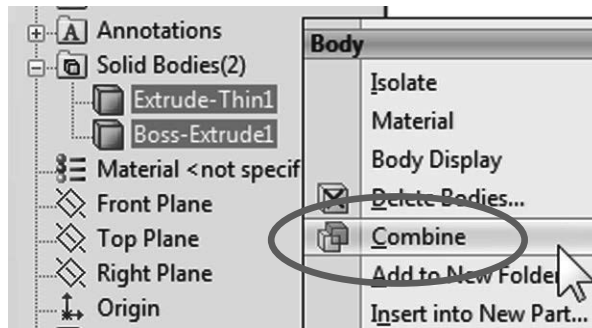


Notice the only dimension we need to add is the hole diameter; everything else is defined with geometric relations (Coincident and Tangent relations).

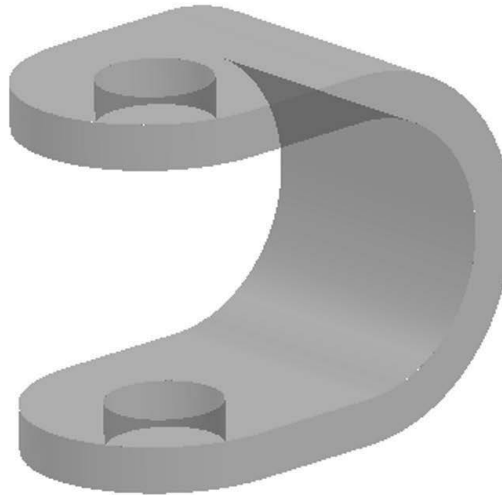
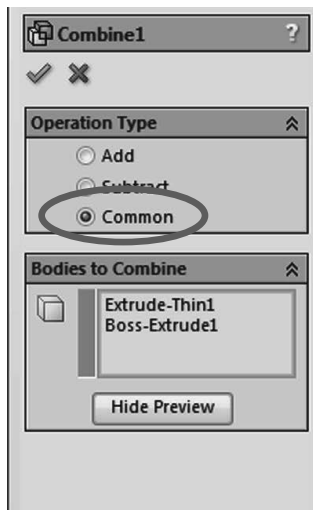
43. - Extrude the second sketch downwards with the **“Through All”** end condition, and uncheck the **“Merge result”** checkbox. We want to have two overlapping bodies.



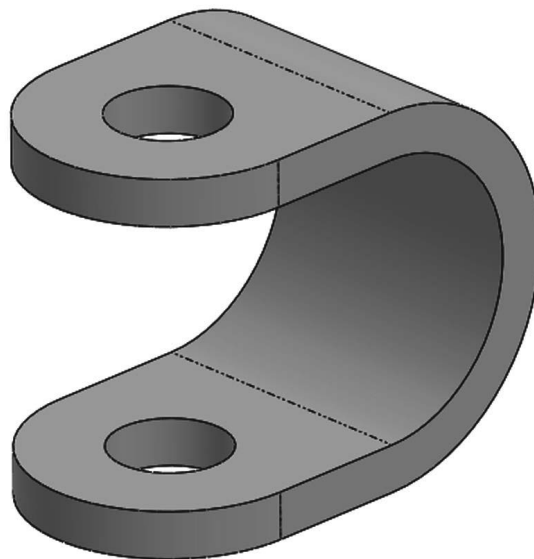
44. - Now that we have two separate bodies, we can combine them to get the common volume. Select the menu “**Insert, Features, Combine**” and select both bodies, or from the “*Solid Bodies*” folder in the FeatureManager pre-select both bodies; right-mouse-click and select “**Combine**” from the pop-up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



