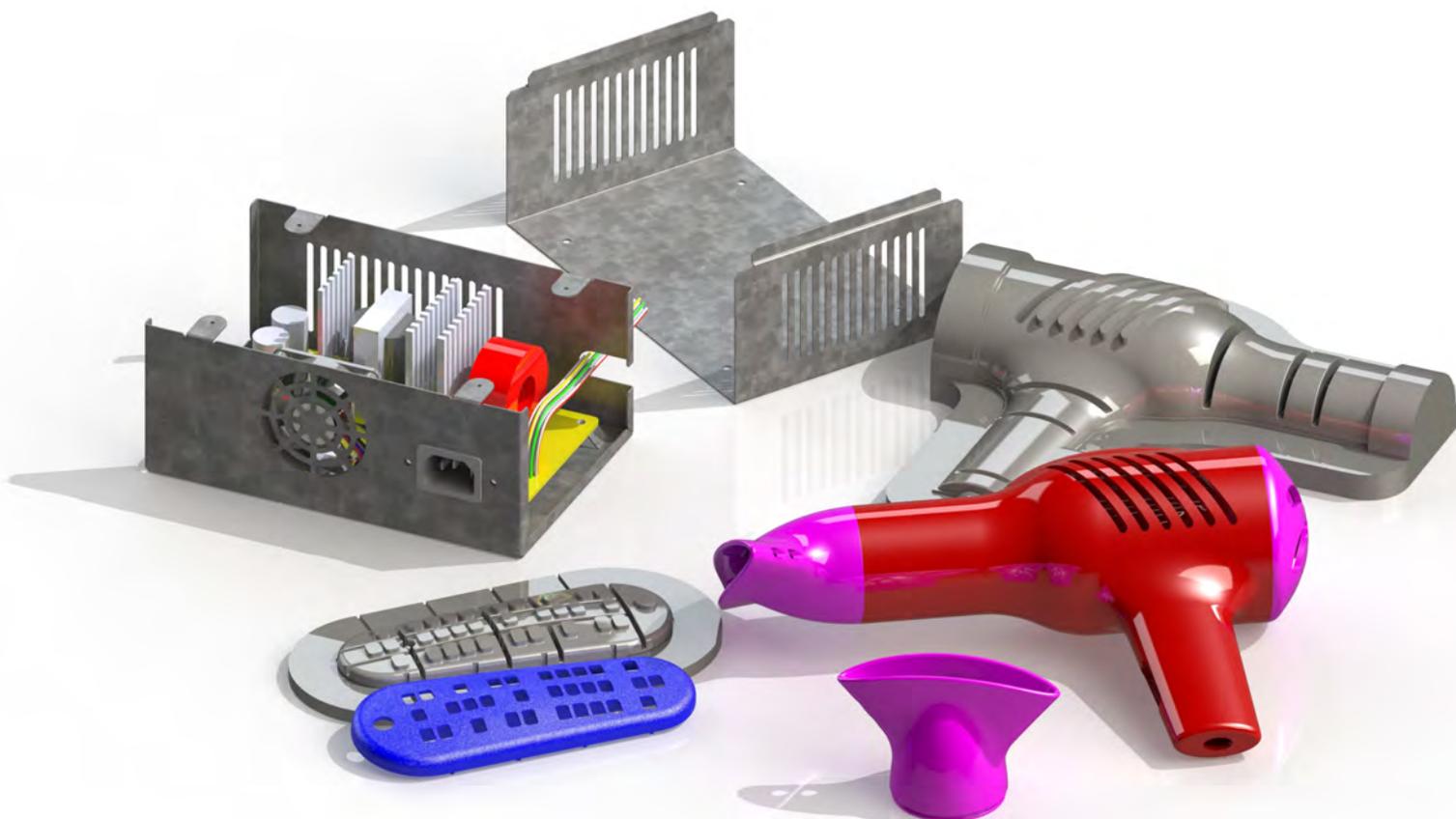


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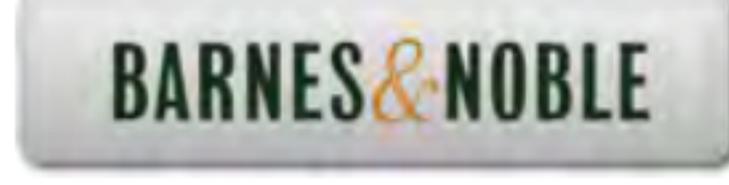
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Multi Body Parts, Sketch Editing and Other Tools

Multi body parts

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” body in a part. Up until now we have been modeling 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 later in this book).

Contour Selection

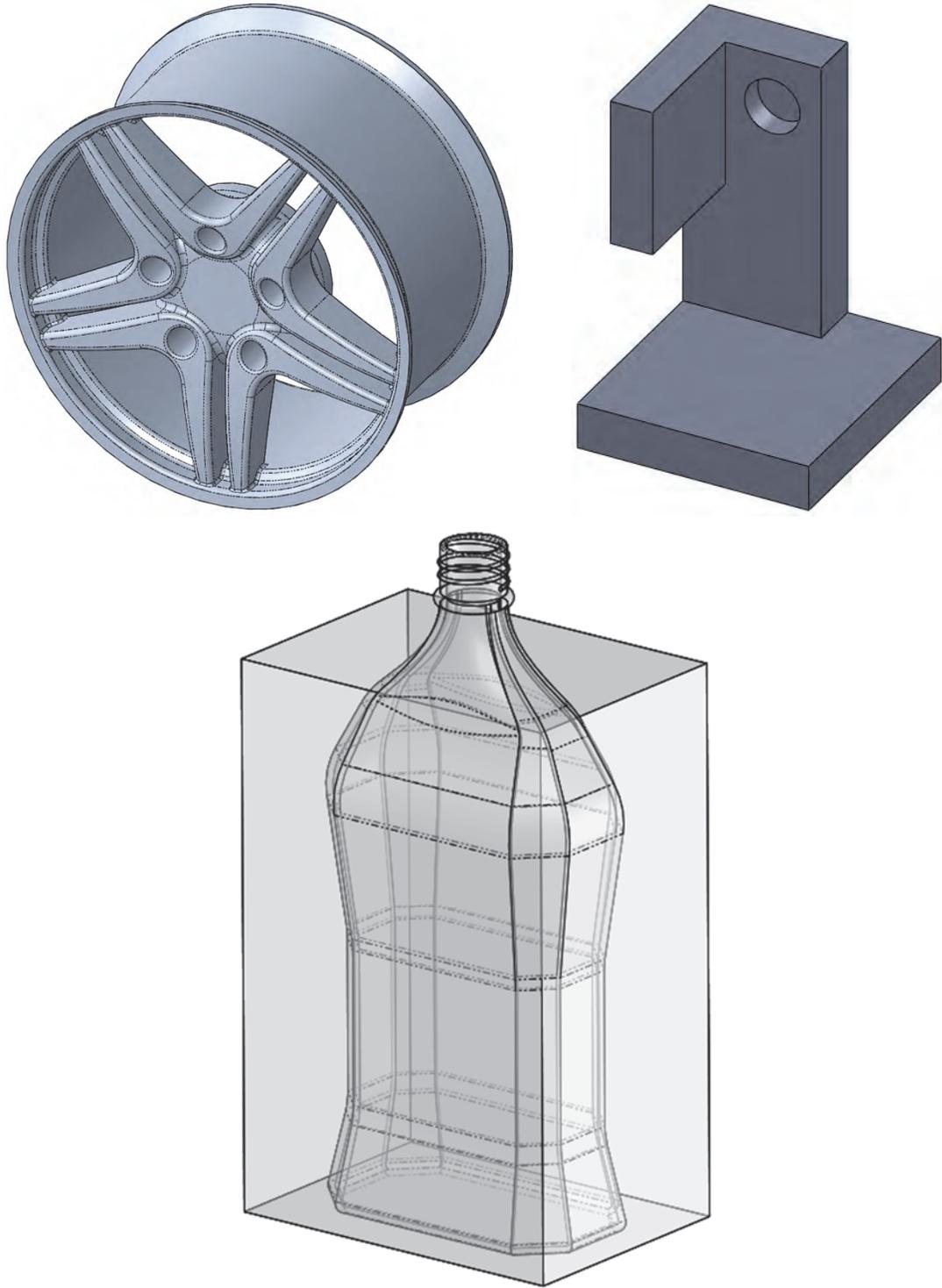
Besides working with single body parts, we have also worked with single and multiple contour open or closed sketches. In this section we'll cover how to use Contour selection to work with sketches that contain multiple entities sharing an endpoint and/or having self-intersecting lines. We'll learn how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

Part Editing

Another topic that is very important to modeling in SolidWorks is editing parts to fix errors or make modifications to components. After all, when we are designing, at some point we will most likely have to go back and make changes to our design, and sometimes those changes can cause trouble in subsequent features. In the Editing section we'll cover different options available when editing parts, as well as fixing and recovering from errors.

Equations

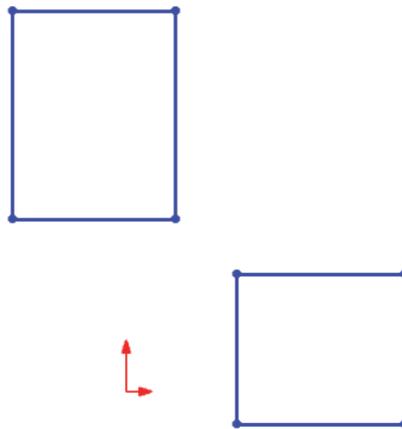
One last topic we'll cover in this section deals with maintaining design intent; we'll learn how to add, edit and delete equations in a part, giving us powerful tools to make our designs more flexible, robust and predictable when making design changes.



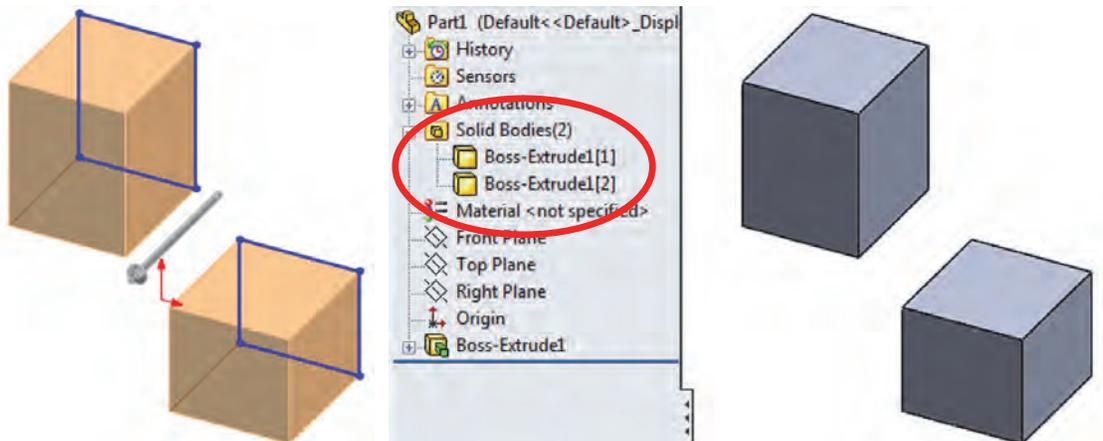
Multi Body Parts

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 pieces (bodies). 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.

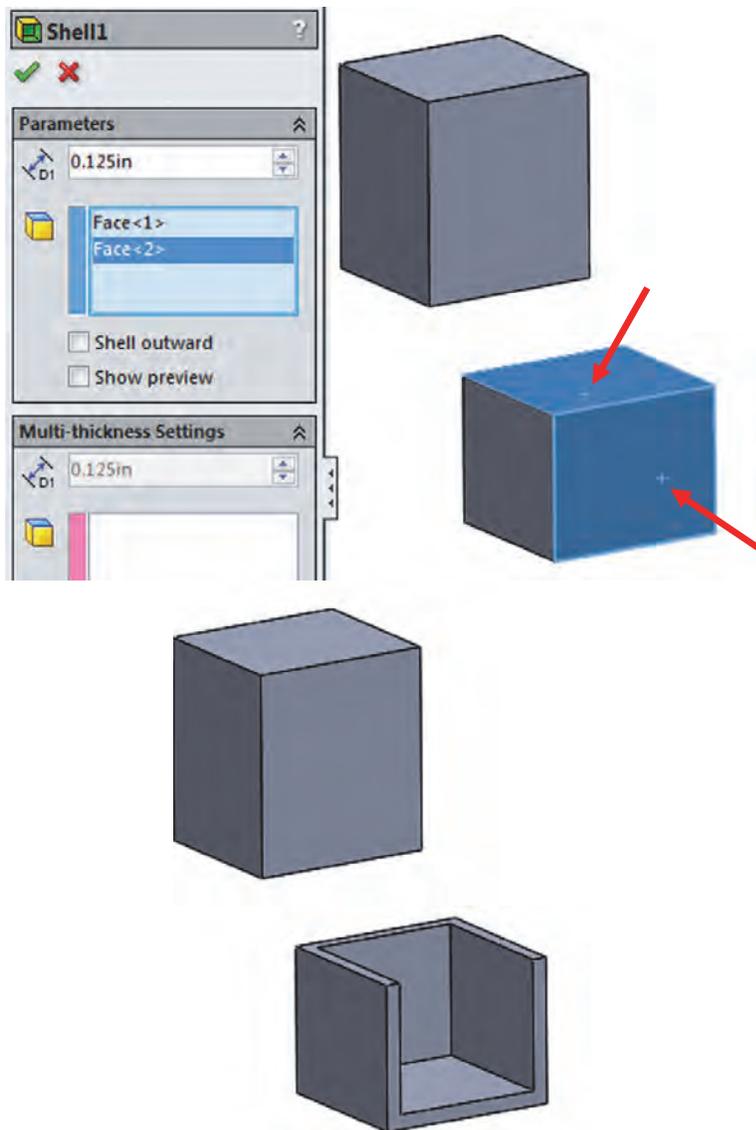


2. – Select the “**Boss Extrude**” command. Notice that we are not given a warning about having two separate bodies; it just works. Extrude any size that looks similar to the following image (dimensions are not important at this time) and click **OK** to finish. (Making two or more extruded/revolved/swept/lofted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

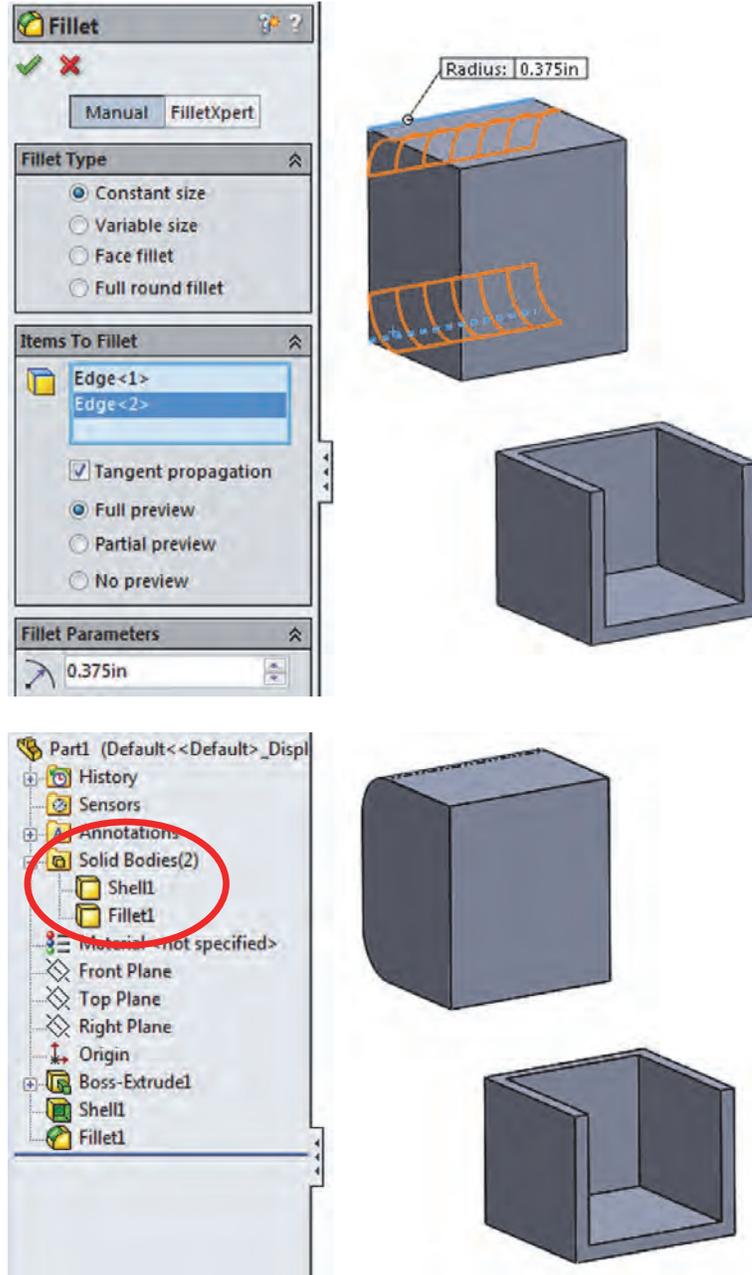


The first thing we notice is a new folder in the FeatureManager called “*Solid Bodies (2)*.” This folder is automatically added when SolidWorks detects multiple disjointed bodies in a part and lists the number of bodies found in the part (in this case 2). If we expand the folder, we can see the two bodies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

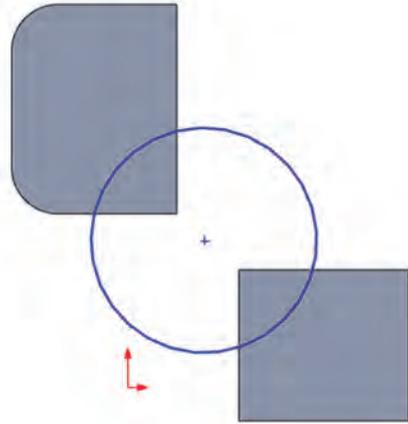


4. – Select the **Fillet** command and round two corners of the top body as shown.

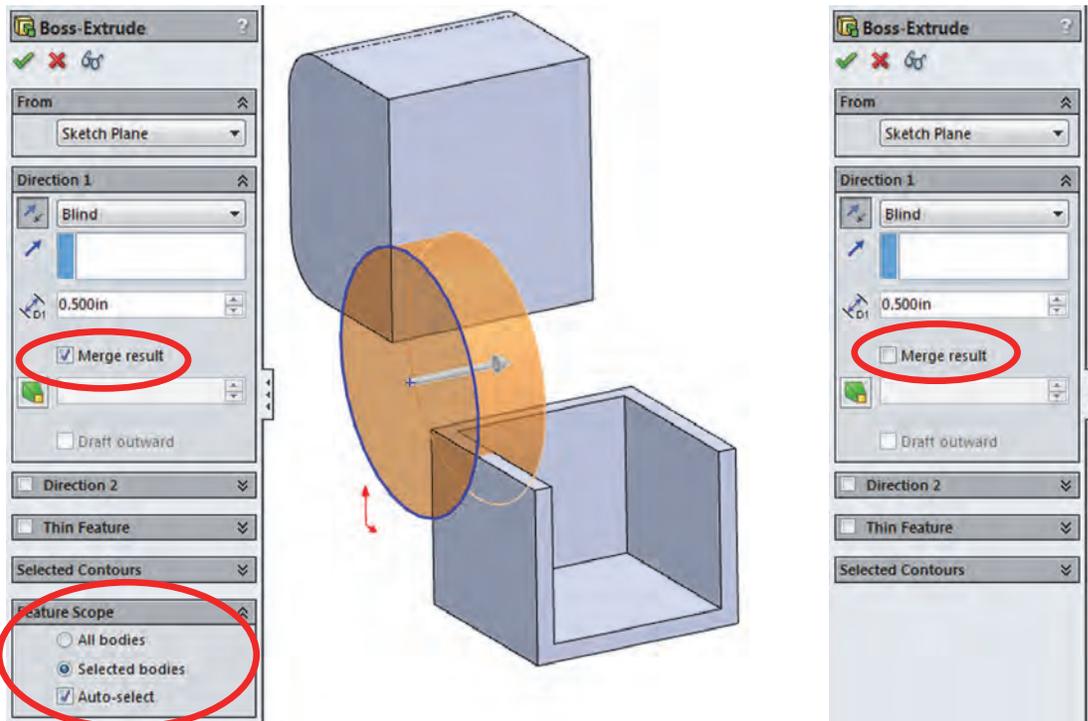


To help the reader understand the concept of local operations better, “local” means that we can add applied features (features that don't require a sketch like Shell, Fillet, Chamfer, Draft, etc.) to one body at a time. Notice the name of a body changes to the last feature applied to it.

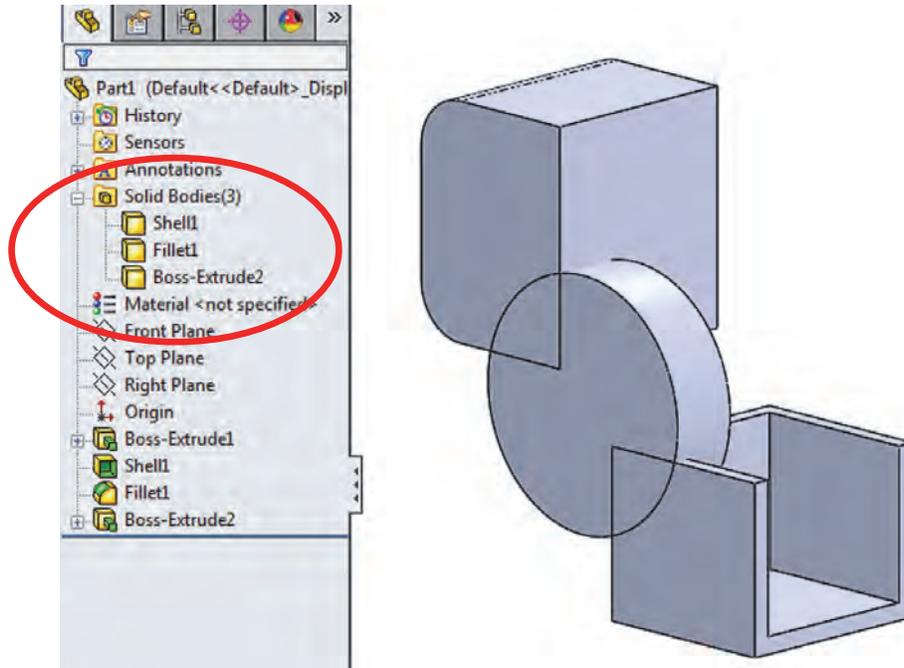
5. – If we add another boss extrude feature that intersects with existing bodies, by default it will automatically merge with them. At the time of making the extrusion we can optionally select which bodies to “merge” (or fuse) with to make a new single body. Add a new Boss-Extrude and explore the option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



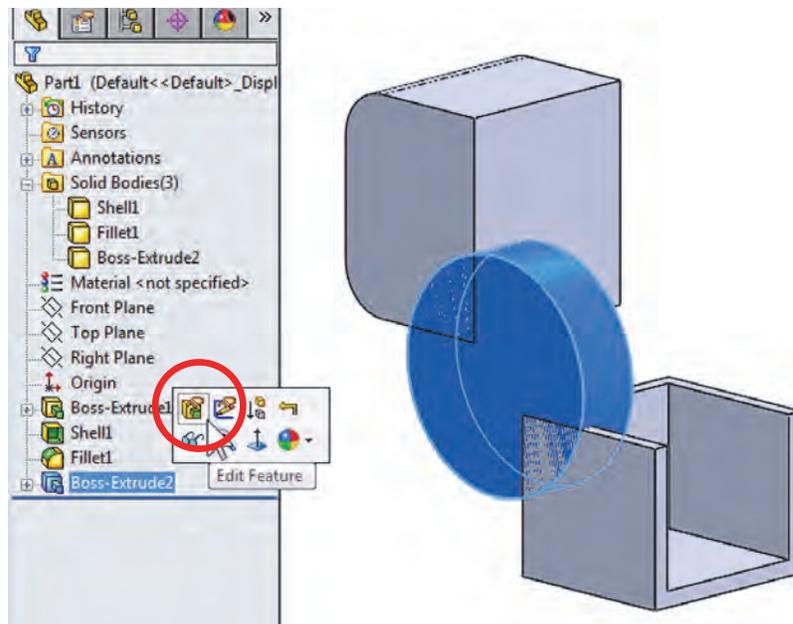
6. – Extrude the sketch *into* the existing bodies. Notice the “Merge result” option in the Extrude command. It’s always been there (except when there are no solid bodies) and by default is always checked. When we have multiple bodies in a part we see an additional option at the bottom called “Feature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

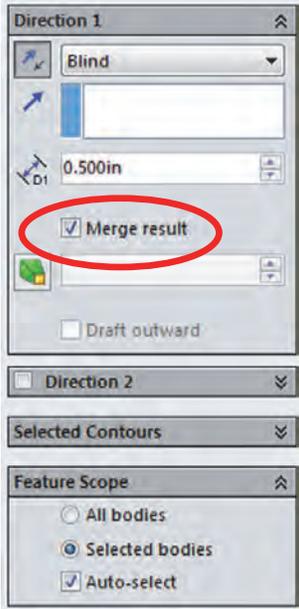
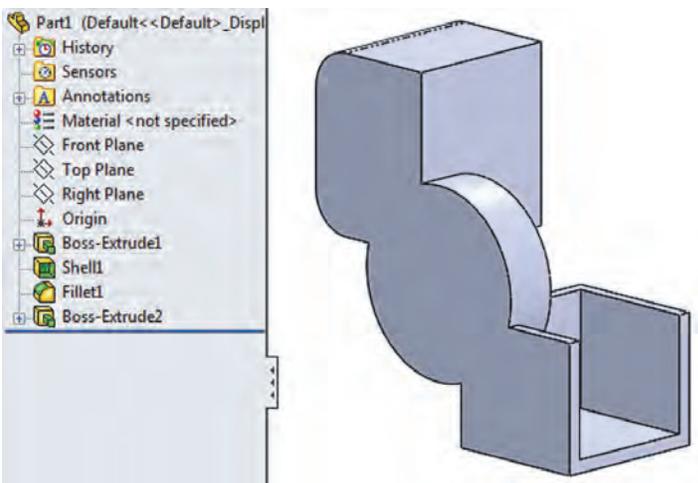


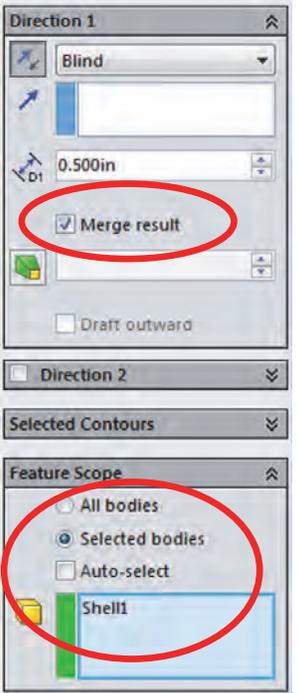
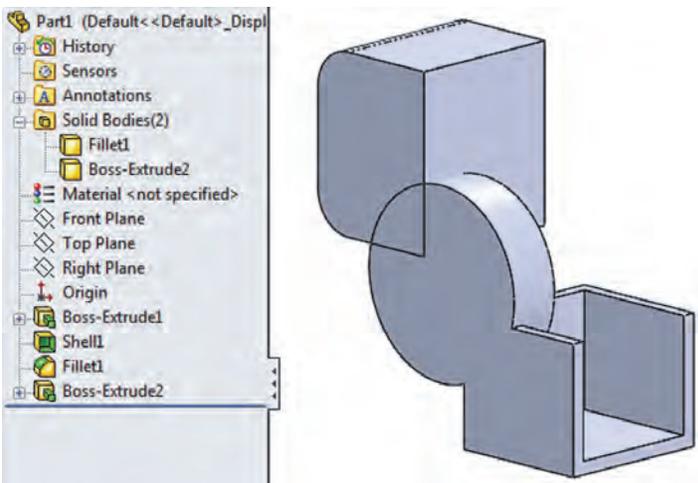
7. – After adding this extrusion we end up with three bodies in our part. See how the different bodies' edges intersect each other. If the “Merge result” option had been checked, we would not see these edges overlapping as they would have merged into a single body.