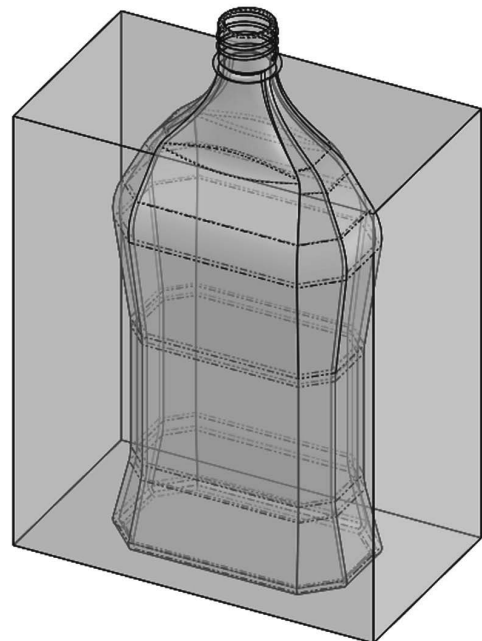
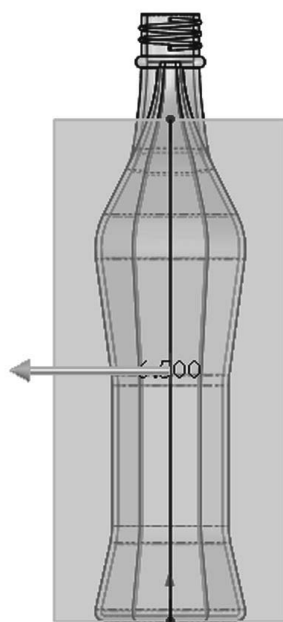
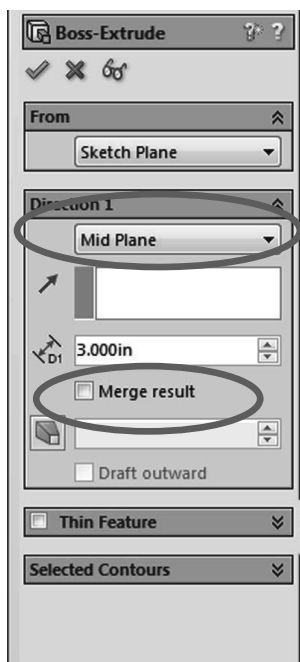
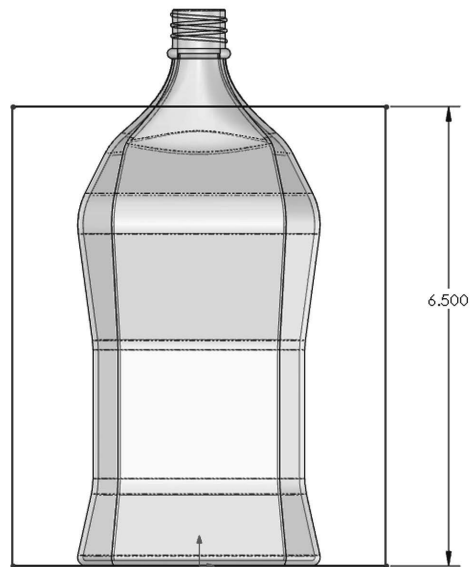
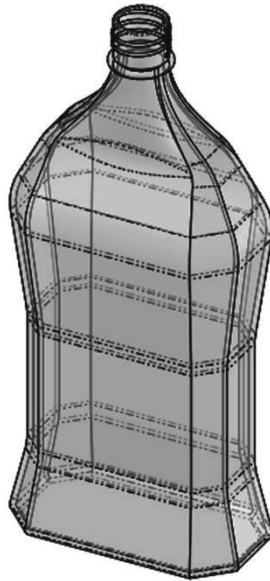
45. - In the Combine operation select the “Common” option under “Operation Type.” Click in the “Show Preview” button to see what the resulting body will look like and click OK to finish.



The finished part will look like this.

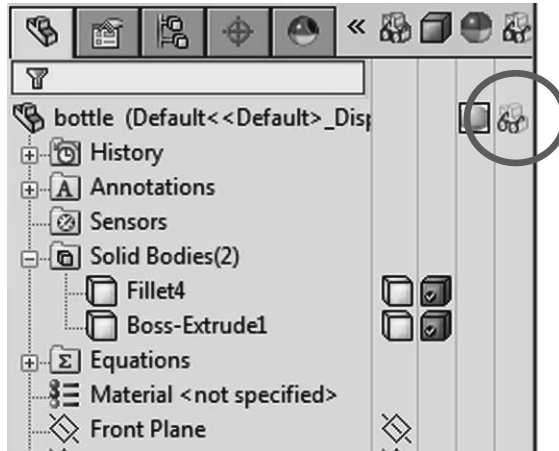


46. - So far we have covered adding bodies and extracting their common volume. The next multi body operation we'll cover is the difference between bodies. One common use for a body difference is to obtain a mold's core and/or cavity (later in the book), but here we'll show how to obtain the volume capacity of an irregularly shaped bottle. Locate the part *'Bottle.sldprt'* from the included files 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. Add a sketch in the *"Front Plane"* and draw a rectangle as shown. Make the sketch and extrusion big enough to cover the entire bottle. Use the *"Mid Plane"* end condition and uncheck the *"Merge result"* option before extruding the new body.