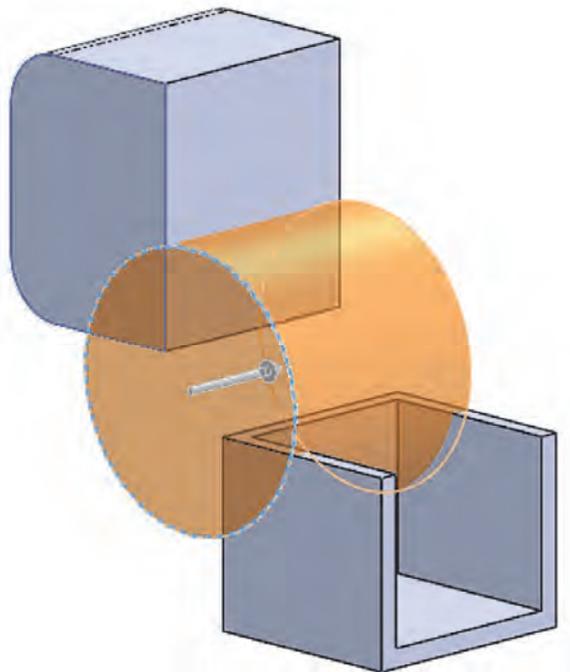
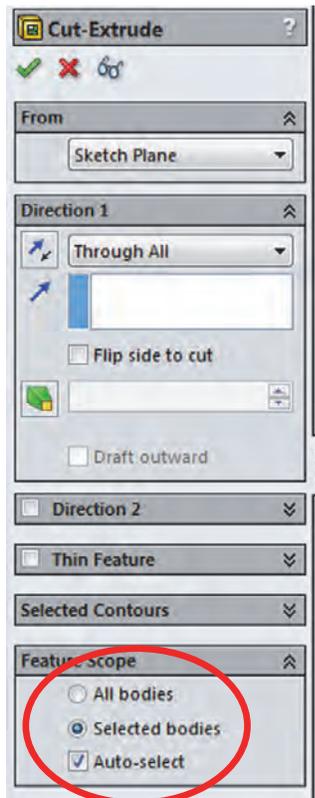
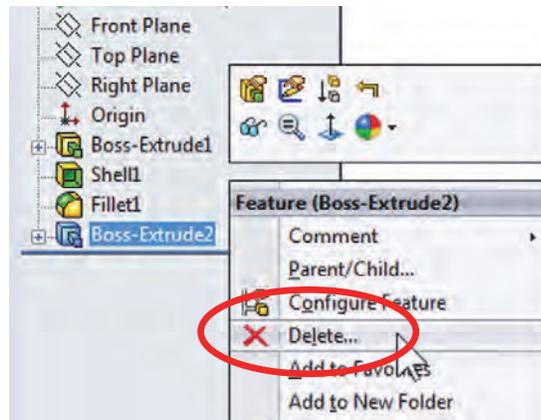
Edit the last extrusion to explore the effect of different “Merge Result” and “Feature Scope” combinations.



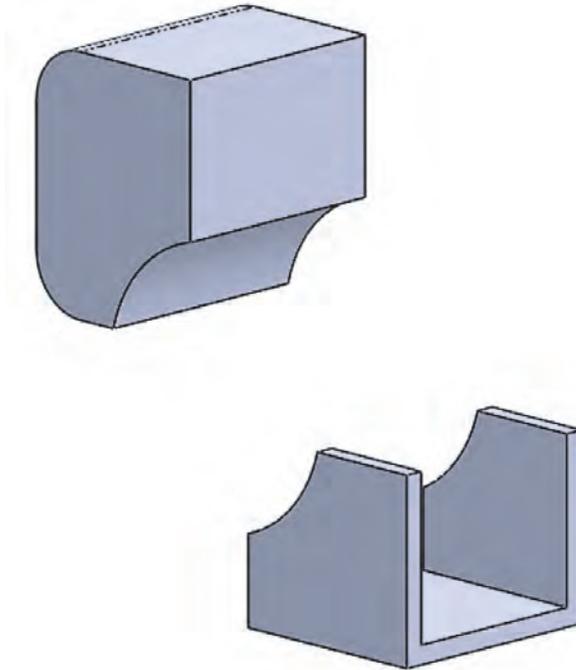
Combination:	Result:
<p>Merge Result: Checked Feature Scope: Auto-Select</p> 	<p>Single body. By touching both of the existing bodies “Auto-select” merges with them, fusing them into a single solid body. The “Solid Bodies” folder is no longer visible as there is only one solid body in the part.</p> 

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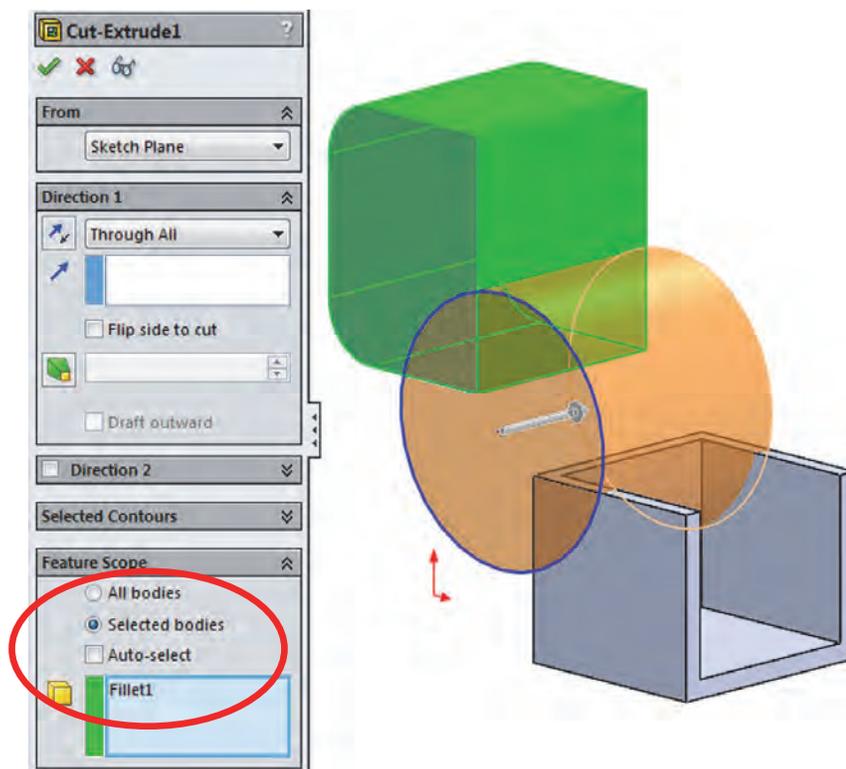
8. –The “Merge result” option works the same way with any feature that adds material to the part, including revolved boss, sweep, loft, etc. Now we’ll see how it works when we remove material. Delete the Boss-Extrude feature and keep the sketch. Select the sketch in the FeatureManager and make a Cut-Extrude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



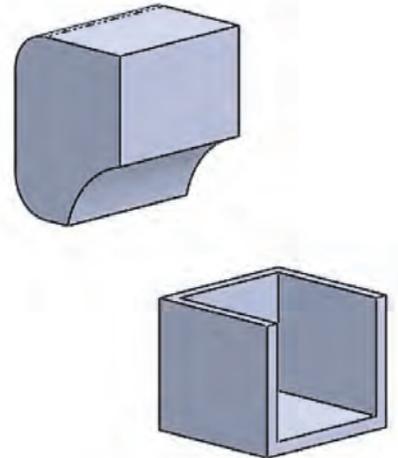
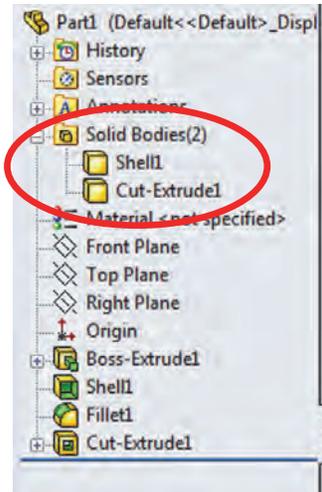
9. – What we end up with is the same two bodies we had before, but now they have a cut through them because the cut automatically selected them.



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.

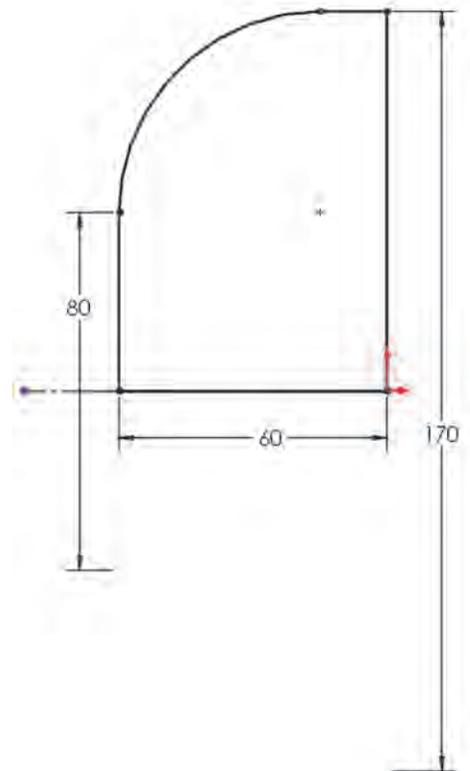


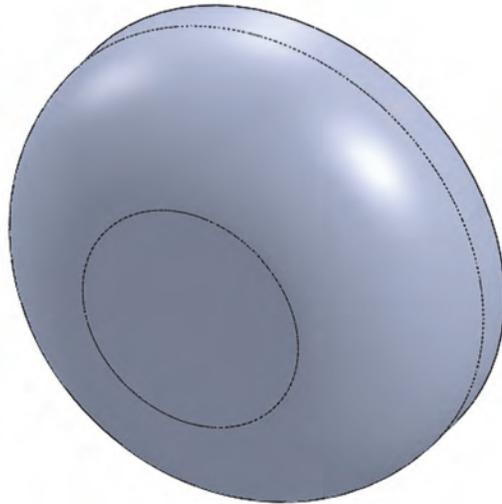
And just as with the features that add material, this technique works the same way with all the features that remove material including revolved cut, sweep cut, loft cut, etc.



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 common volume between two bodies and adding volumes. 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 new 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).

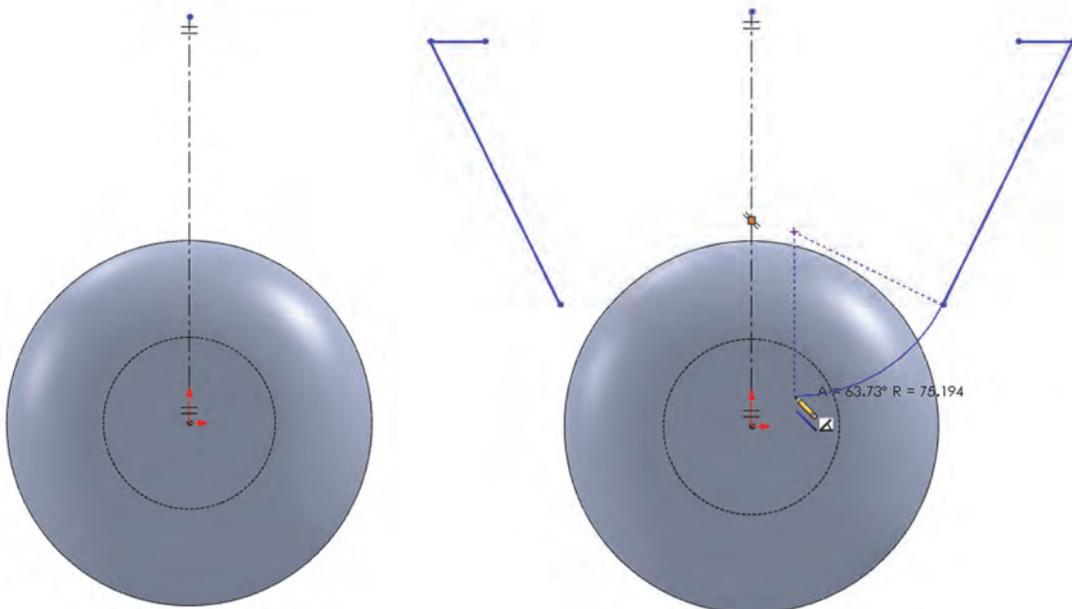
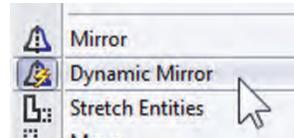


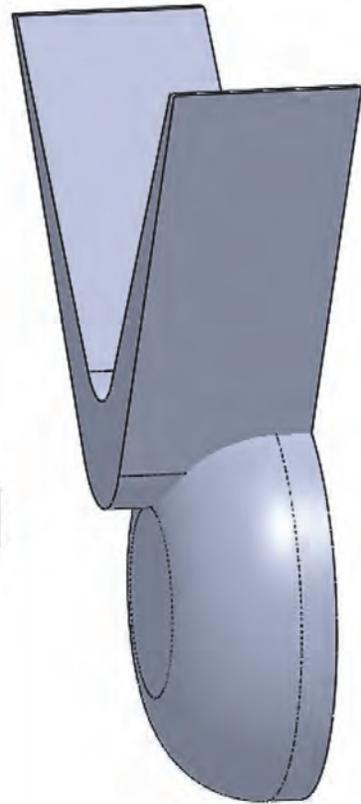
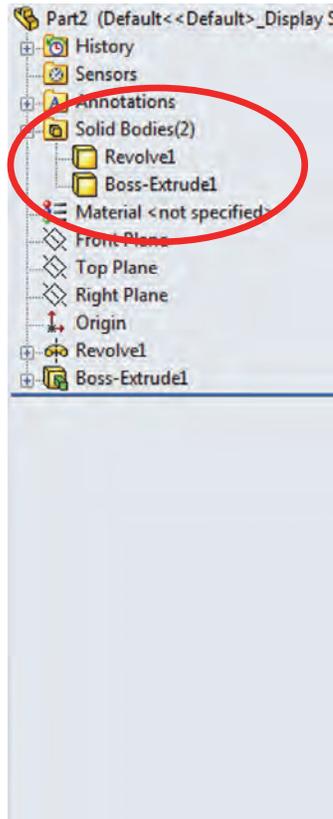
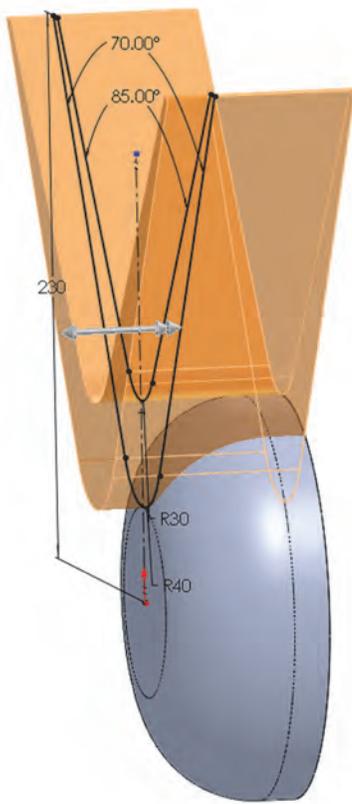
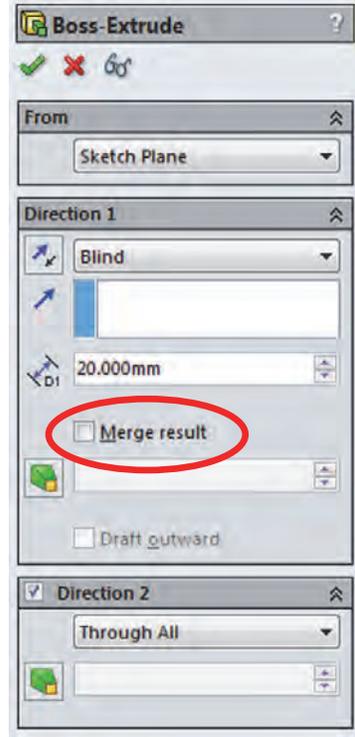
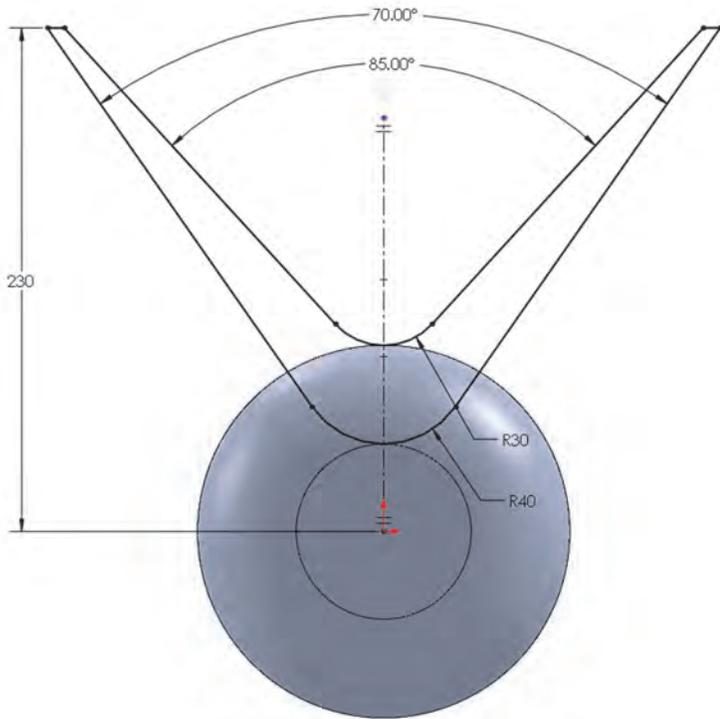


12. – Add a new sketch in the front flat face of the part, and add the following sketch. The arcs are tangent to the circular edges. Make the extrusion 20mm to the front (Direction 1) and “Through All” going to the back (Direction 2.) Turn off the “Merge result” option to have two different bodies.

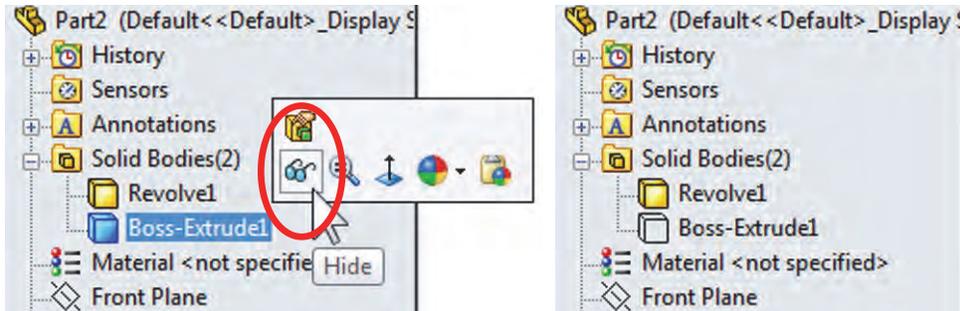


To make the sketch symmetric about the centerline, draw the centerline first, and while it's selected, turn on the “**Dynamic Mirror**” tool in the menu “**Tools, Sketch Tools, Dynamic Mirror.**” After we turn it on the centerline will show an equal sign at both ends letting us know that whatever we 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.

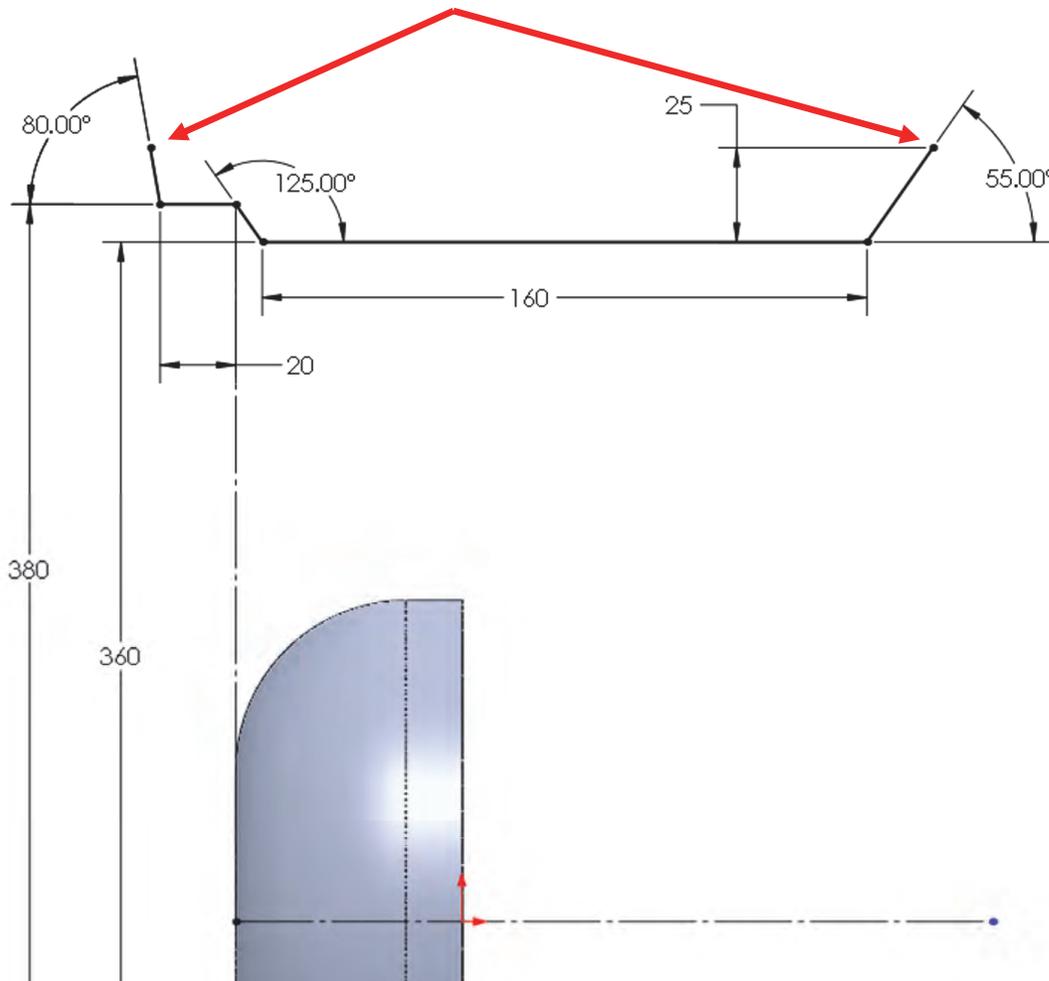




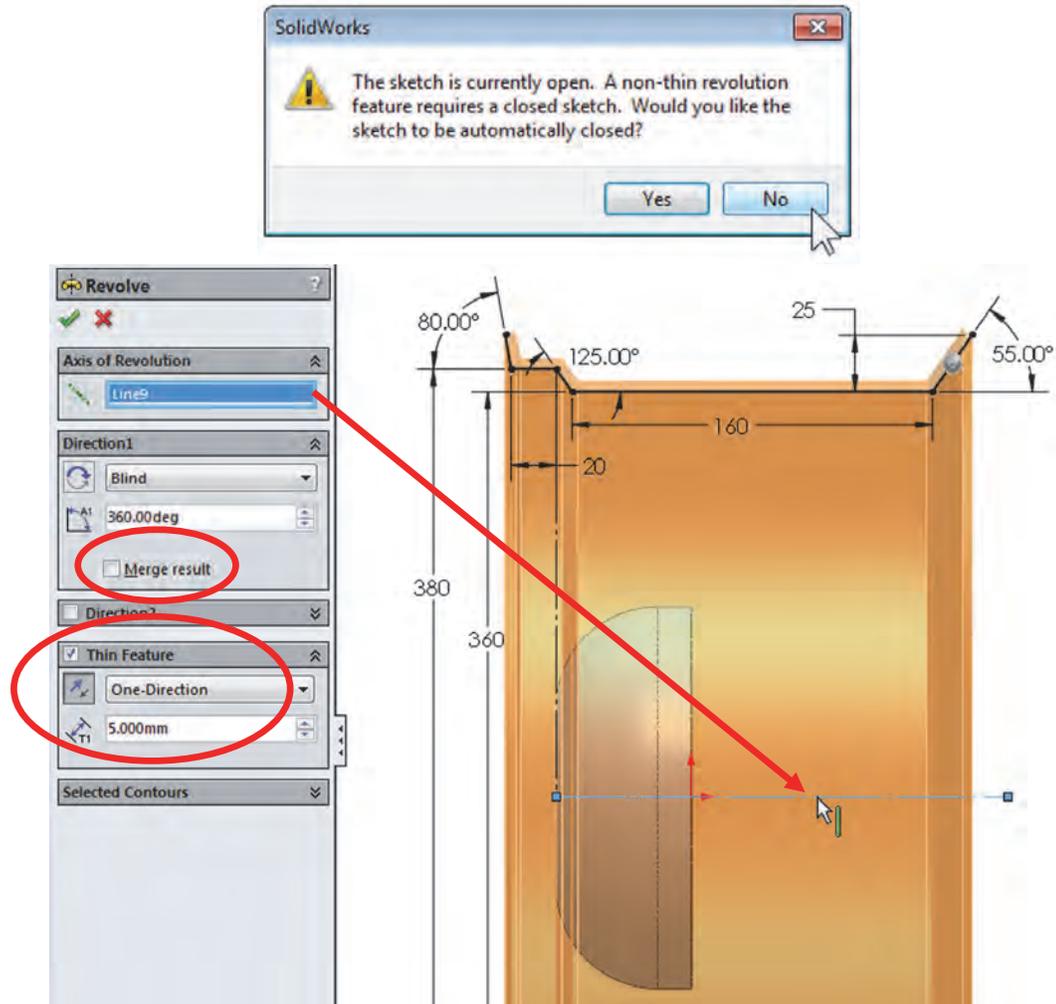
13. – In the next step we'll make the outside rim of the wheel. To better visualize this step, the second body will be hidden from view. Just like we can hide a part in an assembly, we can hide a solid body in a part. In the “*Solid Bodies*” folder, select the “*Boss-Extrude1*” body and select “*Hide*”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



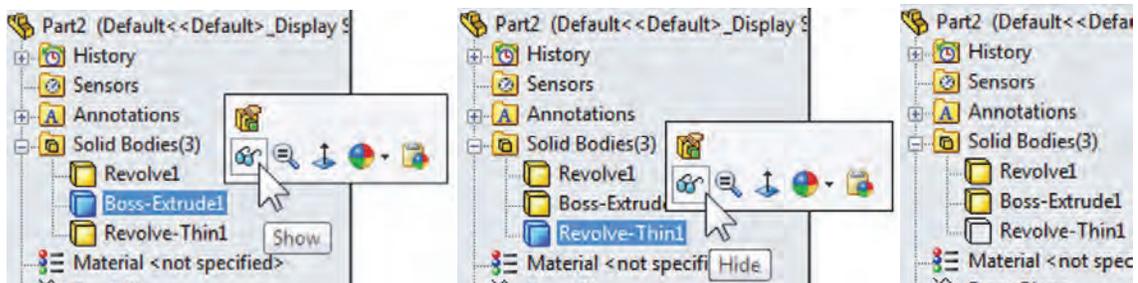
14. – Add the following sketch in the “*Right Plane*”; we'll use it to create a thin revolved feature to build the wheel's rim. The 360mm and 380mm dimensions are doubled about the horizontal centerline. The endpoints at the ends of the sketch are at the same height; add a Horizontal geometric relation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



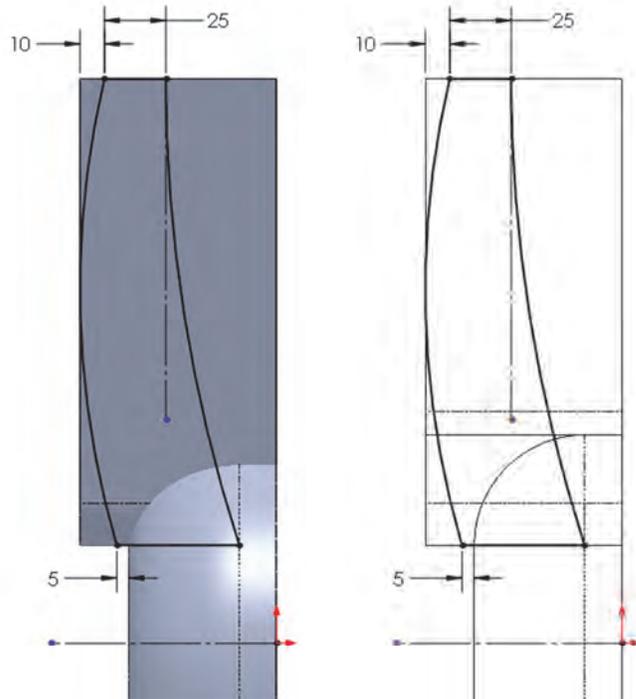
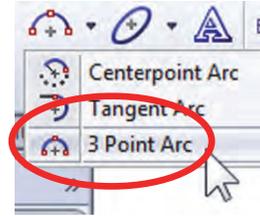
15. – Select the “**Revolved Boss/Base**” command and make a 360 deg. thin revolve. We’ll be asked if we want to close the sketch, select “No” to continue. Use the horizontal centerline as the “Axis of Revolution”, turn off the “Merge result” checkbox and make the “Thin Feature” 5mm thick going outside.



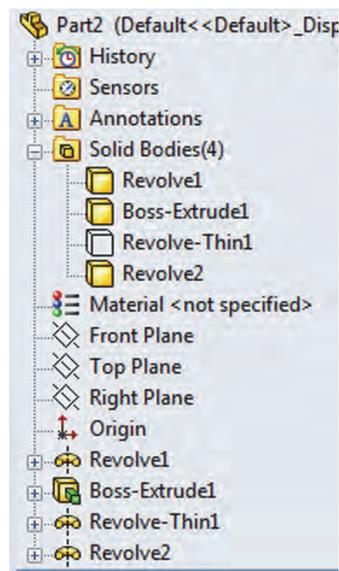
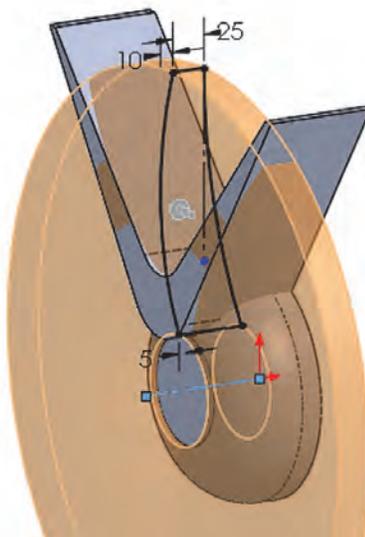
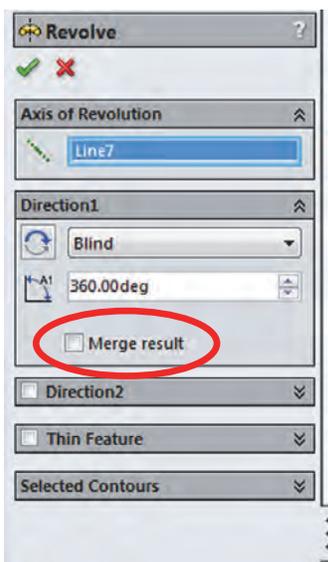
16. – Now we have three different solid bodies in our part. The next feature will be another revolved feature that will be used to build the wheel’s spokes. Before making the new solid body, hide the “*Revolve-Thin1*” body and show the “*Boss-Extrude1*” body.



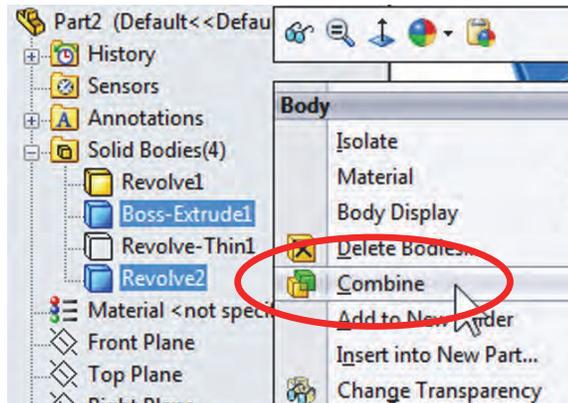
17. – After hiding the revolved thin feature, add a new sketch in the “Right Plane.” Note: The arc on the left is tangent to the solid body’s edge, and the arc on the right is tangent to a vertical centerline. Don’t forget the horizontal centerline at the origin to make the revolved feature about it.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



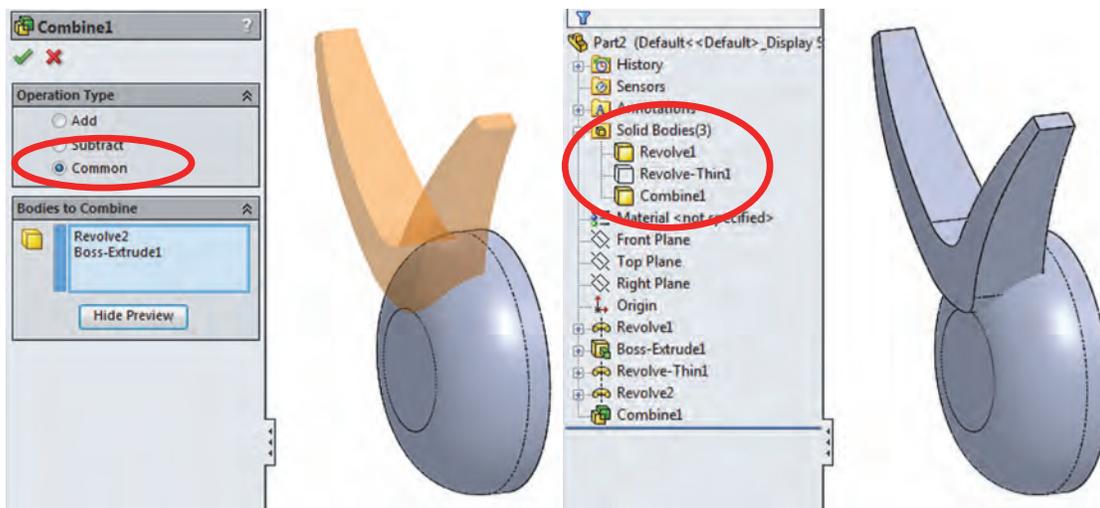
18. – Make a 360 deg revolved boss using the horizontal centerline and turn off the “Merge result” checkbox to create the fourth solid body.



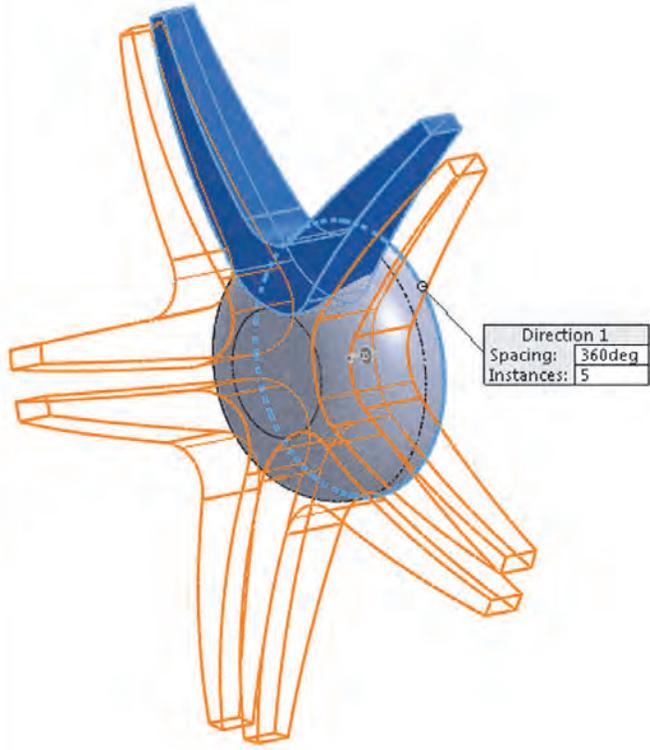
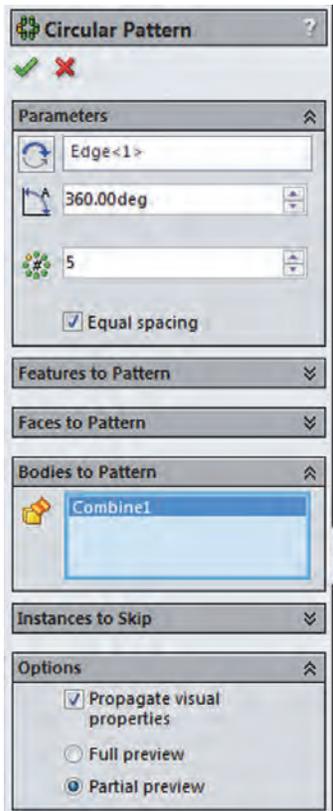
19. – In the next step we will obtain the common volume between two of the bodies to get the spokes of the wheel. Select the “*Boss-Extrude1*” and the “*Revolve2*” bodies in the “*Solid Bodies*” folder holding the “*Ctrl*” key while selecting, and from the right-mouse-button menu select the command “**Combine**”,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



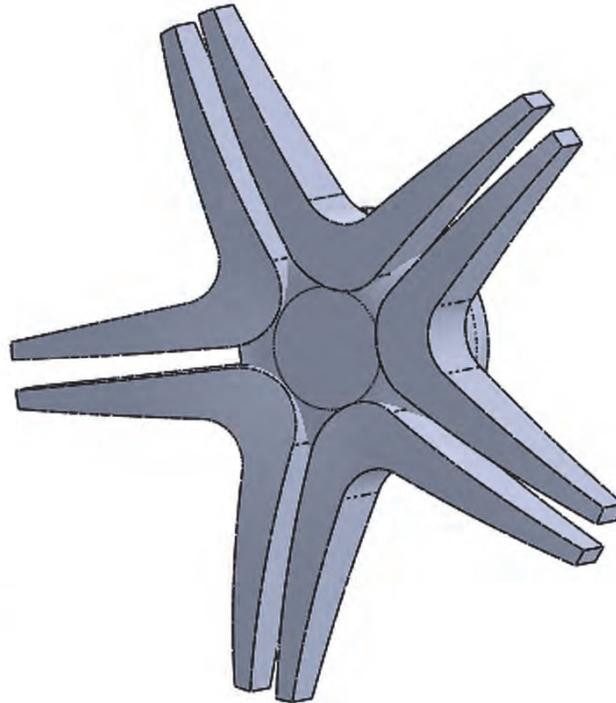
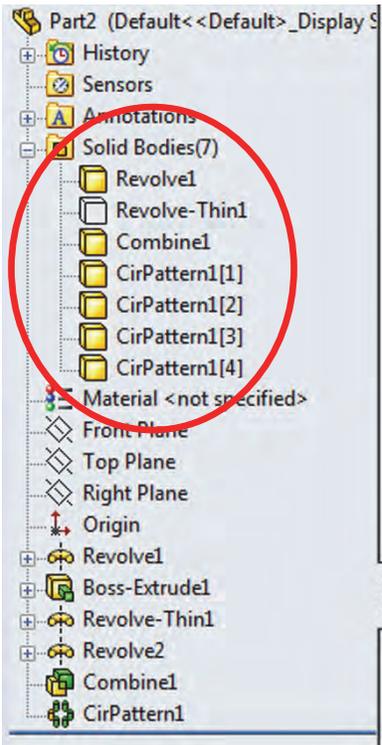
In the “*Operation Type*” options select “*Common*” and then press the “*Show Preview*” button (it will change to “*Hide Preview*”). The “**Combine**” command is used to Add bodies to form a new one, Subtract one or more bodies from another, or get the Common volume between two or more bodies, as is 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.



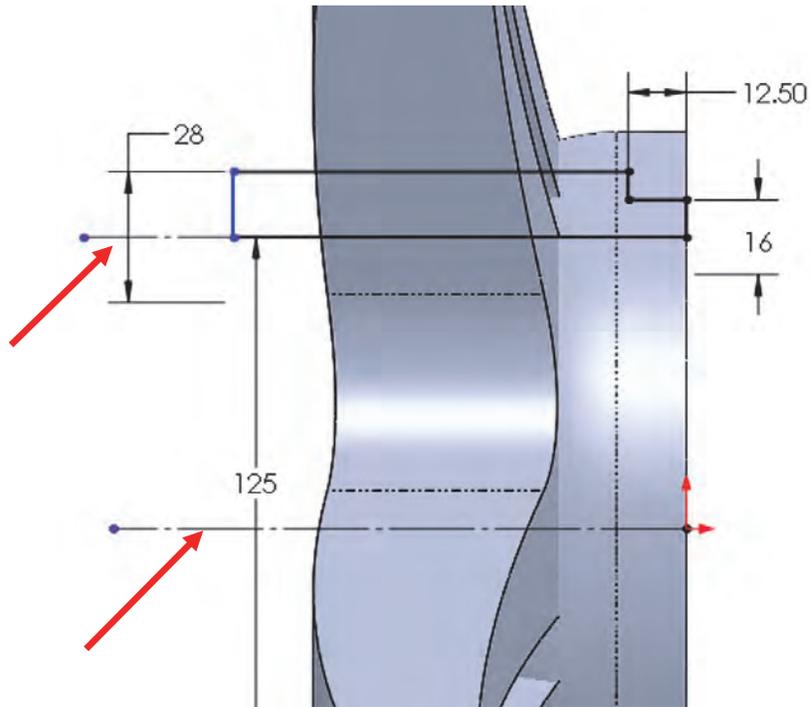
20. – Now that we have the body we are interested in to create the spokes, we need to make a circular pattern using the combined body as a seed feature. Select the “*Circular Pattern*” command using any circular edge for the “*Pattern Axis.*” For this step, instead of selecting the “*Combine1*” feature to make our pattern, expand 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.



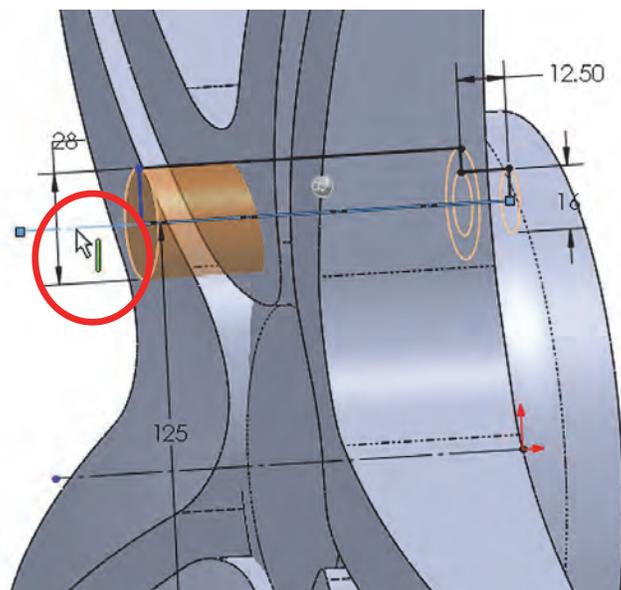
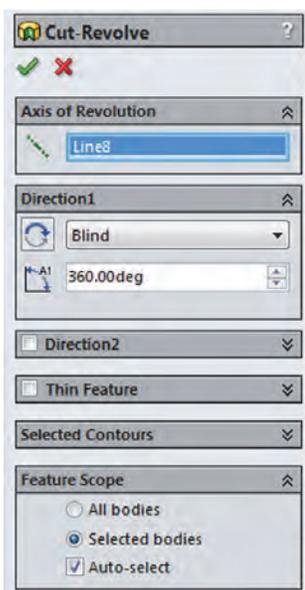
21. – After adding the circular pattern we have 7 different bodies in our wheel.



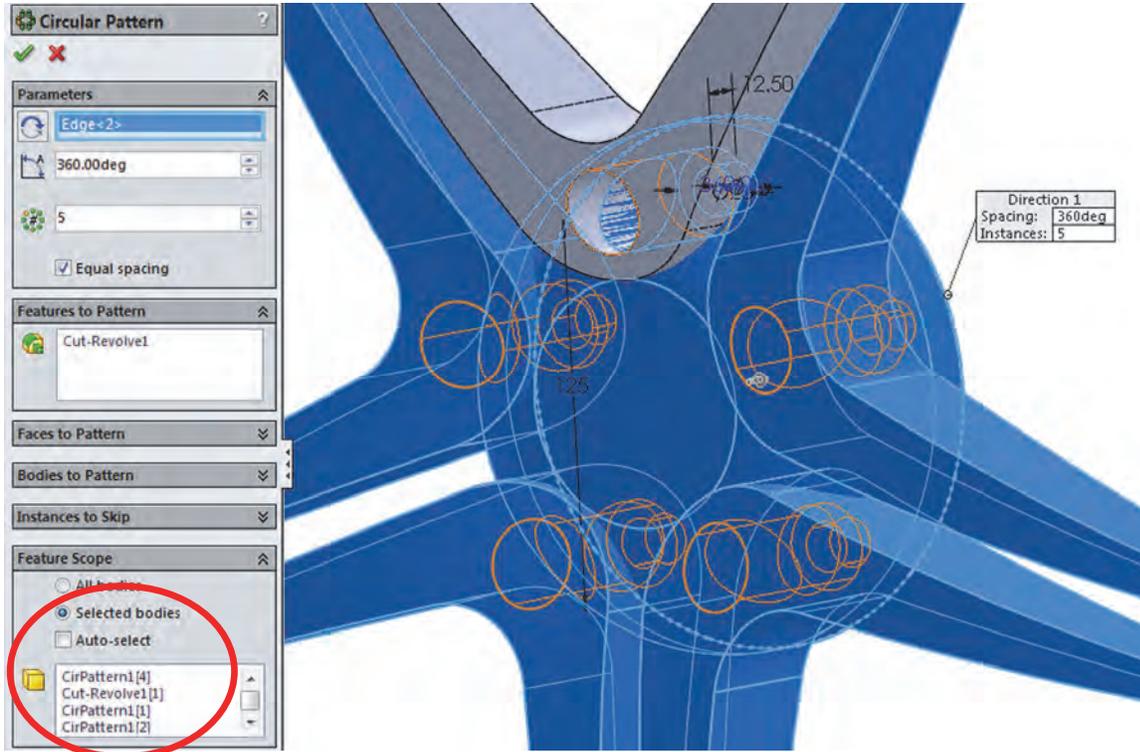
22. – The next step could be done after we combine the bodies into a single body, but we chose to add it at this time to show more multi-body functionality. In this step we'll add the holes for the bolts. Add the following sketch in the “*Right Plane*.” The 28mm and 16mm dimensions are doubled about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



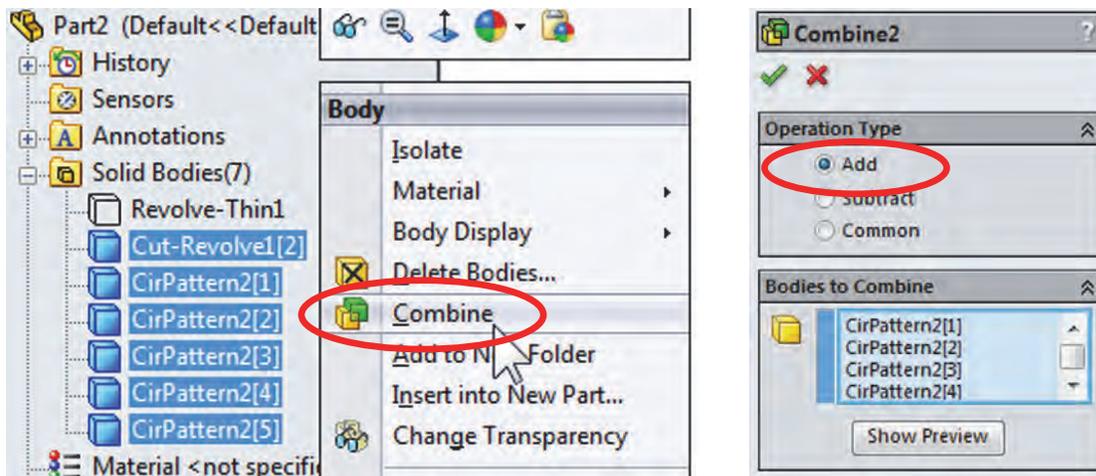
23. – Make a “Revolved Cut” using the top centerline as the axis of revolution, and leave the “Feature Scope” option to “Selected Bodies, Auto-select.”



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.



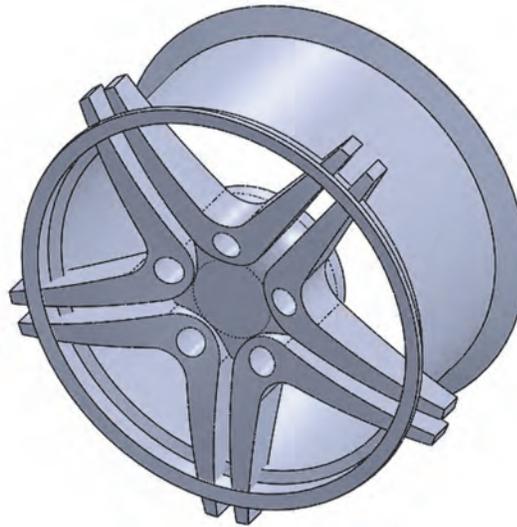
25. – Now that the holes for the bolts are complete we can merge the spokes and the hub into a single solid body. In the “Solid Bodies” folder select the patterned bodies and the hub as indicated, right-mouse-click in any of them and select “Combine” from the pop-up menu, or use the menu “Insert, Features,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



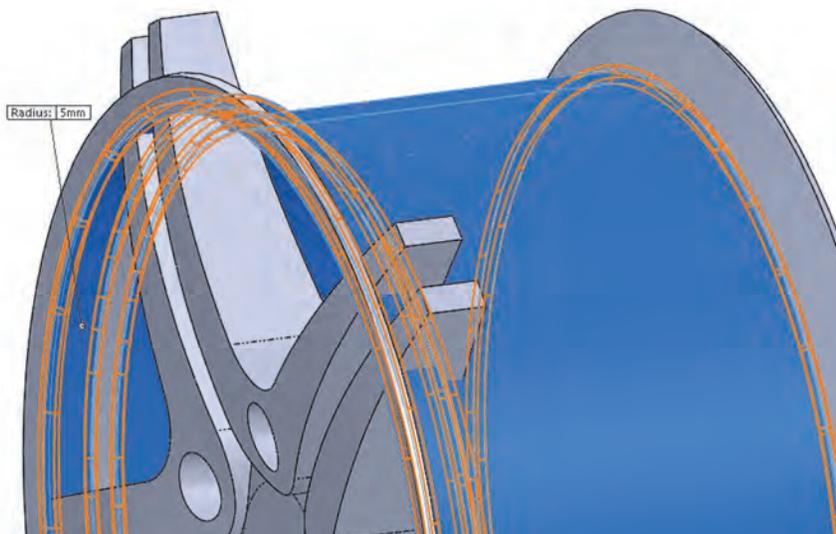
26. – After adding the bodies in the previous step we have two bodies in the part. One is the hub with the spokes and the other (hidden up to this point) is the wheel itself. In the “Solid Bodies” folder show the hidden body. Select it and click in “Show” from the pop-up menu.



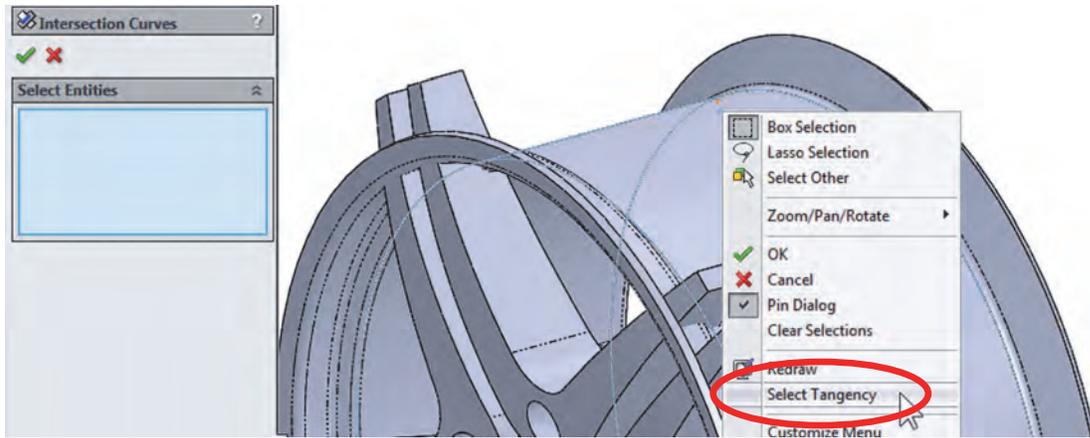
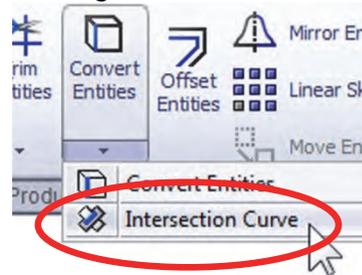
To hide or show a solid body we can select any feature that modified it in the FeatureManager and use the same “Hide/Show” command from the pop-up toolbar, not just in the “Solid Bodies” folder.



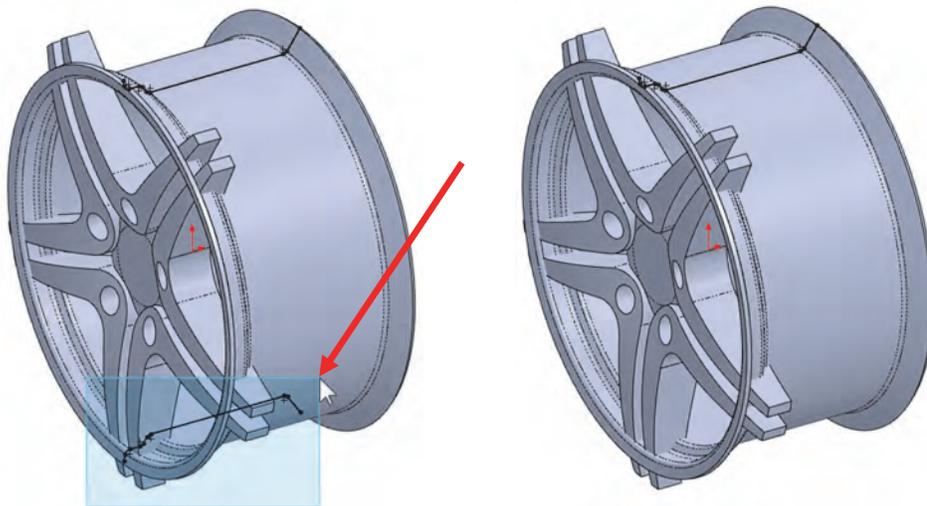
27. – To make the rim better looking add a fillet to round off the sharp edges of the “Revolve-Thin1” feature. Add a 5mm to all of the inside and outside sharp edges in the rim. All the edges can be rounded by selecting four faces.



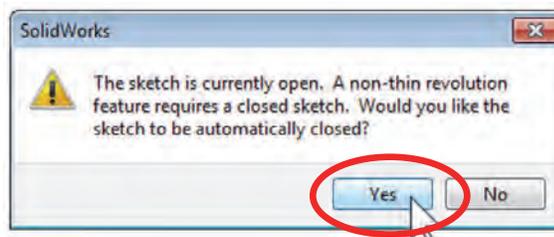
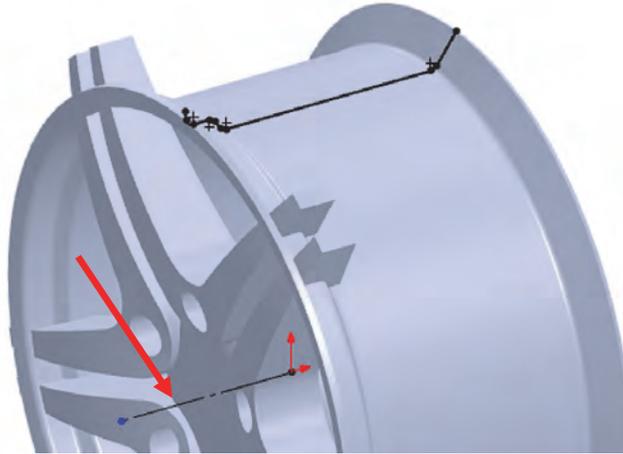
28. – The next step is to trim the excess material from the spokes that protrudes beyond the rim. We'll use a "Revolved Cut" to trim it. Using the "**Intersection Curve**" command we can create a sketch that matches the profile of the rim to cut the spokes. Add a new sketch in the "*Right Plane*" and 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 either will give us the same result) using a right-mouse-click in any face 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.



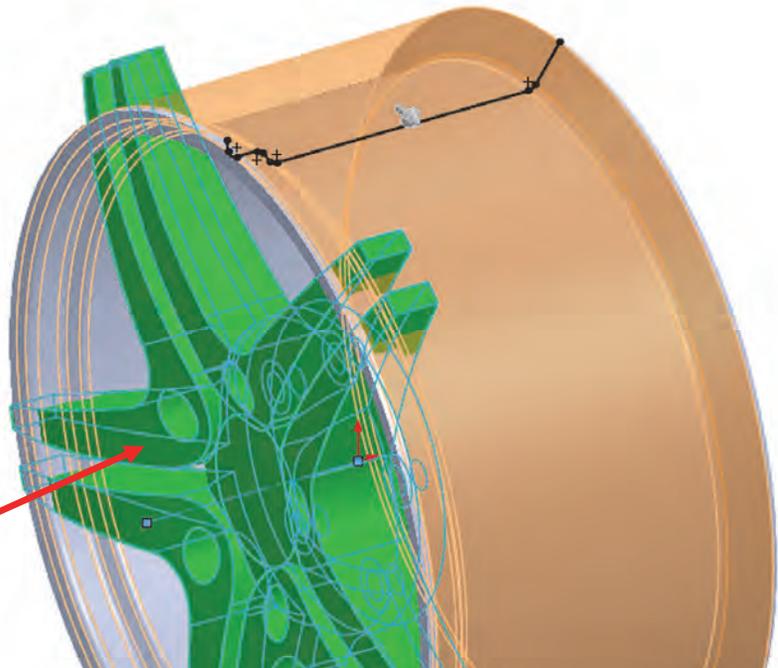
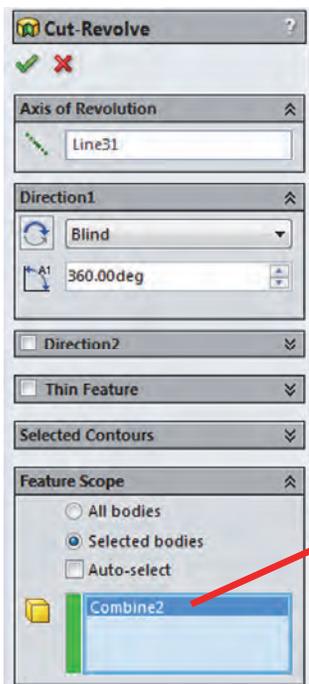
29. – After completing the intersection curve, sketch entities are created where the selected faces cross the sketch plane. Since the faces cross the plane in two different places (top and bottom) we get two profiles. Window-select the bottom profile since we only need the top one for this feature.



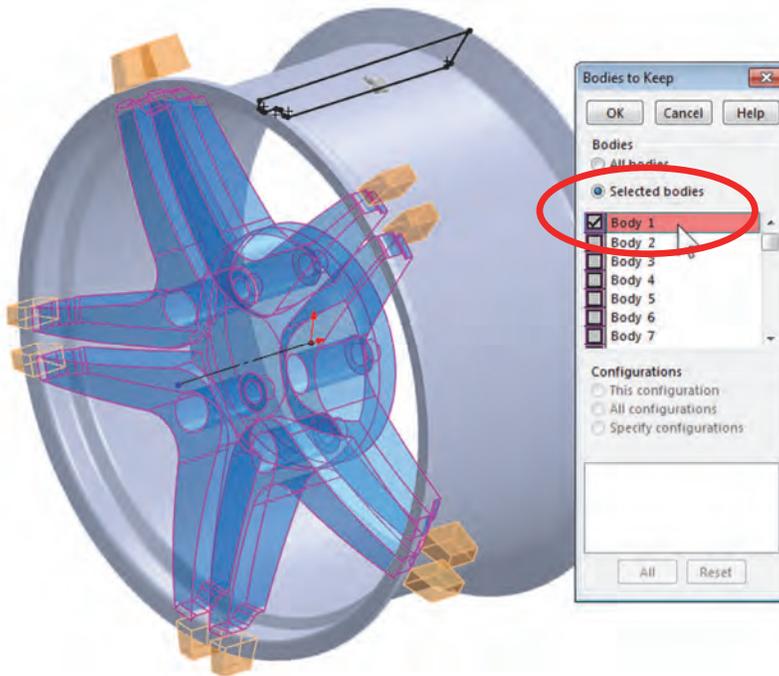
30. – Add a horizontal centerline at the origin and make a **“Revolved Cut.”** We’ll get the message alerting us that the sketch is open and asking if we want to close the sketch. In this case we want to create a closed profile and select **“Yes”** when asked. The model is displayed using **“Shaded”**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



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



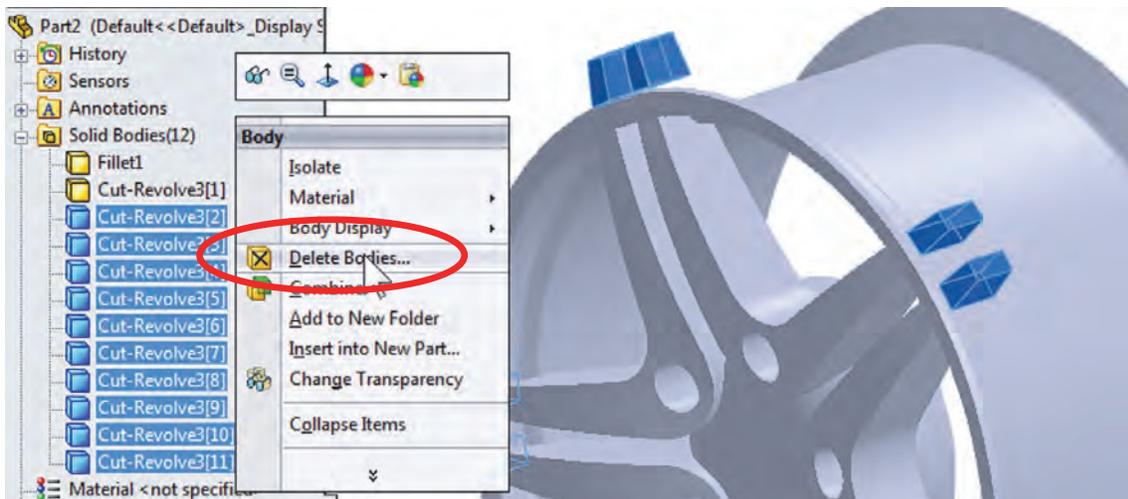
32. – After pressing OK to make the revolved cut, the profile we used generates more bodies by trimming the tips of the spokes, and now we are asked which bodies we want to keep. In the pop-up selection box use the “Selected Bodies” option, and pick the body at the center of the spokes to keep it. You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



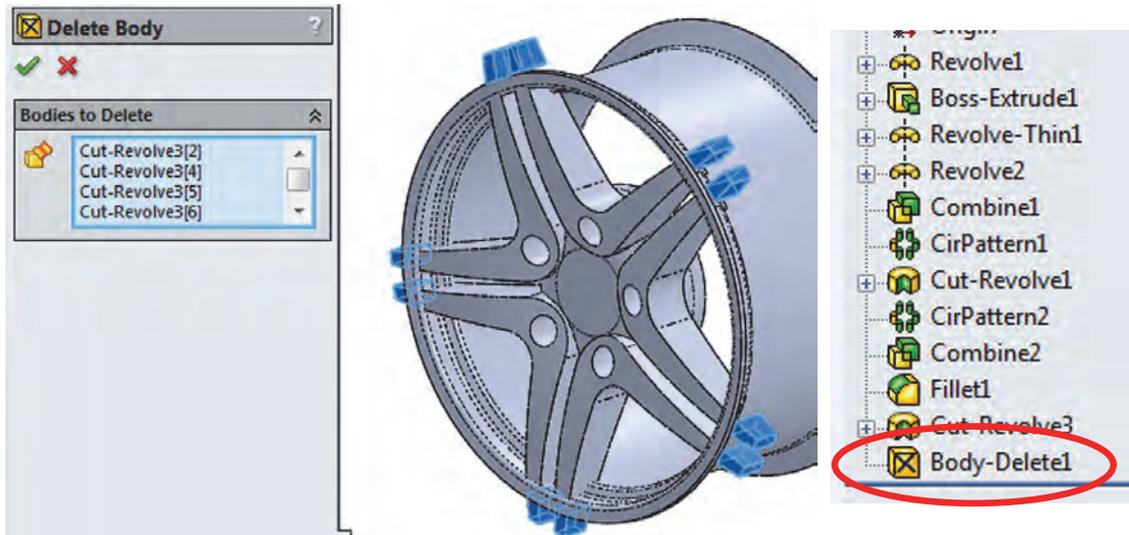
When a cut operation splits a part into multiple bodies, the largest body is usually the first one listed in the “Bodies to Keep” list.



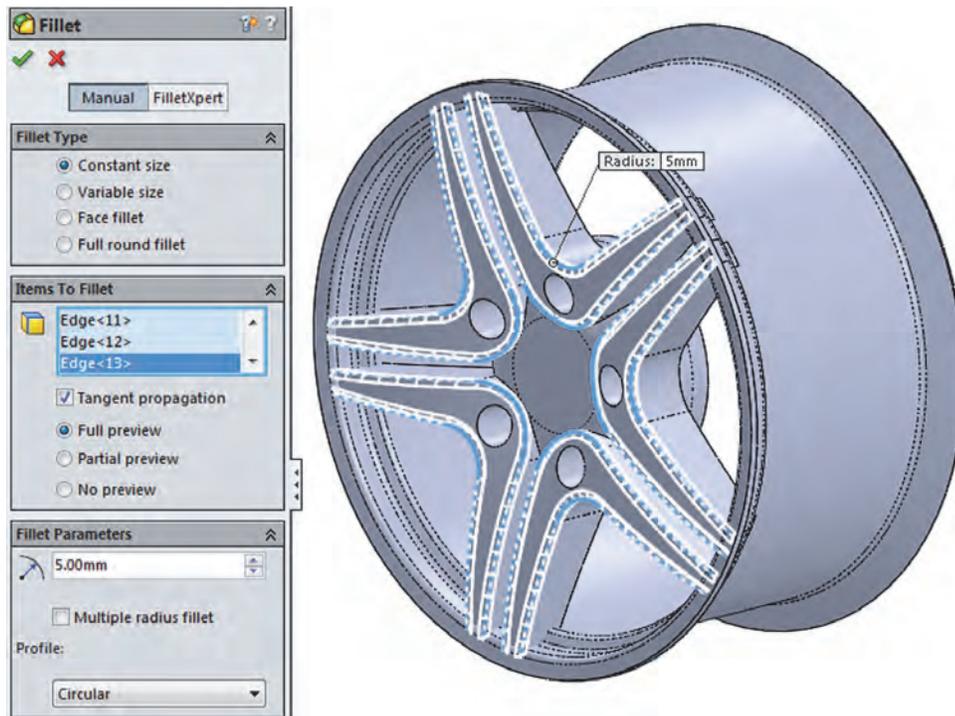
If we had kept all the bodies, we'd have a large number of bodies in the “Solid Bodies” folder. In that case we can delete the unwanted bodies by right-mouse-clicking on them and selecting “Delete Bodies.”



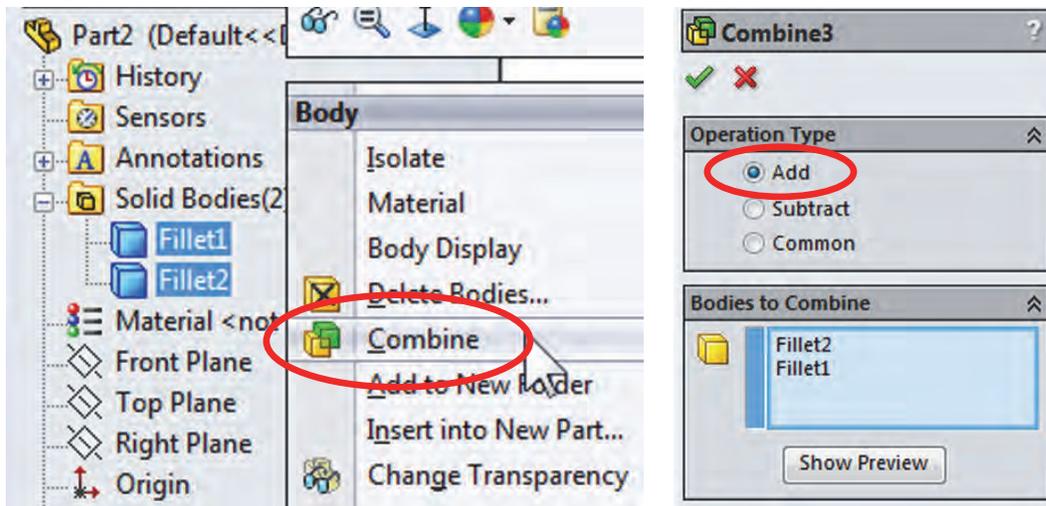
We are asked to confirm which bodies we want to delete. After we click OK to delete them, a new feature is added to the FeatureManager called “*Body-Delete.*”



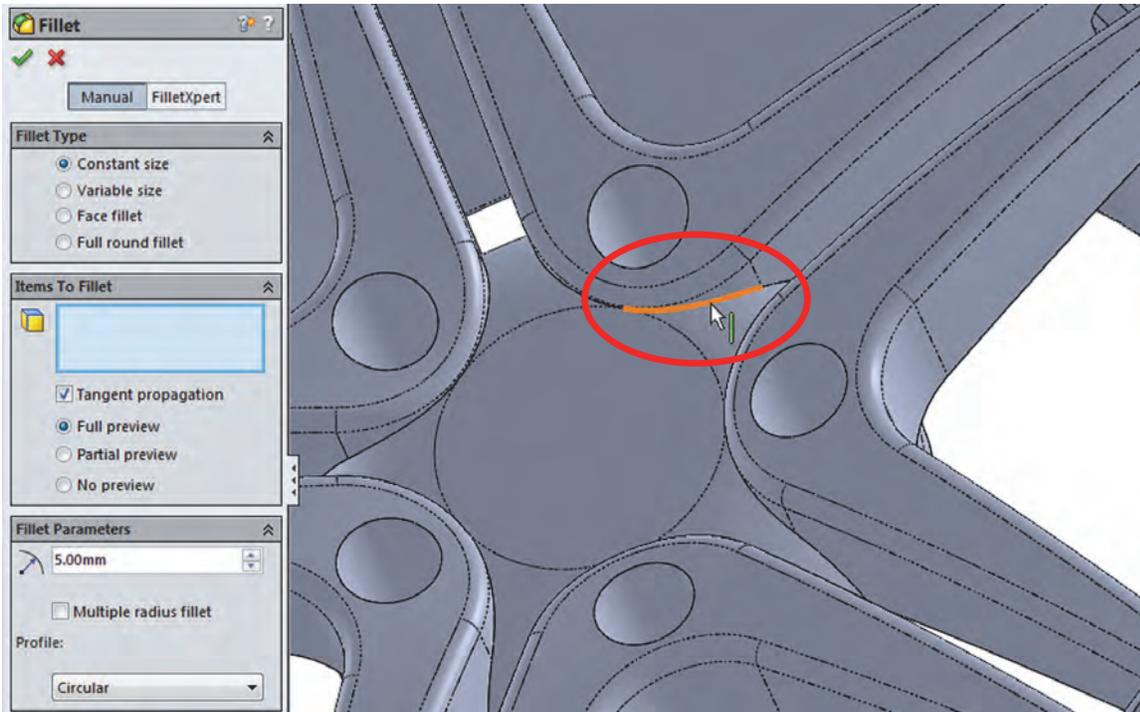
33. – Add a 5mm fillet to the spoke arms selecting the radial edges of the spokes and click OK to finish.



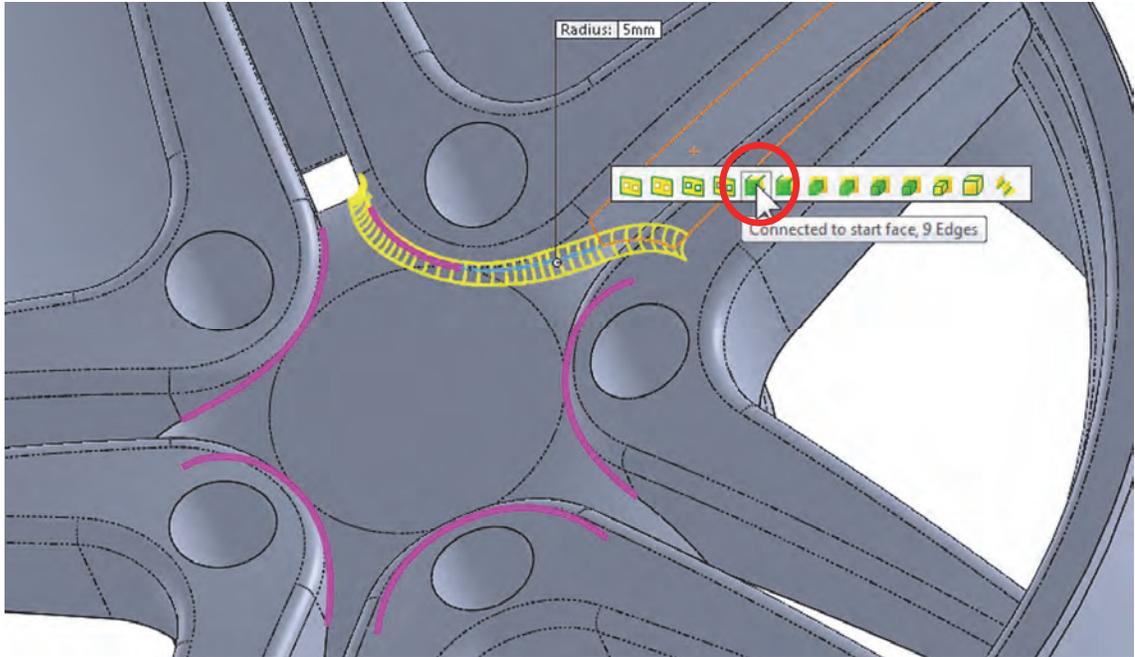
34. – Select the remaining two bodies and use the “**Combine**” command to add them together. In the “Operation Type” select “Add”, make sure we have both bodies selected and click OK to finish. After combining the last two bodies, the “*Solid Bodies*” folder goes away and we have a single body left.



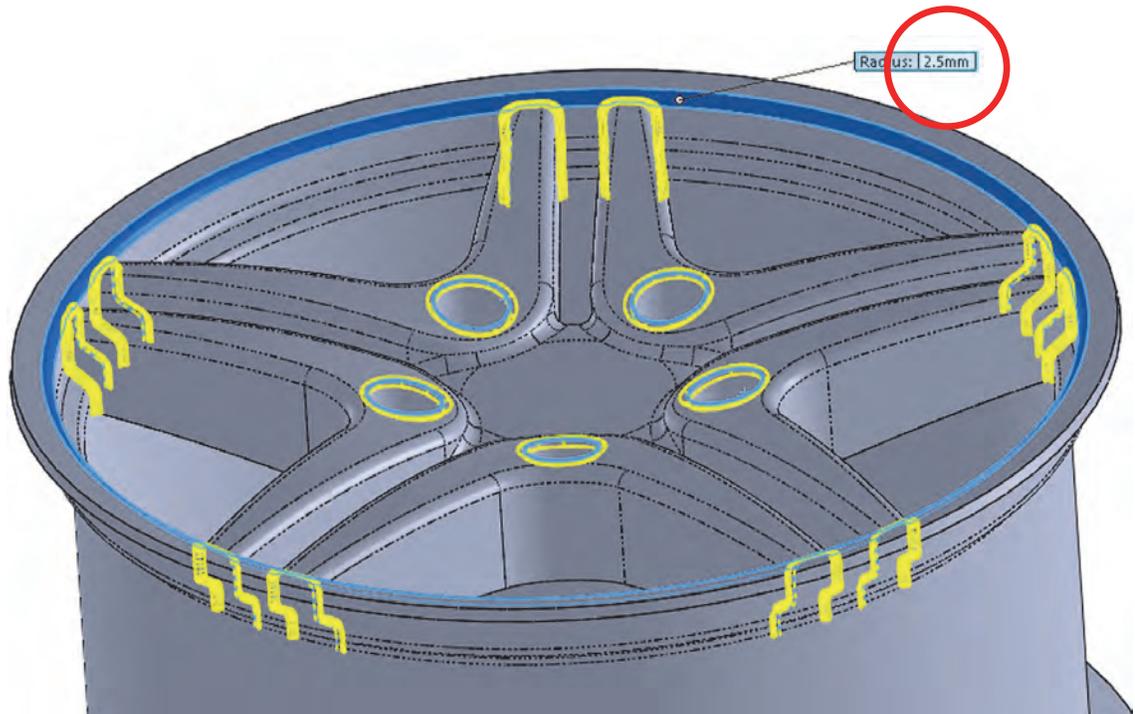
35. – Add a 5mm fillet to the edges in the center. After selecting the edge indicated, we see the expanded selection toolbar. Click in the “Connected to start face” icon to automatically select all the edges needed. Click OK to finish the fillet.



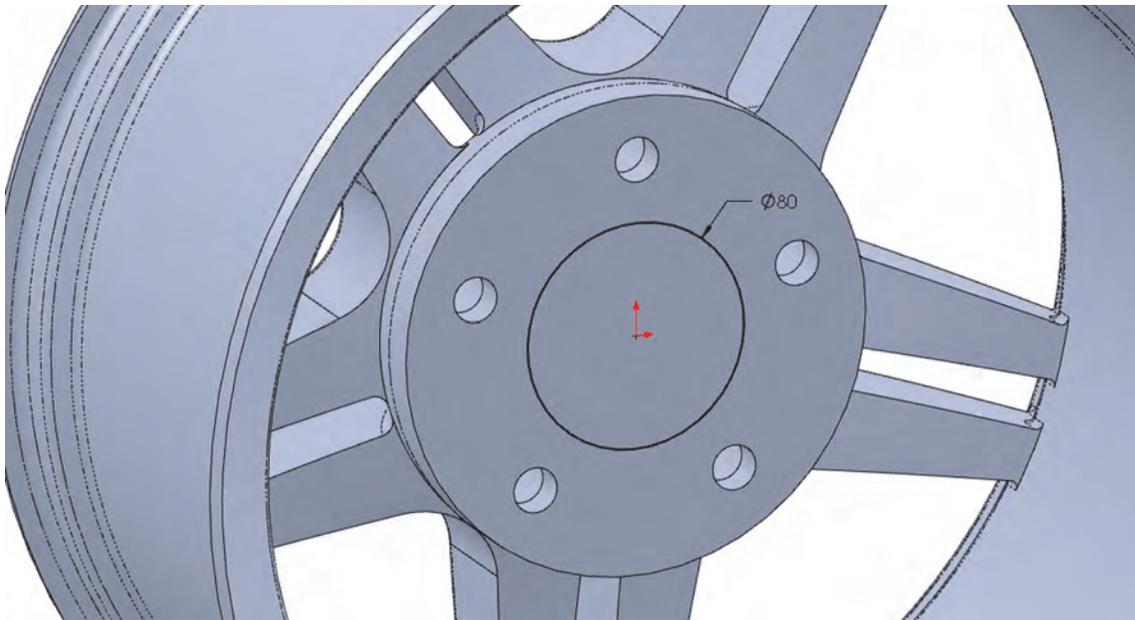
When we move the mouse over each of the icons in the selection toolbar we can see the edges that would be selected if we clicked on it. This is an easy way to make the selection of multiple edges easier.



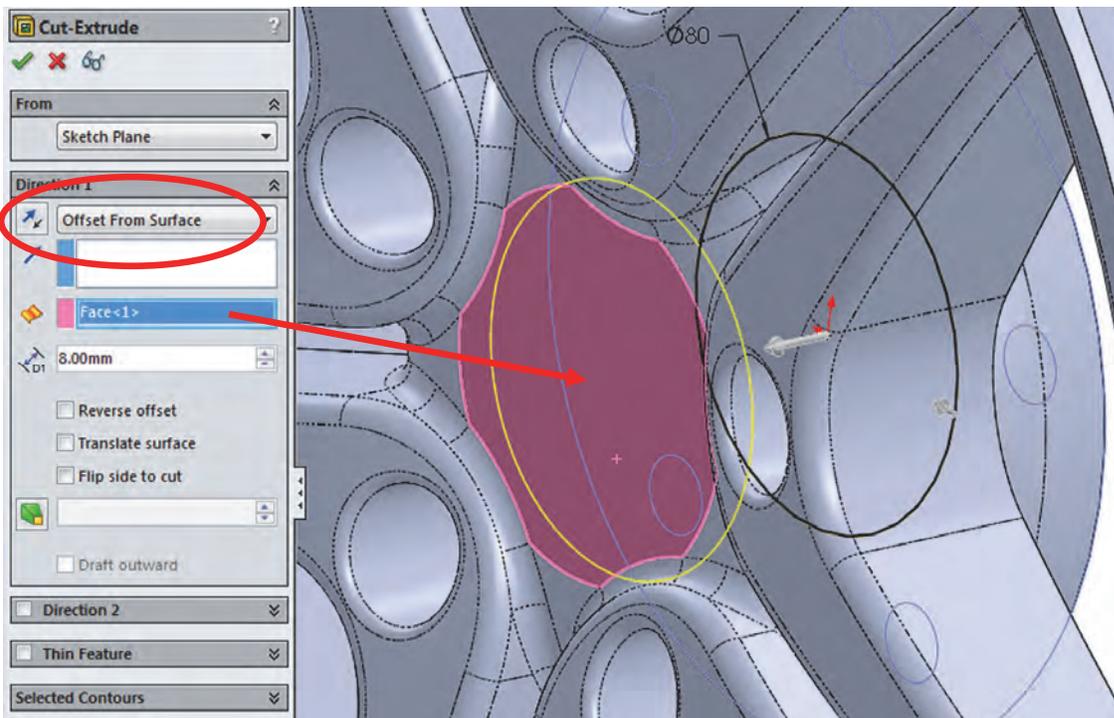
36. – Add a new 2.5mm fillet to the bolt holes and the connections between the spokes and the rim. Use the selection toolbar if needed.



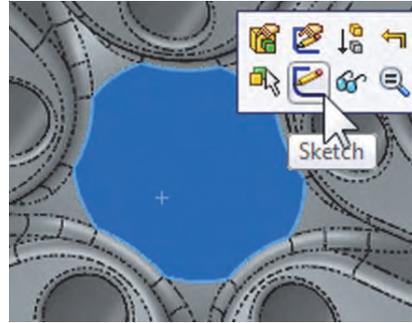
37. – To make the wheel lighter, add an 80 mm diameter hole in the rear of the hub. We want to make the cut deep enough to leave 8mm thick at the bottom. To accomplish this we'll use an end condition called "Offset from Surface."



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 included files locate the image file 'logo.png' for use in this step.



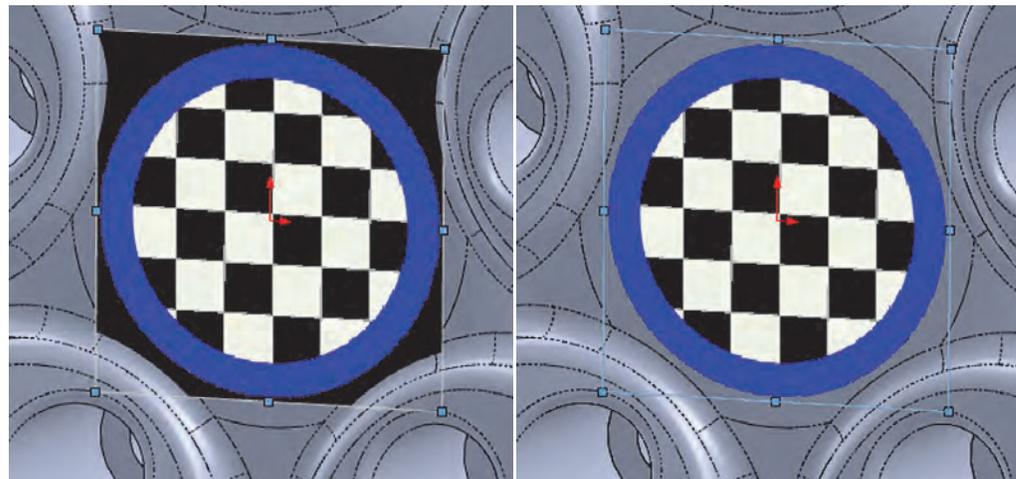
Add a new sketch in the front face of the wheel, where the logo will be located. Go to the menu **“Tools, Sketch Tools, Sketch Picture.”**

Browse to the location where the logo image was saved and open it. The image is automatically 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 enter the image values in the “Properties” box. 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.



After adding the Sketch Picture we can edit it by double clicking on it in the FeatureManager.

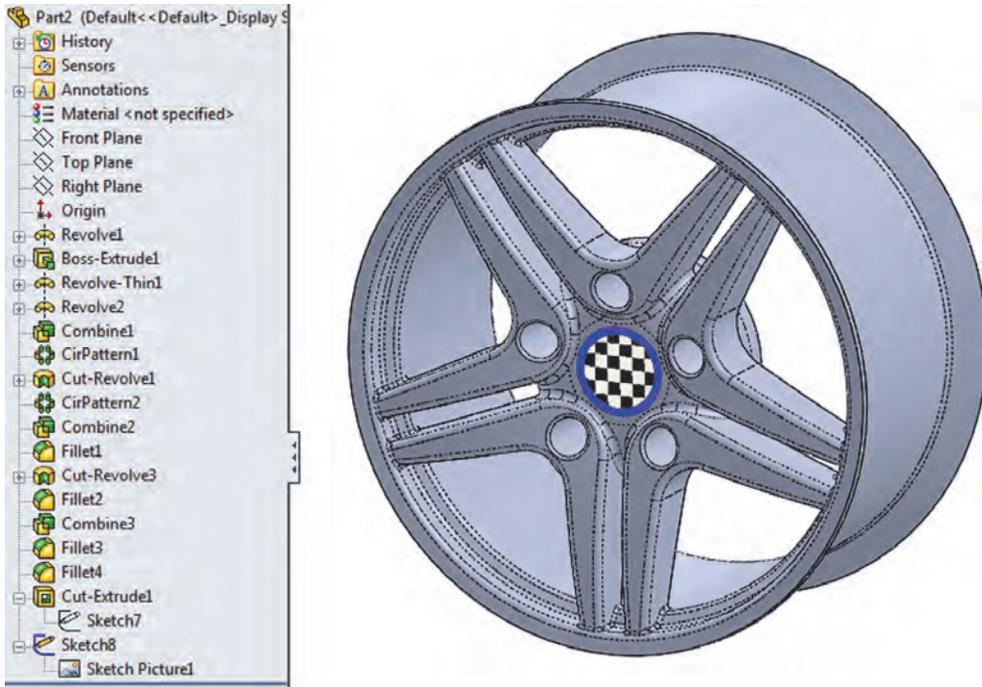


Transparency None

From file

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

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 it can be hidden by hiding the sketch.

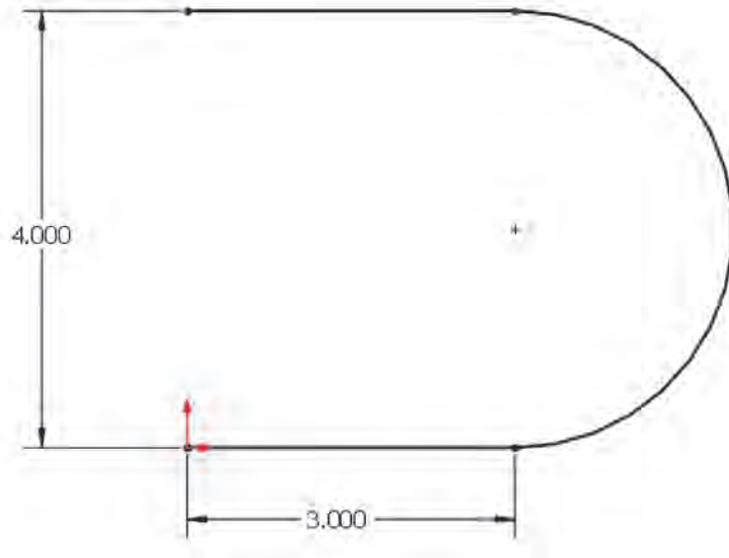


Feel free to add your favorite automobile logo. Due to copyright restrictions we cannot use registered logos in this book, but we can confirm that there are some that look VERY good!

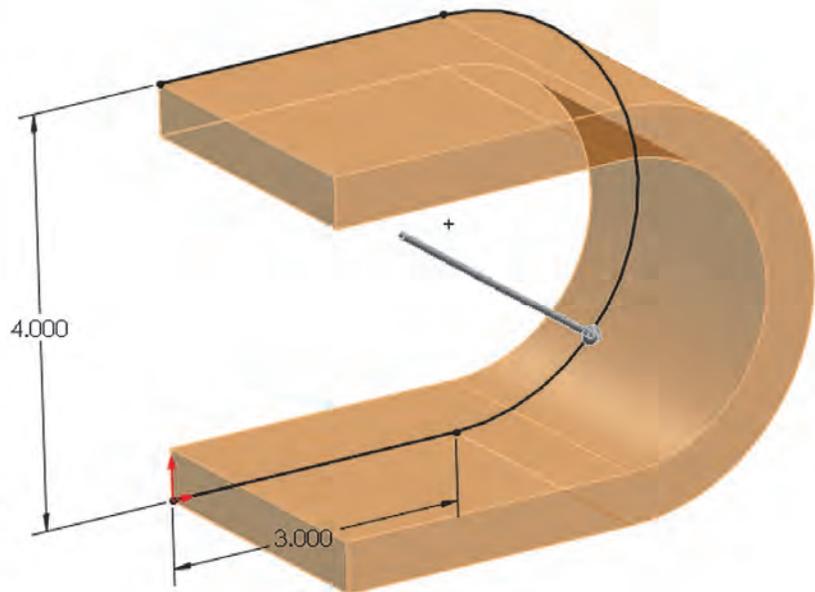
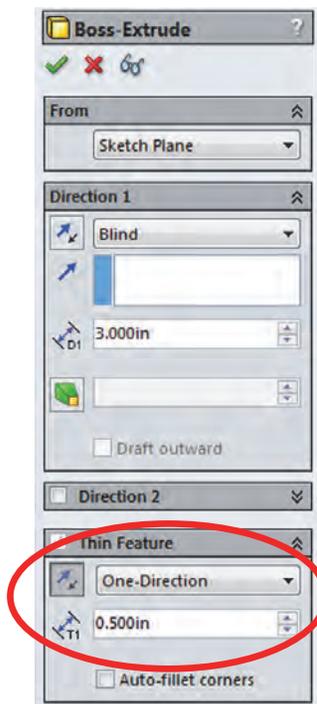
This is a good example to show how to work with multi body parts and perform local operations. Save the part and close it.



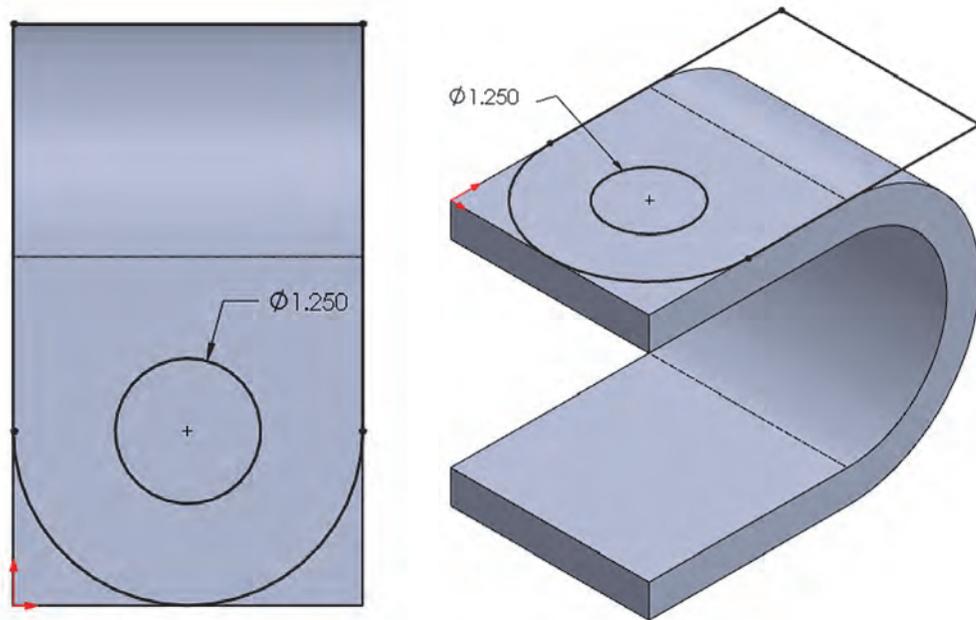
40. – The next area to cover in the multi bodies topic will be to learn more about combining bodies. The other two operations we can do when combining bodies is to subtract one or more bodies from another, or get the common volume between them. As we saw in the previous example, to obtain the common 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.

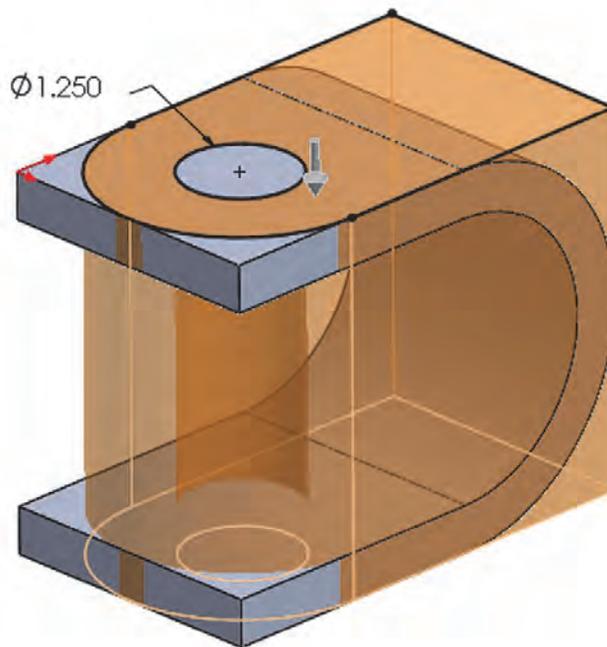
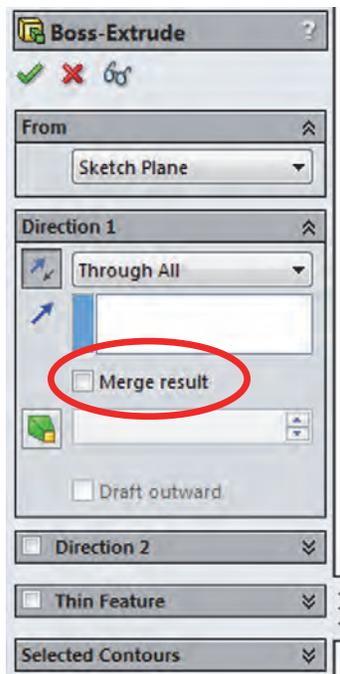


42. – For the next feature, add a sketch in the top face of the first feature. Looking at it from a top view should look like this:

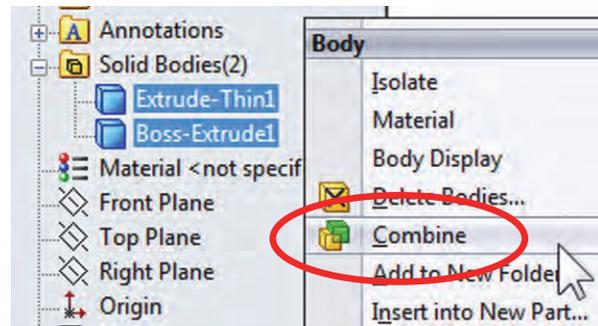


Notice the only dimension we need to add is the hole diameter; everything else is defined with geometric relations (Coincident and Tangent relations).

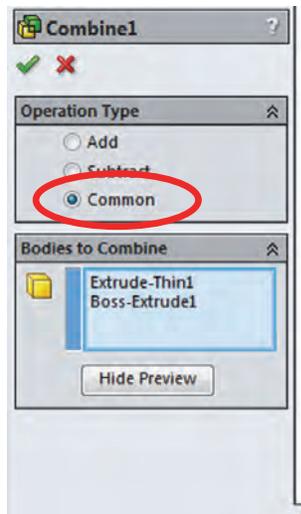
43. – Extrude the second sketch downwards with the “**Through All**” end condition, and uncheck the “**Merge result**” checkbox. We want to have two overlapping bodies.



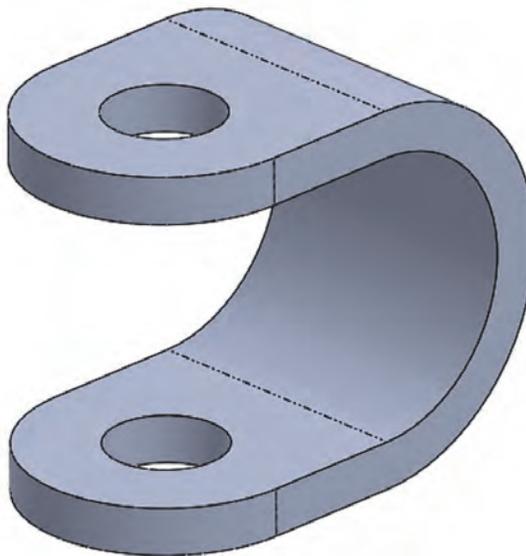
44. – Now that we have two separate bodies, we can combine them to get the common volume. Select the menu “**Insert, Features, Combine**” and select both bodies, or from the “*Solid Bodies*” folder in the FeatureManager pre-select both bodies; right-mouse-click and select “**Combine**” from the pop-up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



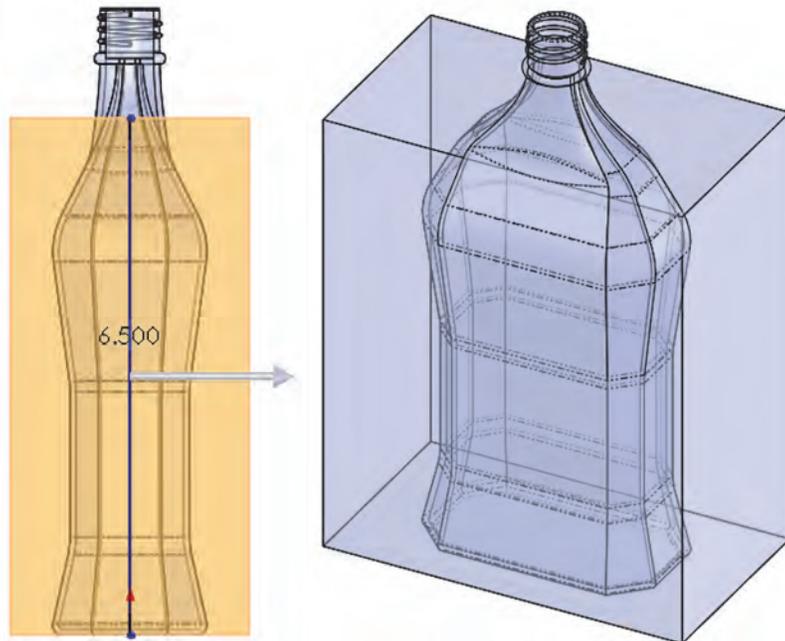
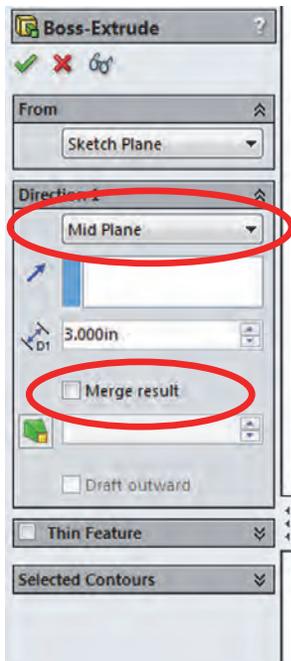
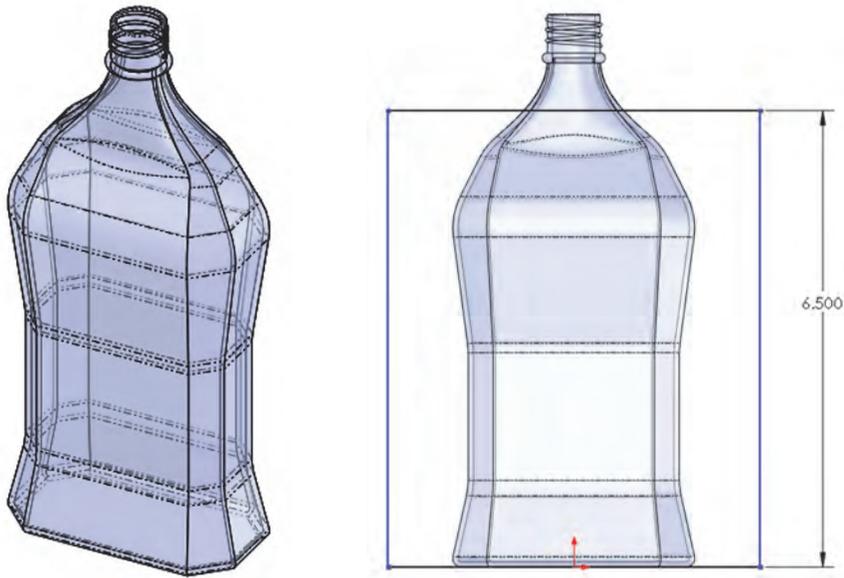
45. – In the Combine operation select the “Common” option under “Operation Type.” Click in the “Show Preview” button to see what the resulting body will look like and click OK to finish.



The finished part will look like this.

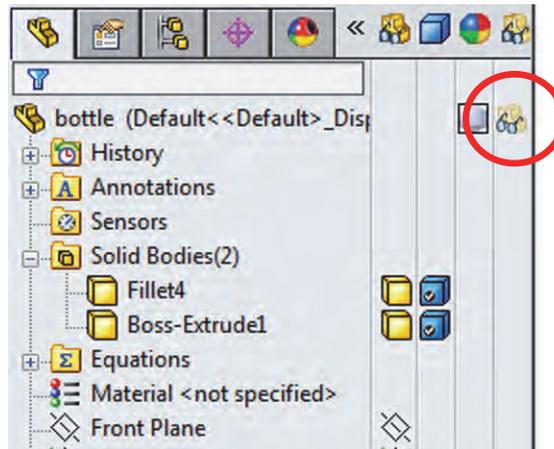


46. – So far we have covered adding bodies and extracting their common volume. The next multi body operation we'll cover is the difference between bodies. One common use for a body difference is to obtain a mold's core and/or cavity (later in the book), but here we'll show how to obtain the volume capacity 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.

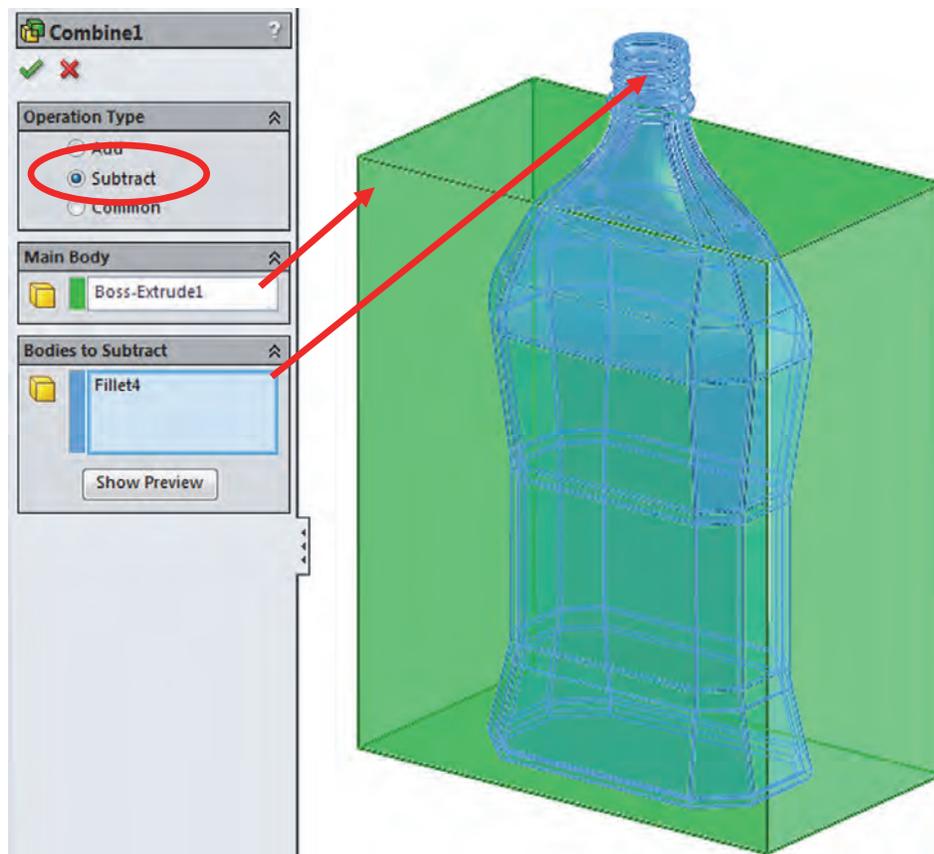




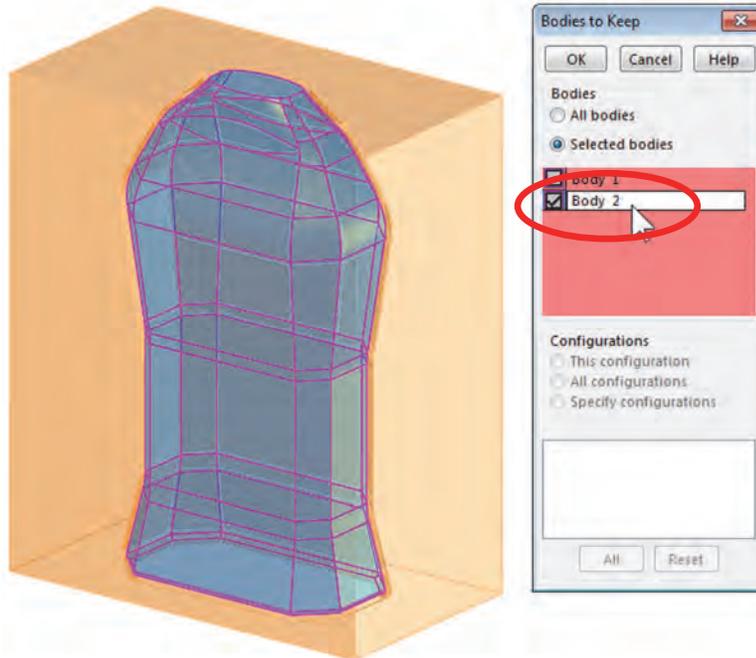
To change a part's transparency use the **“Display Pane.”** Activate the transparency at the part level (can also be done to each body individually).



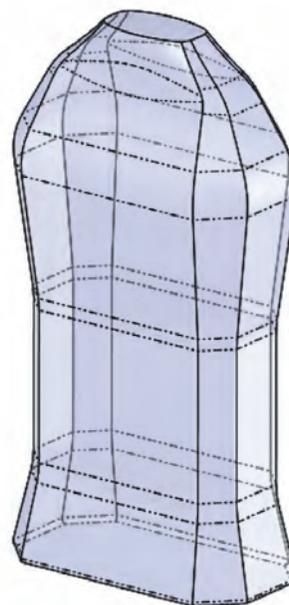
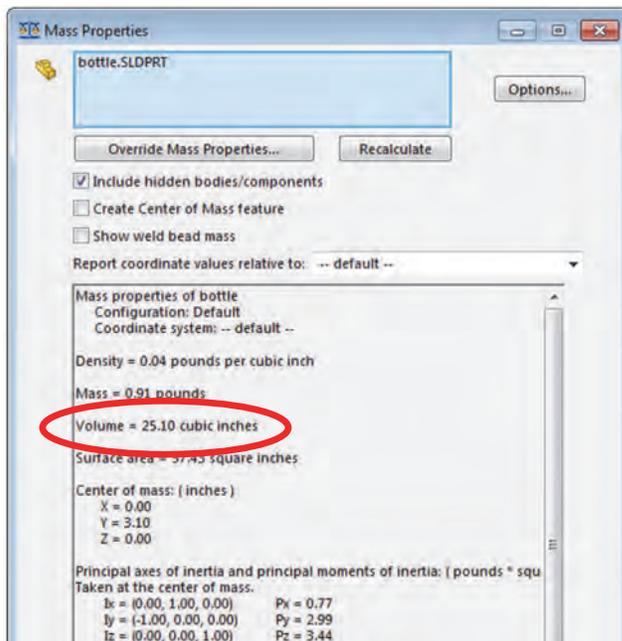
47. – Now that we have the two solid bodies, select the menu **“Insert, Features, Combine.”** Under **“Operation Type”** select the **“Subtract”** option. In order to get a difference, we need to select the body that we want to remove material from (Main Body) and the body (or bodies) that we want to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



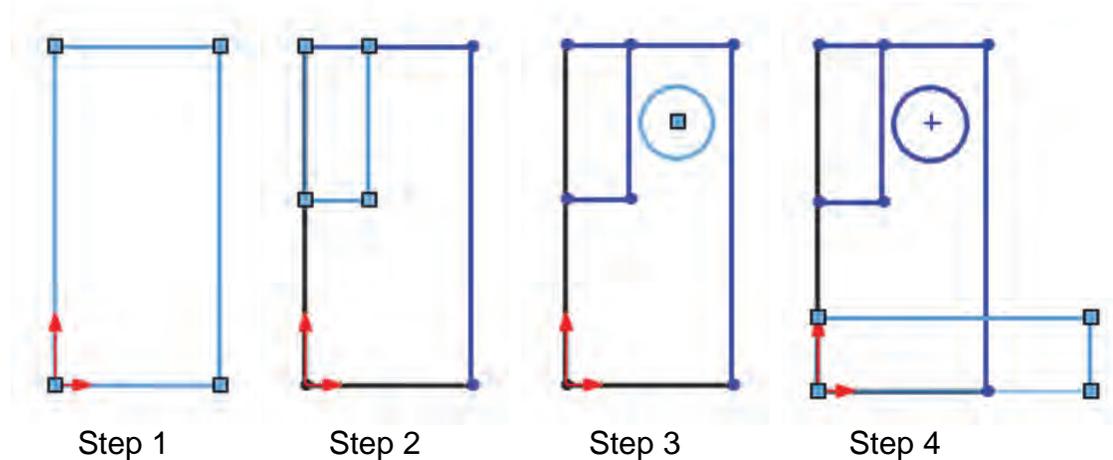
49. – Now we have the actual volume of the bottle up to the fill line. Notice that when we subtract one body from another, the original bodies are consumed and we are left only with the difference. After running a “**Mass Properties**” analysis, we can see that the volume of liquid inside the bottle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



Contour Selection

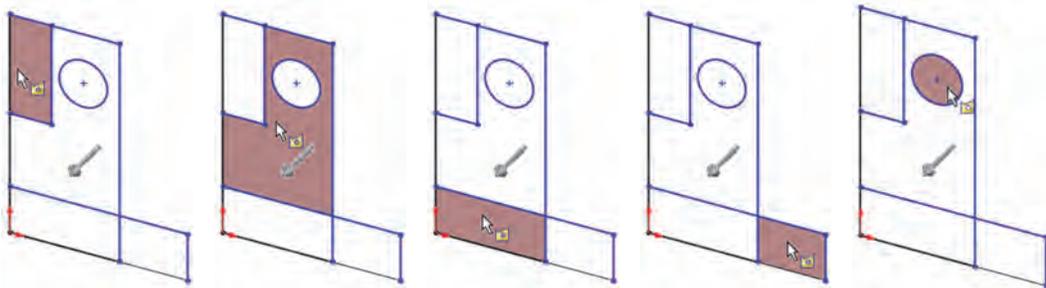
Contour Selection is a way to work with a sketch that has intersecting entities, endpoints shared by multiple entities and many of the common problems that would otherwise prevent us from using a sketch for a feature. We'll also learn how to reuse a sketch for multiple features and a previously unused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

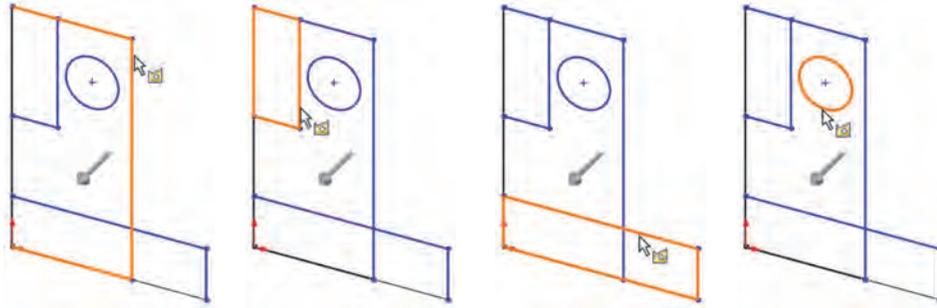
50. – Make a new part and add a sketch to the “*Front Plane*” as shown. In this example we will not worry about dimensions to simplify the explanation and concentrate on how contour selection works.



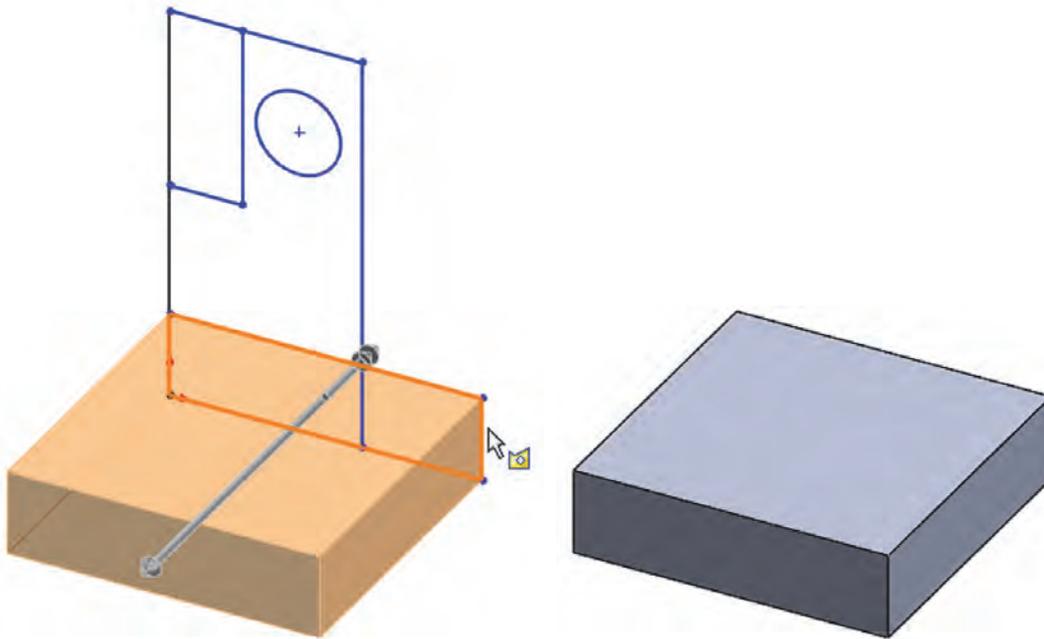
In this sketch we are drawing three rectangles and one circle. Note that the rectangles are overlapping and sharing endpoints.

51. – 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 by selecting their perimeter.

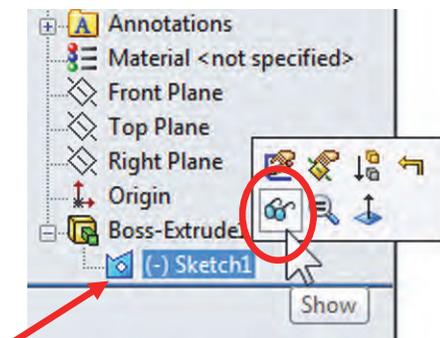




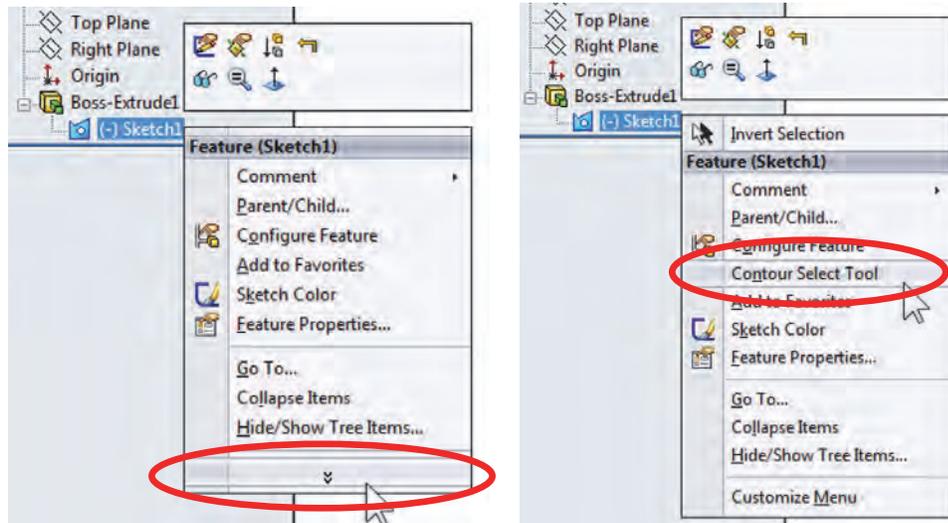
52. – Select the bottom square profile (or both bottom regions) and extrude *approximately* as shown. Just like with any other feature the sketch is automatically hidden after we finish. Do not worry about size at this point.



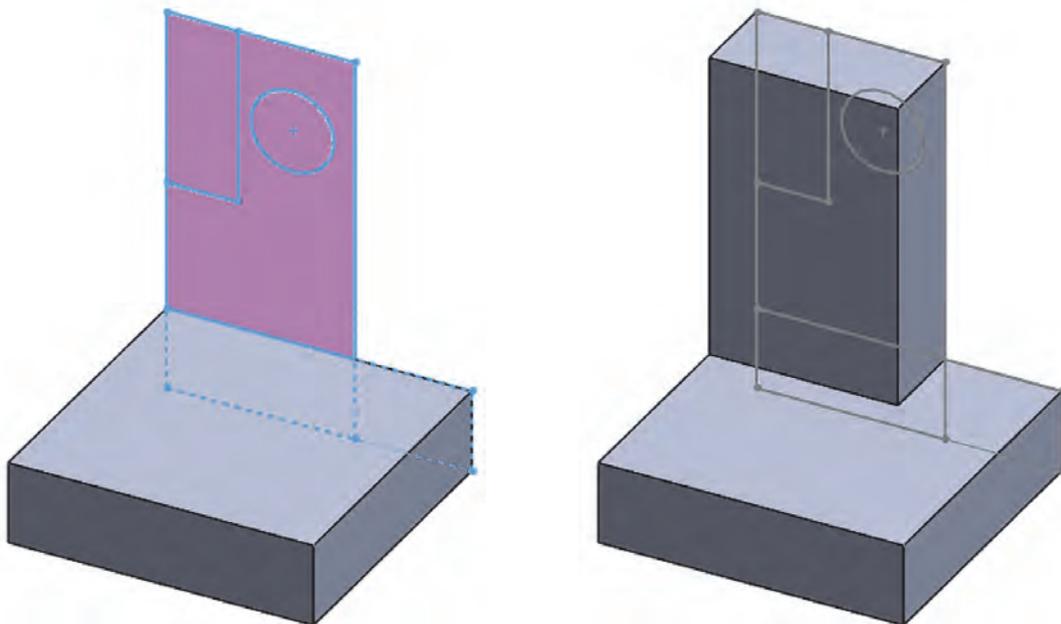
53. – To re-use the sketch for more features, expand the “Boss-Extrude1” feature, select the sketch in the FeatureManager and click “Show” in the pop-up menu. Note the “Contours” icon next to the sketch’s name to let us know the feature was made using contours.



54. – We will now re-use the sketch from the first extrusion to create 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.

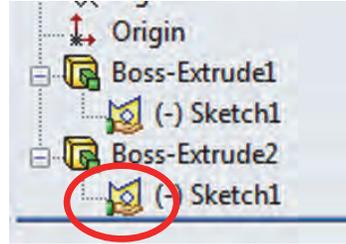


55. – After activating the “**Contour Select Tool**” select the regions indicated and extrude *approximately* as shown. You may have to click in the sketch to enable (activate) selection of regions, and either: hold down the Ctrl key to pre-select all three regions and then extrude, OR select the Extruded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

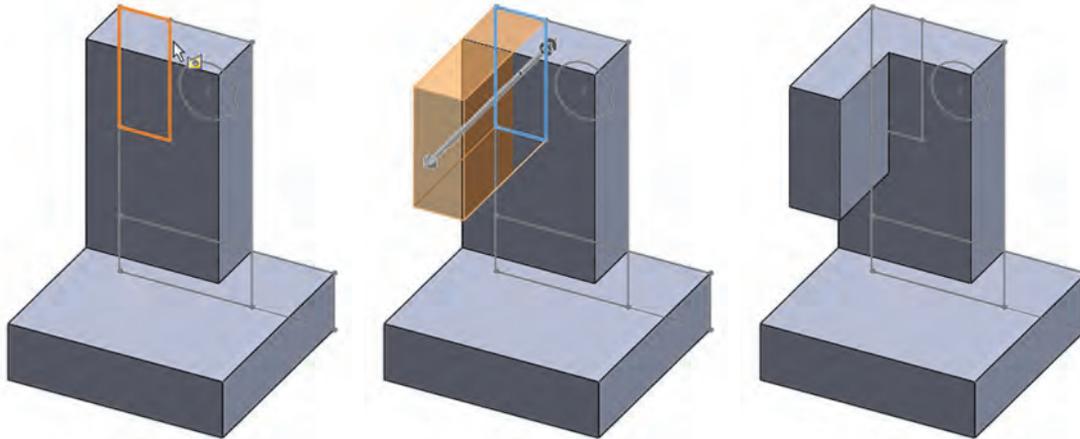




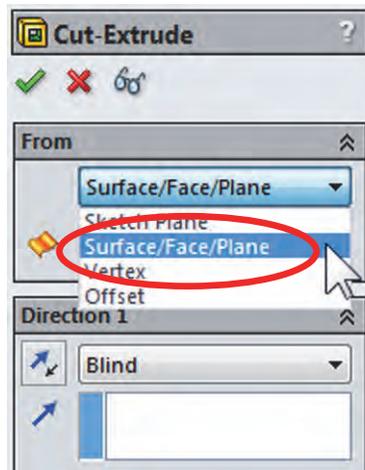
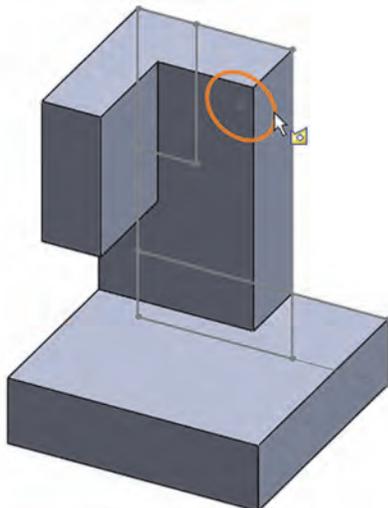
After using the same sketch for two or more features we can see a slightly different icon next to the sketch name with a little hand under it. This means the sketch is “shared” by more than one feature, and the sketch name is the same. Also note that the sketch remains visible after we use the “**Show**”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



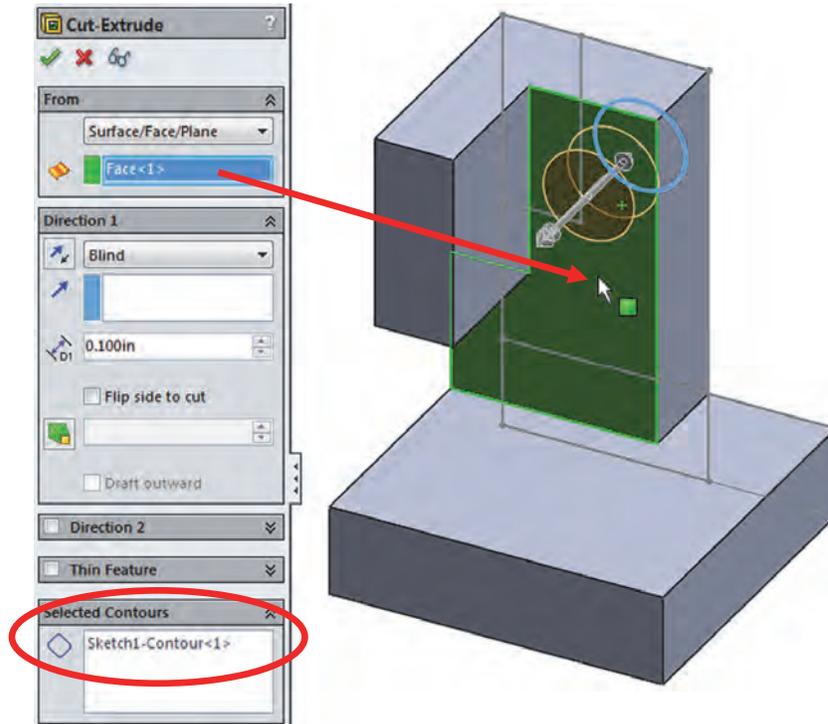
56. – Repeat the previous process to select the next contour and extrude as shown. Remember at this time we are only showing how it works and are not concerned about the dimensions.



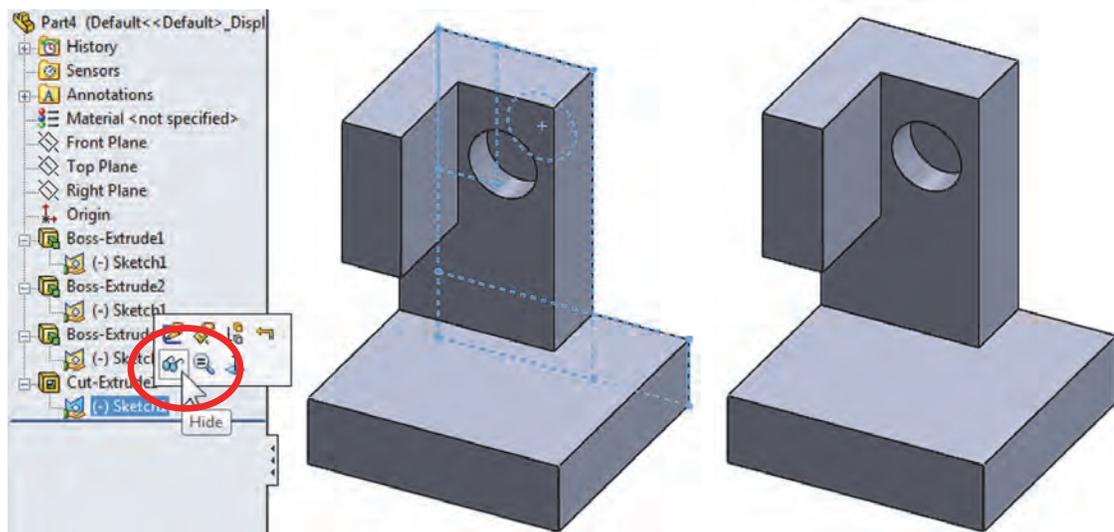
57. – For the final feature select the “**Contour Select Tool**” and select the circle. What we have done up until now is to make features that start at the sketch plane. SolidWorks has a powerful (yet sometimes under used feature) that allows us to start the feature somewhere *other* than the sketch plane. After selecting the circle using “**Contour Select Tool**”, 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 an offset distance from the sketch plane.)



58. – After we select “Surface/Face/Plane” a new selection box is revealed to select where we want the feature to start. Select the front face of the second extrusion as indicated to start the cut feature in this face. 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.



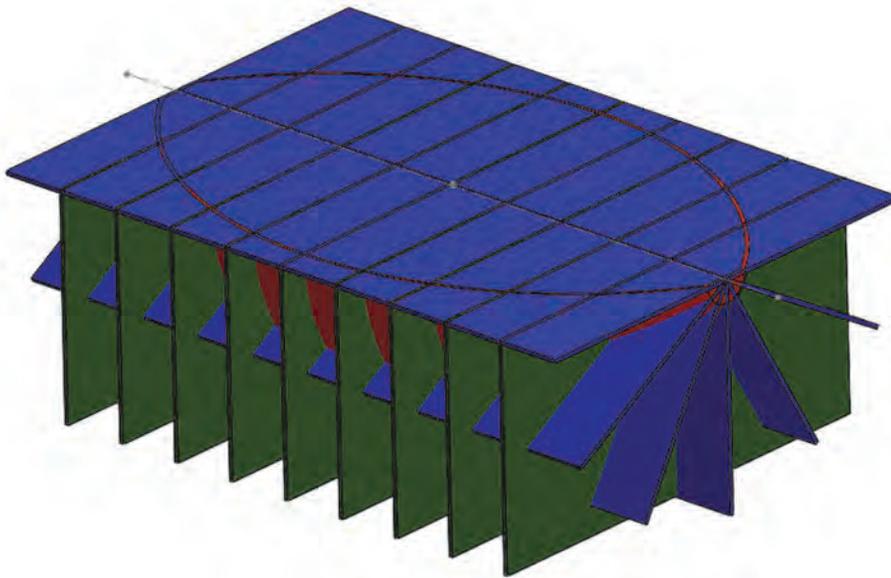
59. – Now that we have made four features from the same multiple contour and self-intersecting sketch, we can hide it. Save and close the file.



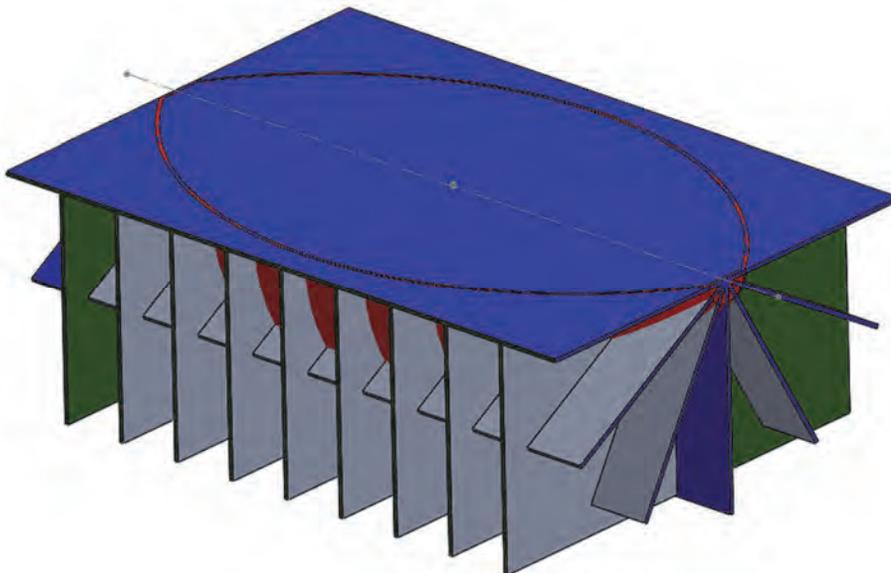
Exercises: Build the following parts using the knowledge acquired in this lesson. Try to use the most efficient method to complete the model. Open the required files from the accompanying disk or download them from our website.

Multi Body Exercise

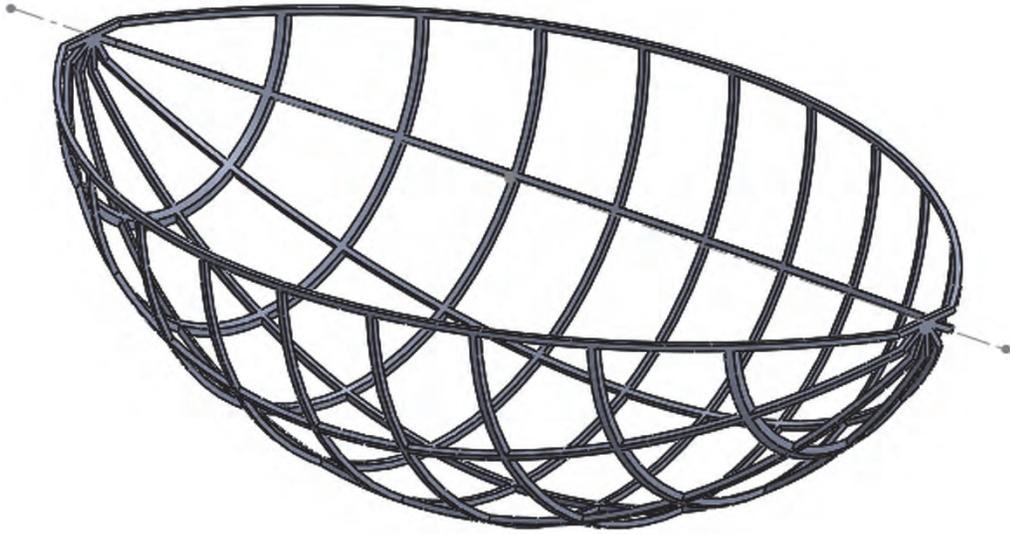
Open the part '*Multibody Exercise.sldprt*' to build a mesh basket using a multi body part. Make a circular body pattern of the "Extrude-Thin1" feature (Blue body) using the centerline in the "Circular Pattern Axis" sketch with 7 instances spaced 30 degrees. Add a linear body pattern using the "Extrude-Thin2" (Green body) with 9 instances spaced 1".



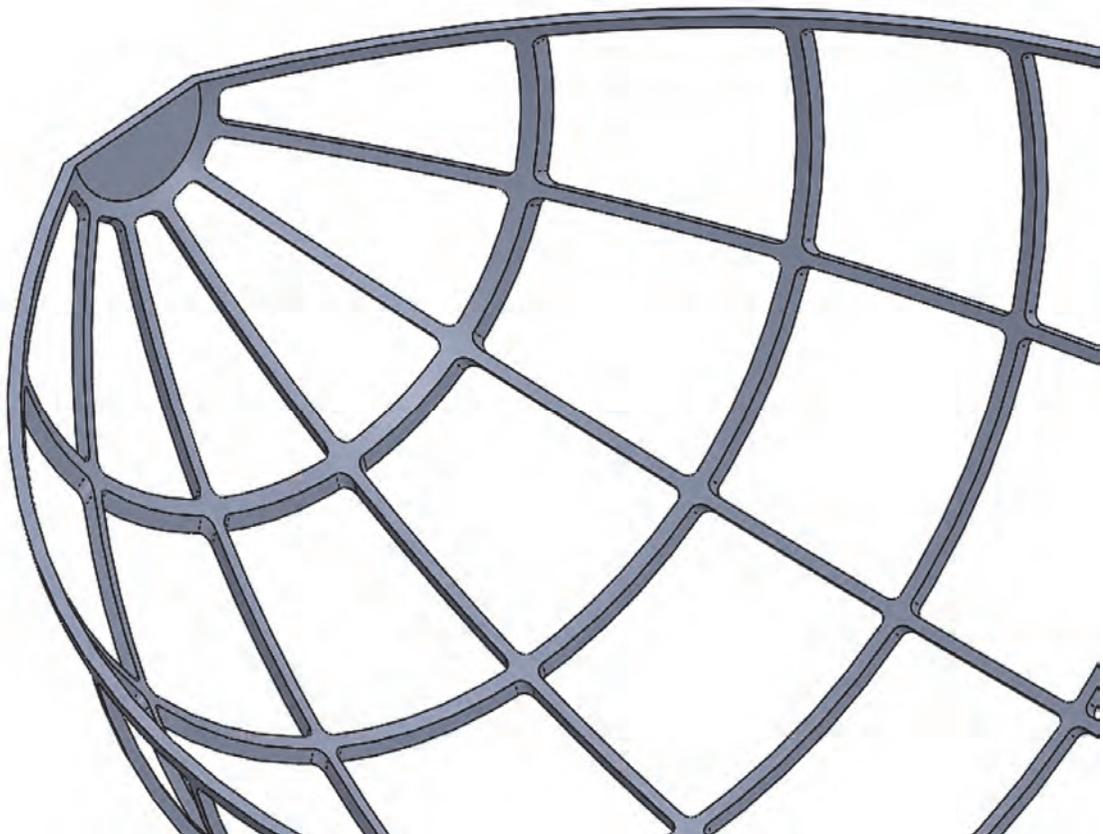
Combine (Add) all the bodies except the "Revolve-Thin1" (Red body) to make a single body with them.



Finally combine the remaining two bodies to get their common volume.

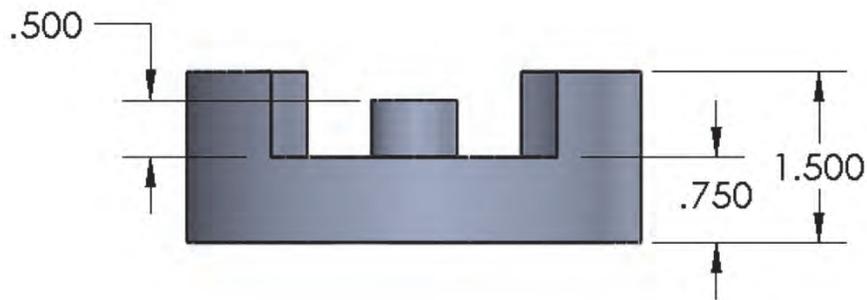
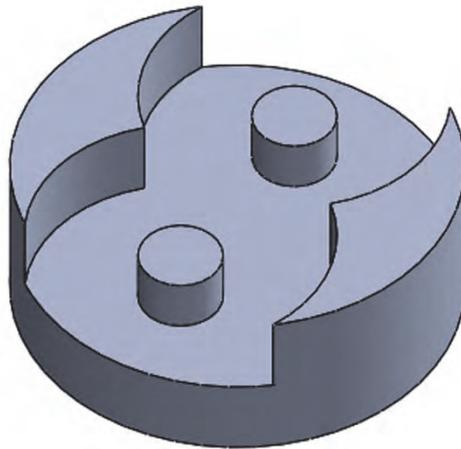
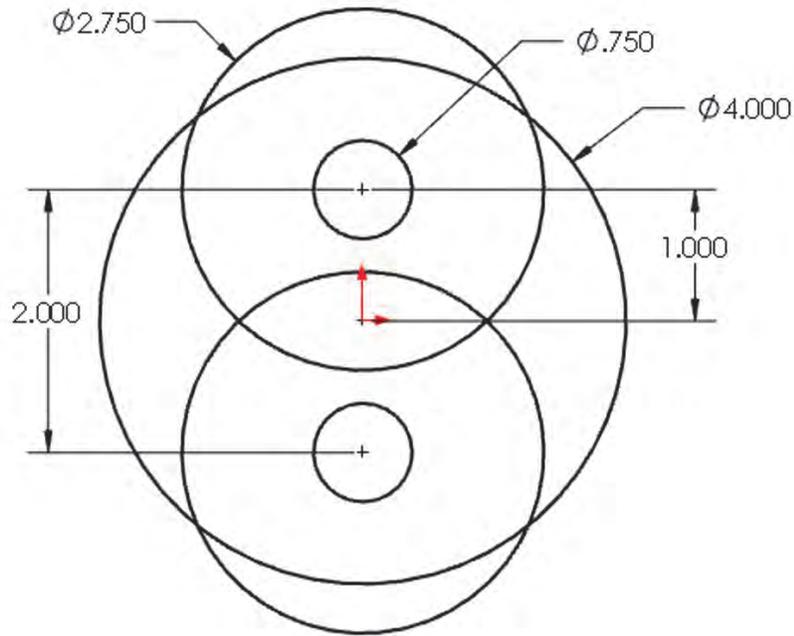


As an optional finishing touch, add a reinforcement in the ends of the basket and fillet the intersections with a 0.050" radius. TIP: Use the expanded selection tool to automatically select all the small edges.



Contour Selection Exercise

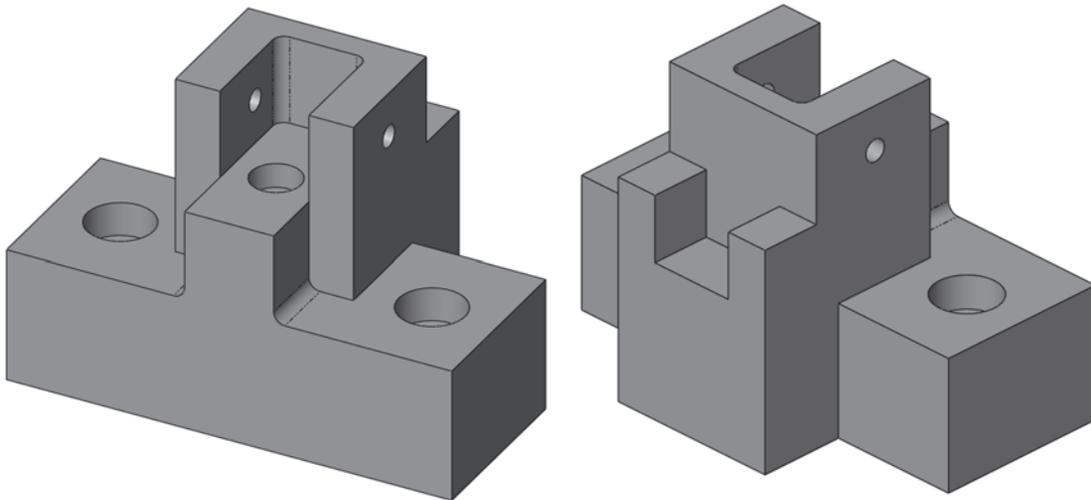
Make the following sketch and make all four features off of it using the “Contour Selection” tool.



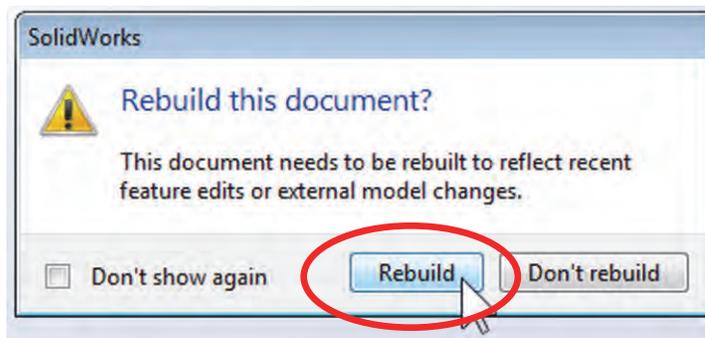
Part Editing

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. For the most part those are the types of errors we are talking about.

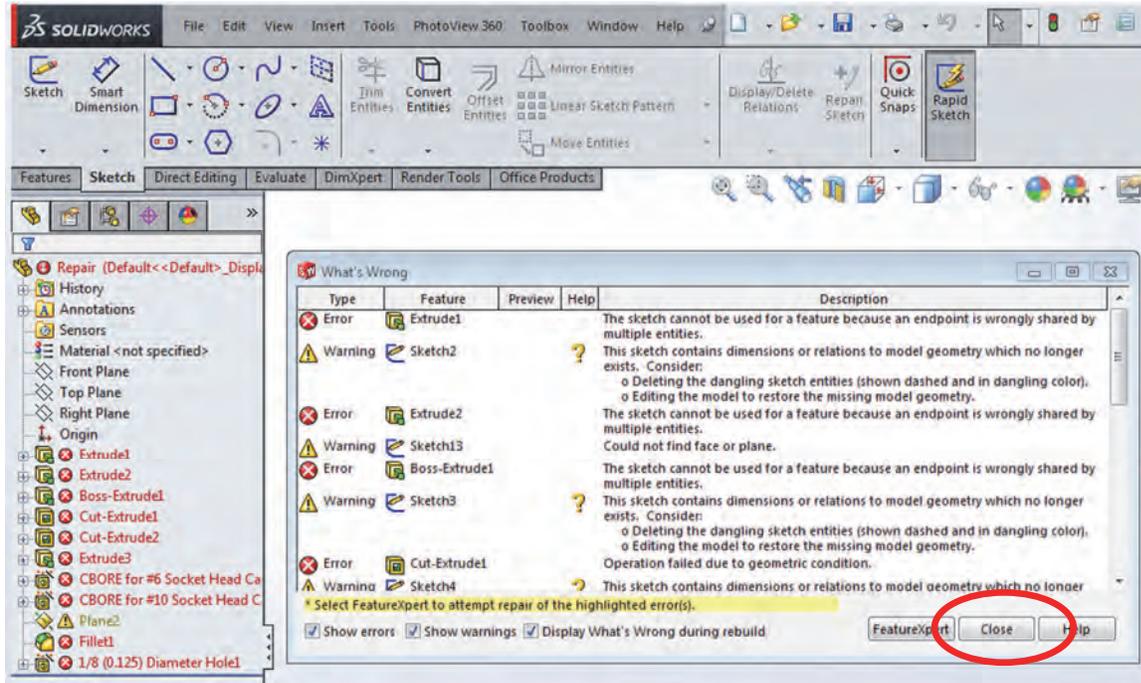
60. – To practice editing models and fixing errors, open the part *'Repair.sldprt'* from the included files. After we are done modifying and fixing the part, it will look like this:



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



62. – When the model is rebuilt we see a list of errors and no geometry at all.

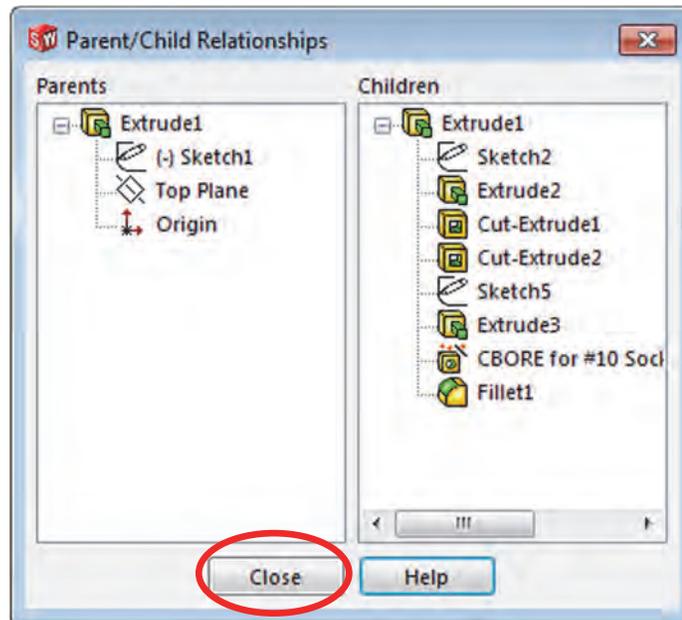


63. – This part has so many errors that no geometry can be generated, and the “What’s Wrong” dialog box contains the full list of things that need to be fixed. Click the “Close” button for now. Since features are added chronologically starting at the top in the FeatureManager, the logical order to start fixing errors is from the top and work our way down. The reasoning behind this order is that, 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, right-mouse-click in the “Extrude1” feature and select “Parent/Child.”

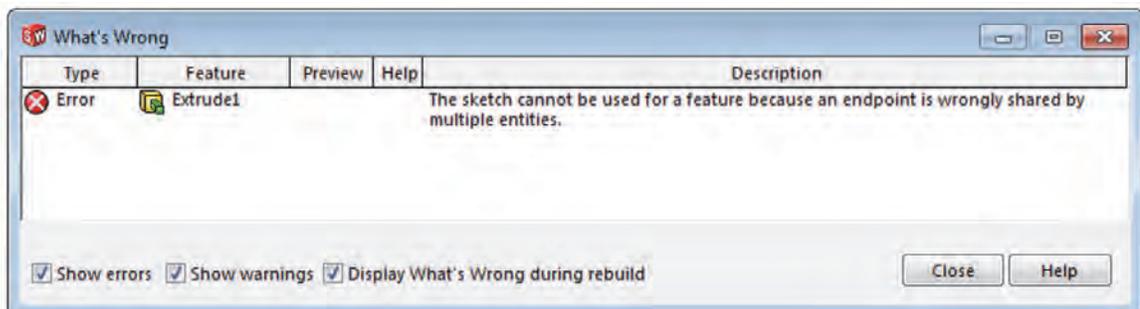
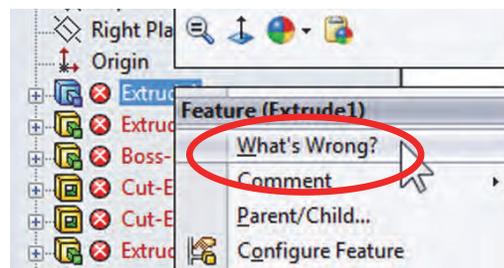


There are two types of errors: a red X means the feature failed to build; the yellow warning triangle means the feature has an error, but SolidWorks was still able to build it.

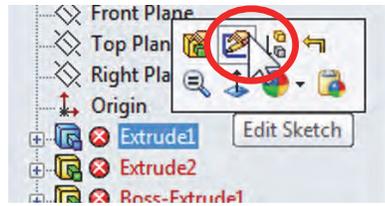
64. – Here we can see the features that “*Extrude1*” depends on (Parents), and which features depend on it (Children). The higher we go in the FeatureManager, the more Children a feature may have. Click “Close” to continue.



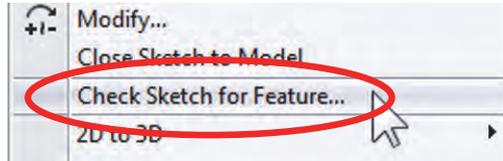
65. – To start fixing errors, we’ll check what the error at the “*Extrude1*” feature is. Right mouse click on it and select “**What’s Wrong?**”



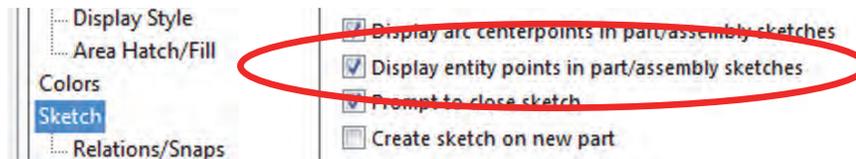
This error means that the sketch has intersecting lines and/or two or more lines are connected to the same endpoint. One common cause of this problem is when sketching we accidentally add overlapping lines. Close the “What’s Wrong” dialog and edit the “*Extrude1*” sketch.



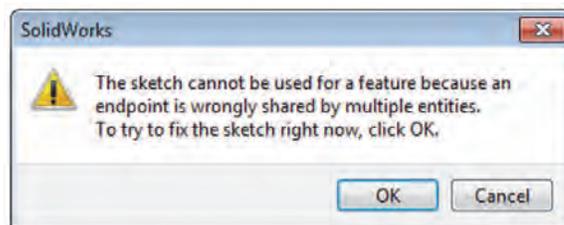
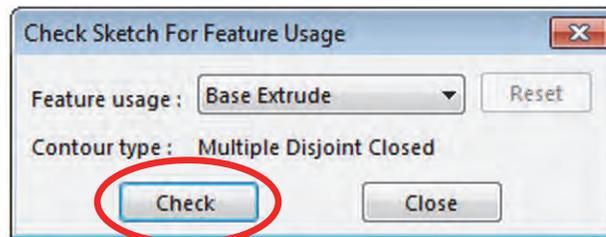
66. – Sometimes it's easy to see the geometric elements causing the problem in a sketch and we can correct it, but sometimes it's not that obvious. To help us identify the problem, select the menu **"Tools, Sketch Tools, Check Sketch for Feature."**



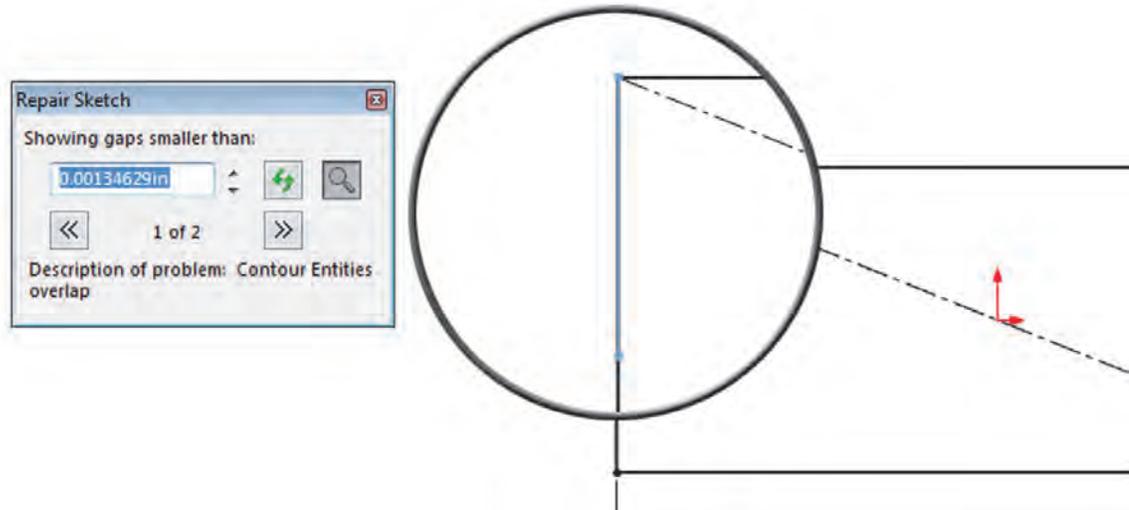
Setting the option to show sketch endpoints is usually a good idea. Go to the menu **"Tools, Options, System Options, Sketch, Display entity points in part/assembly sketches"** to identify overlapping geometry



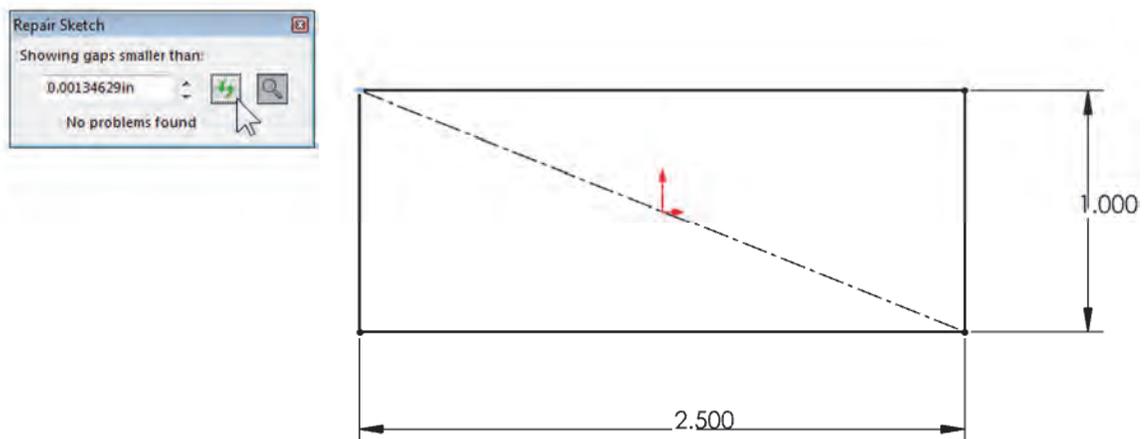
If the sketch has already been used for a feature, the feature type will be pre-selected in the drop-down menu. If the sketch has not been used for a feature we have to select the type of feature that we intend to use it for. In our case "Base Extrude" is pre-selected. Click on "Check" to analyze the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



67. – 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 problem is that we have overlapping entities, and the second, multiple (2 or more) elements sharing the same endpoint.

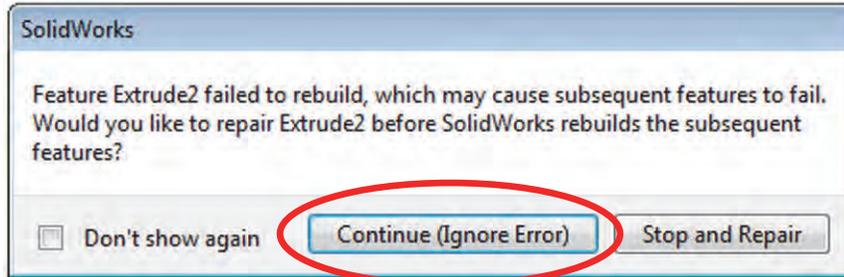


68. – The advantage of using this tool is that we can quickly identify where the problem is and correct it, instead of hunting down small line segments in a sketch. Usually, small line segments would be pre-selected to show them and, if that line is not part of our design, we can delete it. If no line is pre-selected, 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.

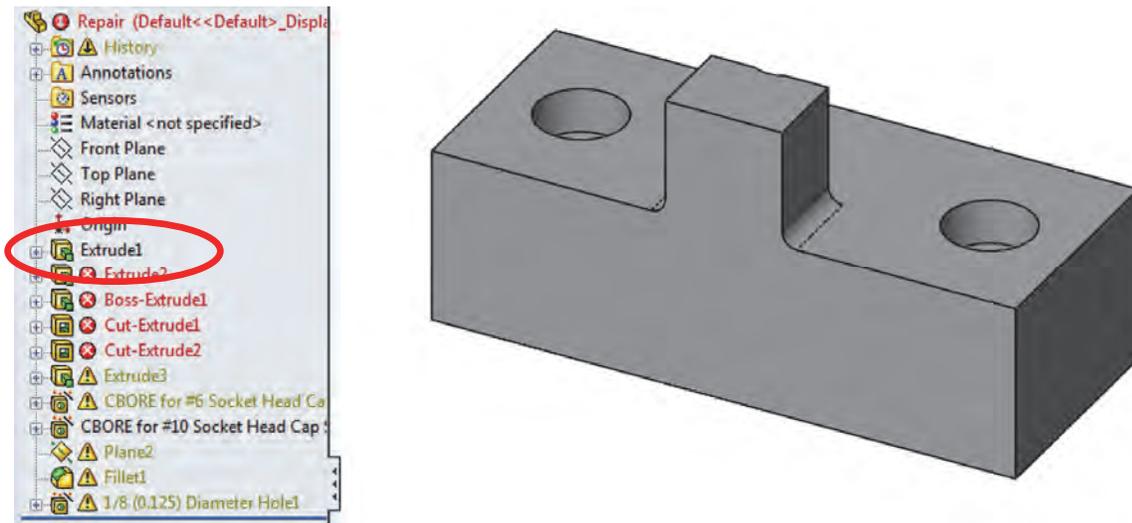


Window-selecting from left to right selects geometry completely enclosed by the window; window-selecting from right to left selects geometry crossed by the window.

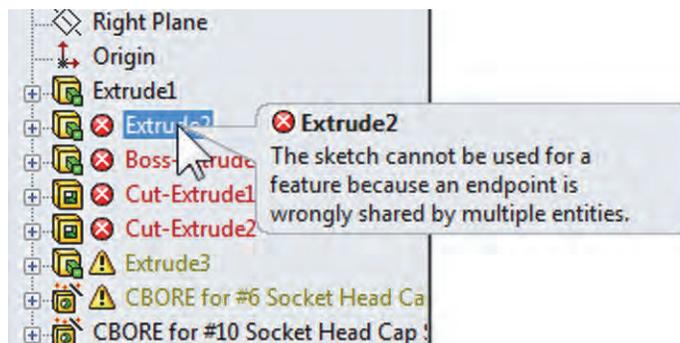
69. – Exit the sketch (or rebuild the model) to continue. We are notified that a subsequent feature has an error and we are asked if we wish to repair it now or continue with the error. Click on “Continue (Ignore Error)” and also close the “What’s Wrong?” message.

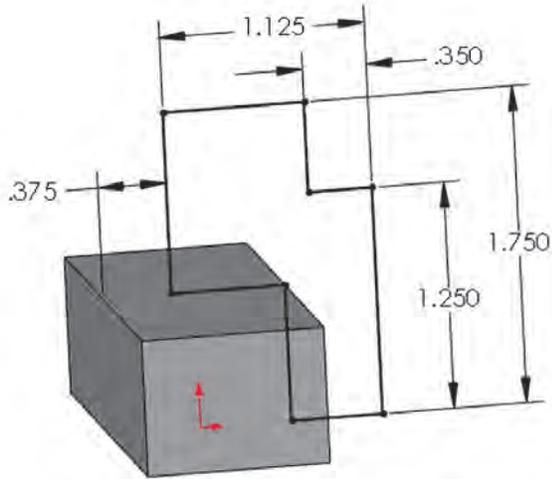
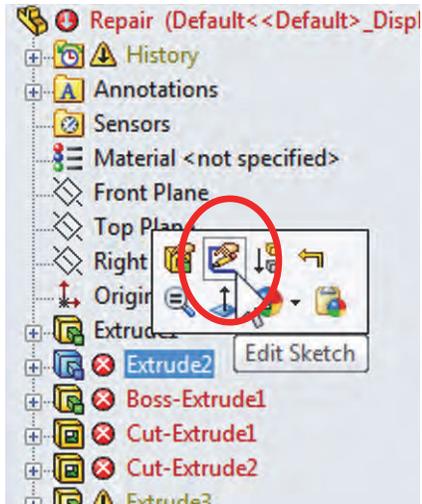


We still have some errors, but now we can see some features and (more importantly) the error from the first feature is gone.

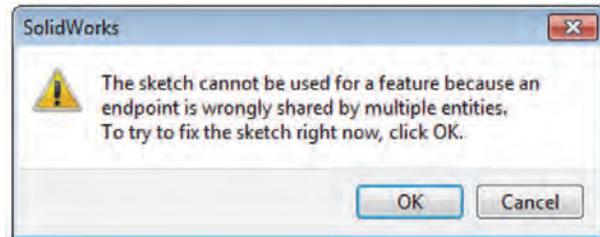


70. – Select the “Extrude2” feature, right-mouse-click and select “What’s Wrong?” to view the error as before, OR rest the mouse pointer on top of it and wait to see the error in the pop-up bubble. Essentially we have the same problem as the first feature. Edit the “Extrude2” sketch to fix it.

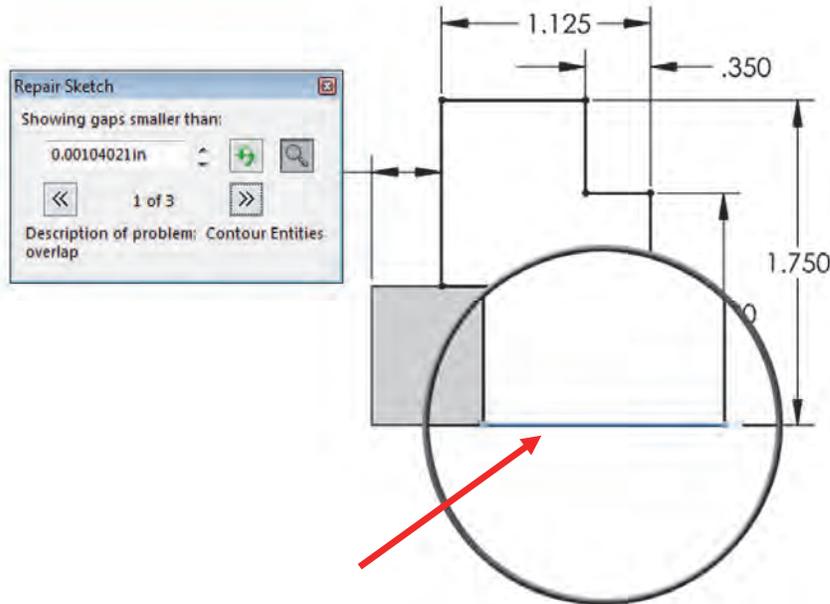




71. – Once we are editing the sketch, select the menu “**Tools, Sketch Tools, Check Sketch for Feature**” to find out what the problem is. Dismiss the error message to continue.

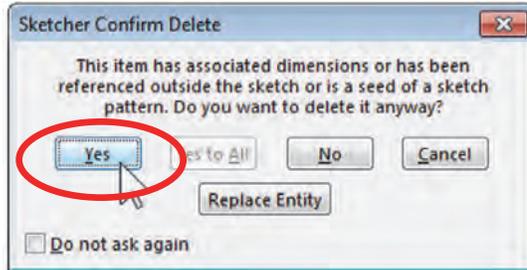


In this case, looking at all three problems listed in the “Repair Sketch” dialog (one overlapping entity and two with more than two entities at an endpoint), we can tell that we have two identical lines overlapping. Close the “Repair Sketch” dialog and delete one of the lines.

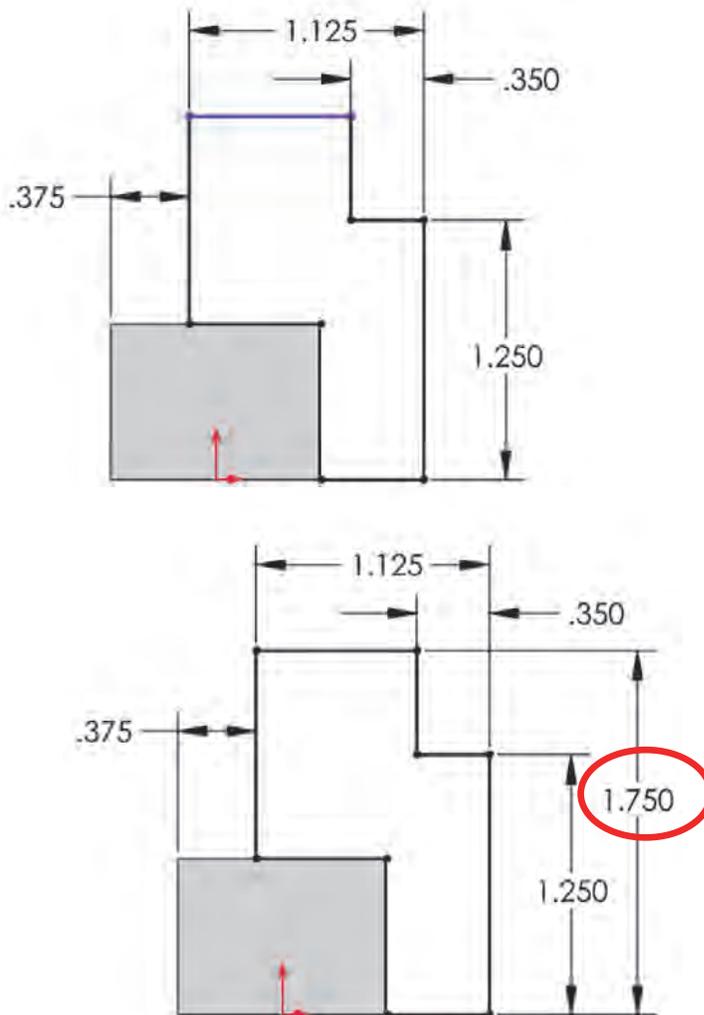
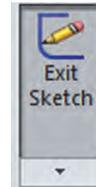


If we use window-selection we'll delete both lines; if we just click to select the line, only one line is selected.

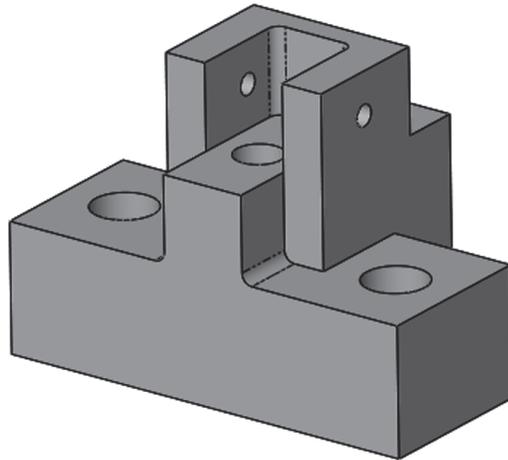
72. – After selecting one of the overlapping lines and deleting it, we may be warned that other entities will also be deleted, most likely dimensions attached to the line being deleted. If this is the case, click “Yes” to continue.



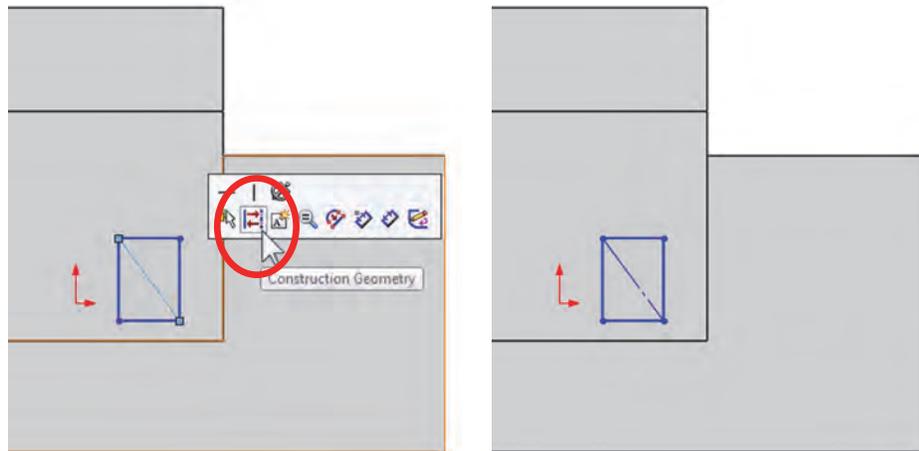
73. –Notice the top horizontal line’s dimension was also deleted. This is because it was referencing the line we deleted. Manually add the missing dimension again and exit the sketch. Select “Continue (Ignore Error)” after we exit the sketch and dismiss the “What’s Wrong?” message.



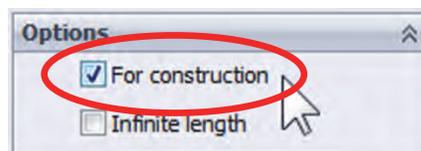
74. – Now our model is starting to look better. We have only fixed two errors and we have cleared many errors, better illustrating the importance of understanding parent/child relations.



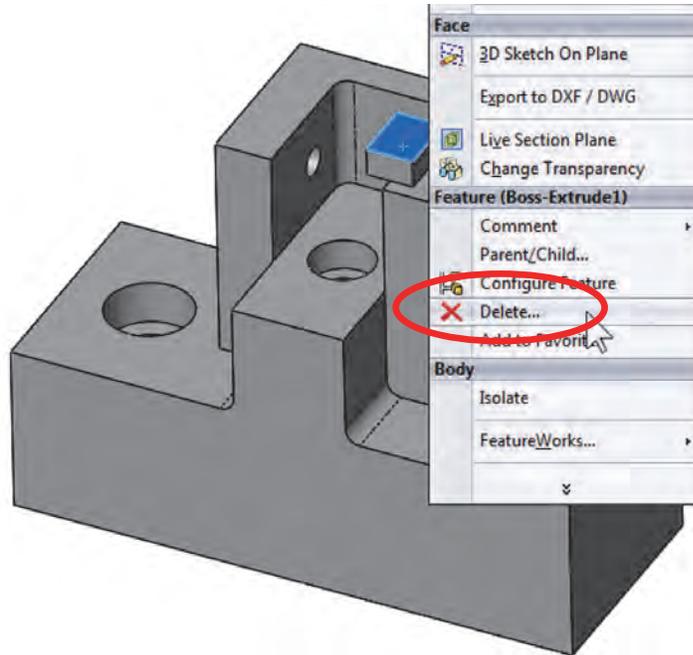
75. – After reviewing the next error with “**What’s Wrong?**” we see that it’s the same error as the two previous features. Editing the sketch we can see an extra diagonal line. We can either delete it, or convert it to **Construction Geometry**. The second option is usually safer, as we could lose dimensions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



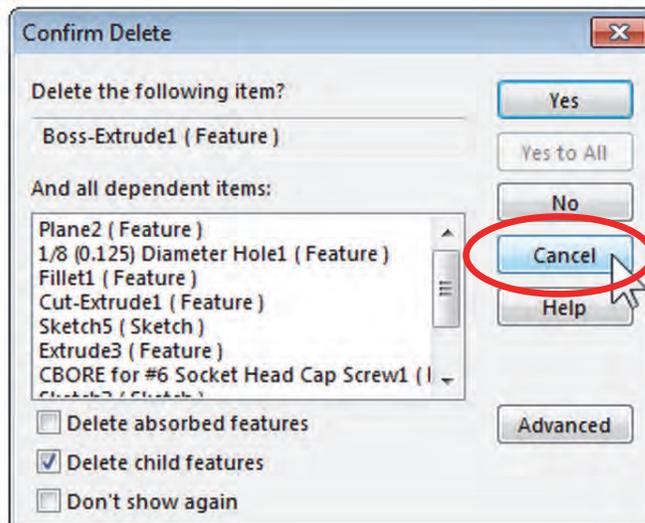
Optionally we can change the line (or any geometry element) to construction geometry by turning on the “For construction” checkbox in the Property Manager after selecting it.



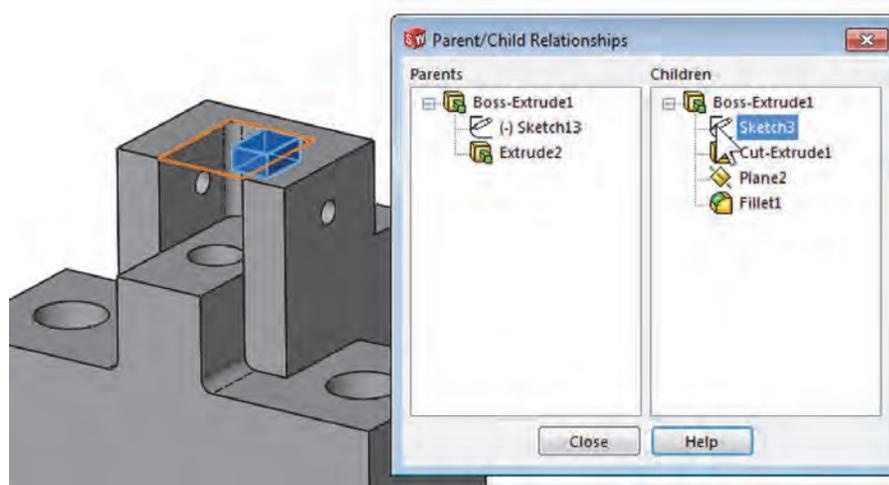
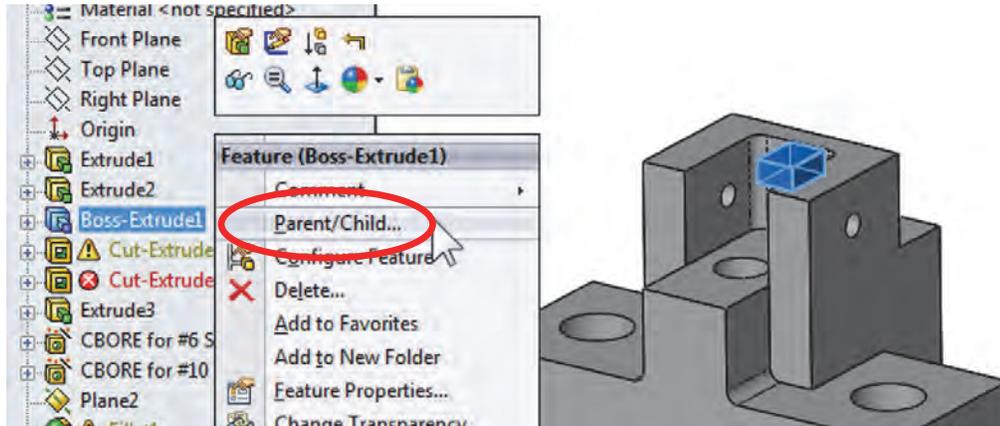
76. – Exit the sketch to rebuild the part dismissing the error dialog to continue. Since the newly created feature does not seem to be part of the original design, the logical step would be to delete it. Select the feature with the right-mouse-button, and after selecting “Delete” we get a “Confirm Delete”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



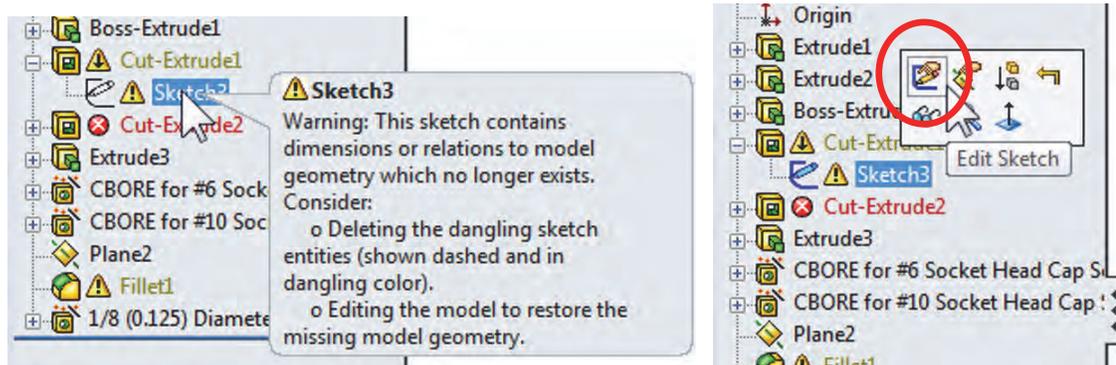
77. – In the confirmation we can see that if we delete this feature we would also delete its dependent features, all of which need to be in our part. The reason those features would also be deleted is because they are “children” features of “Boss-Extrude1.” Activating the “Delete child features” checkbox shows all the features that would be affected. Select “Cancel.” Since we don’t want to delete any of these features, we’ll edit those features to remove the dependencies *and then* delete this extra feature.



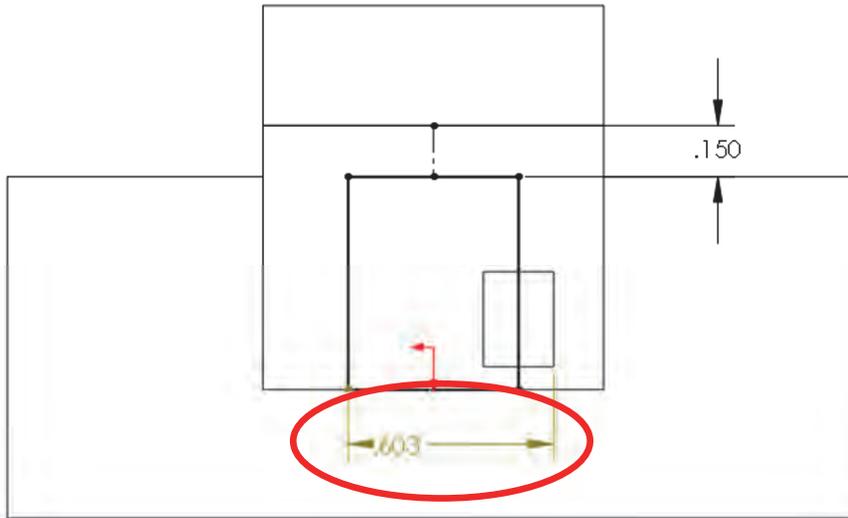
78. – Click in the “*Boss-Extrude1*” with the right-mouse-button and select the “**Parent/Child**” command from the pop-up menu.



79. – We can see that the first dependent feature in the Children list is “*Sketch3*” (from the “*Cut-Extrude1*” feature). Close the “**Parent/Child Relationships**” window, select “*Sketch3*” in the FeatureManager, and edit it. At the same time, the error in this sketch says that a sketch element(s) or dimension(s) are referencing geometry that no longer is there, and so we need to fix it.



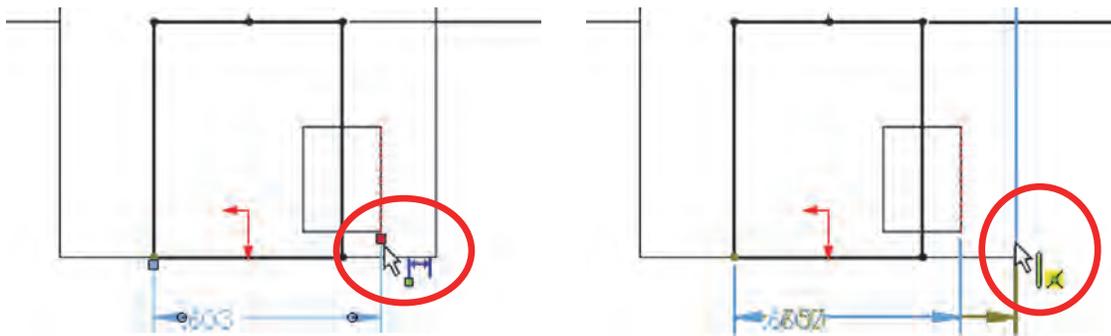
80. – Change to a Top view and “Hidden Lines Removed” mode for clarity. We can see a dimension colored in brown (0.603). The brown colored dimension (default color settings) tells us it is **‘dangling,’** which means that it is referencing geometry that no longer exists. What we need to do is re-define what the dimension is referencing to fix this error.

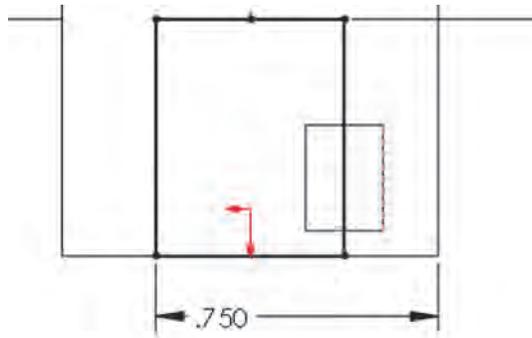


We can do this using one of two techniques:

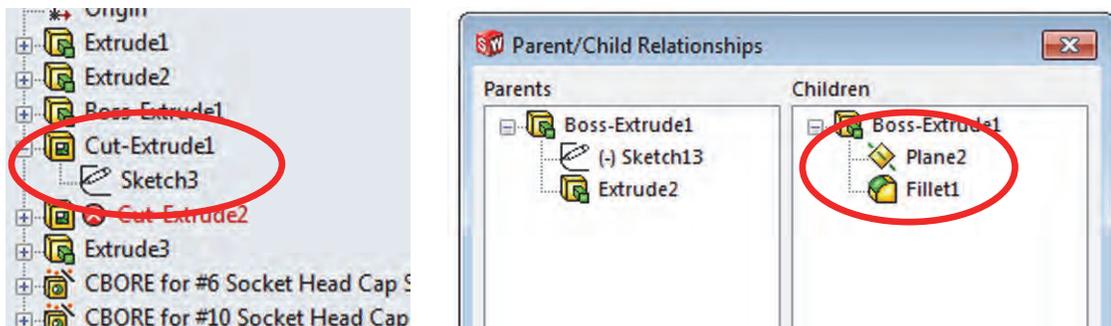
- a) Delete the dimension and add it again to a valid reference, or
- b) Re-attach the dimension to a valid reference.

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's referencing geometry that is no longer in the model. To re-attach it, **drag the red dot** onto a valid reference; it can be any edge or vertex. For this dimension we'll use the right side edge. After re-attaching the dimension note that the geometry does not change, the value of the dimension is what updates.

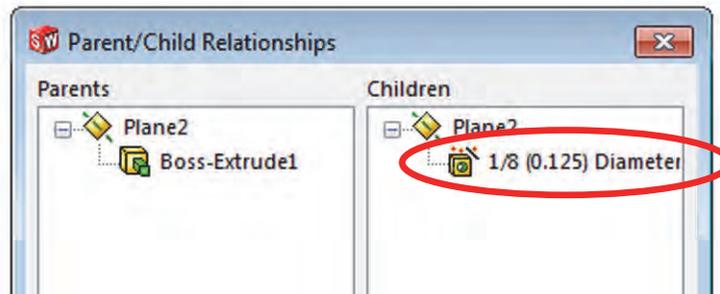




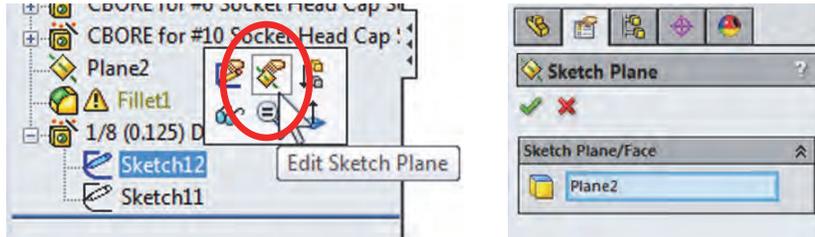
81. – After we exit the sketch dismiss the “**What’s Wrong**” dialog. Notice that the error in “*Sketch3*” is fixed, and after reviewing the “*Boss-Extrude1*” “**Parent/Child**” relationships we can see, “*Sketch3*” and “*Cut-Extrude1*” are no longer listed as children features of “*Boss-Extrude1*” since we changed the dimension to reference a different edge, breaking the relation and fixing the error at the same time.



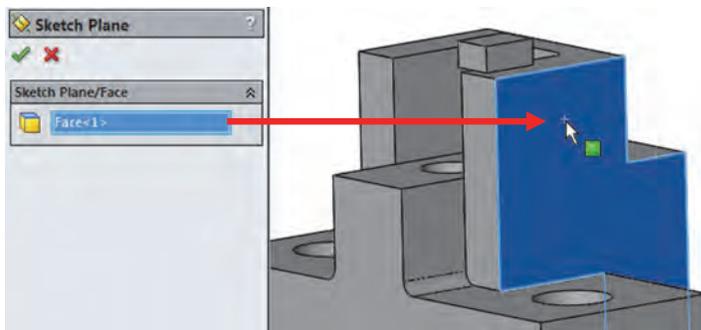
82. – From the relationships dialog we can see that the only children left are “*Plane2*” and “*Fillet1*.” Close the dialog and review the “**Parent/Child**” relations of the “*Plane2*” feature. Here we can see the only child feature is the “*1/8 (0.125) Diameter Hole1*.” A Hole Wizard feature has two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



The most likely reason why the “1/8 (0.125) Diameter Hole1” feature is a child to “Plane2” is that it was most likely made in “Plane2.” We can find out in which plane or face a sketch is made by editing its 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 the sketch is located in it.



83. – To change the sketch to a different plane (or face) and delete the relationship to “Plane2” (and by extension to “Boss-Extrude1”), select a new face to locate the sketch. Select the face indicated and click **OK** to finish. Close the “**What’s Wrong?**” dialog to continue.