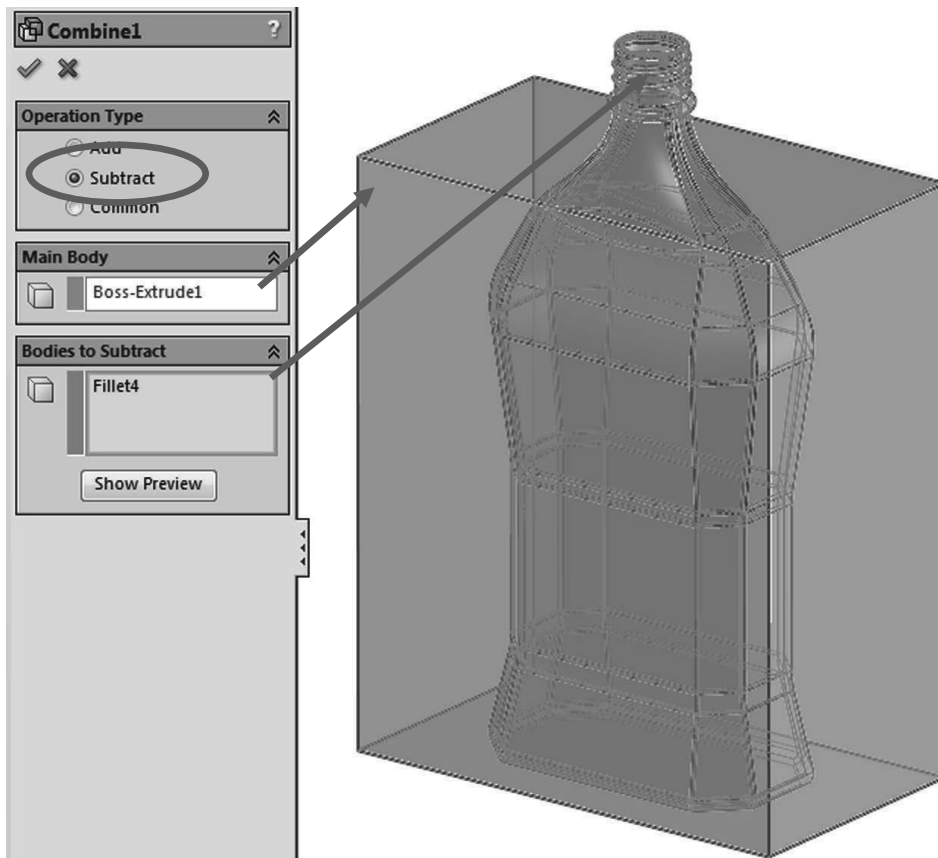




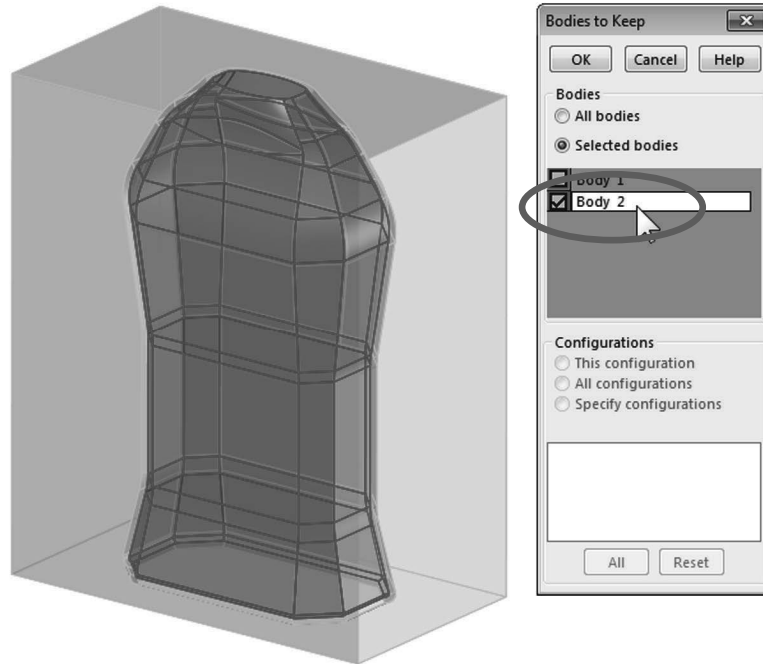
To change a part's transparency use the **“Display Pane.”** Activate the transparency at the part level (can also be done to each body individually).



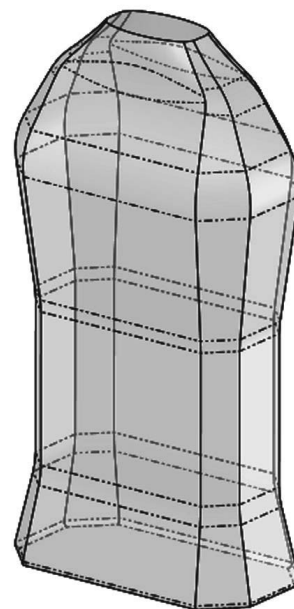
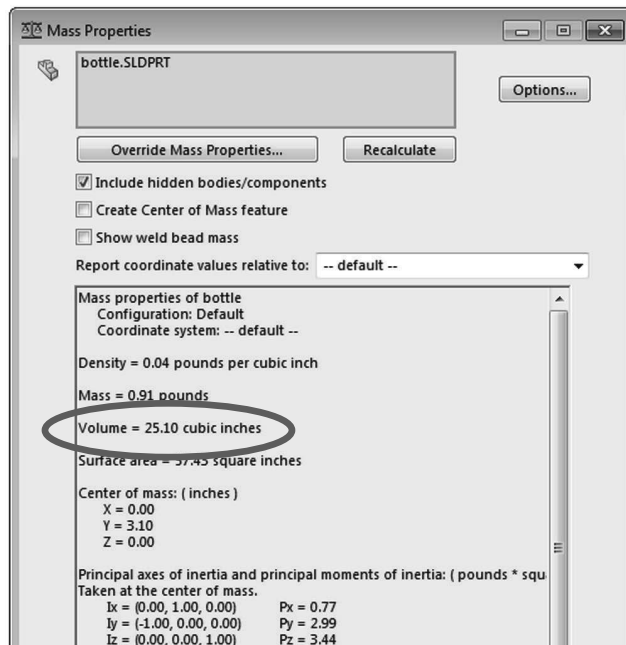
47. - Now that we have the two solid bodies, select the menu **“Insert, Features, Combine.”** Under **“Operation Type”** select the **“Subtract”** option. In order to get a difference, we need to select the body that we want to remove material from (Main Body) and the body (or bodies) that we want to 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 to continue.



48. - After pressing OK we are immediately presented with the “Bodies to Keep” dialog, just like when we trimmed the ends of the spokes in the wheel. There are two bodies resulting from the operation, but we are interested only in the inside body. Select the inside body to keep it and click OK to finish.



49. - 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: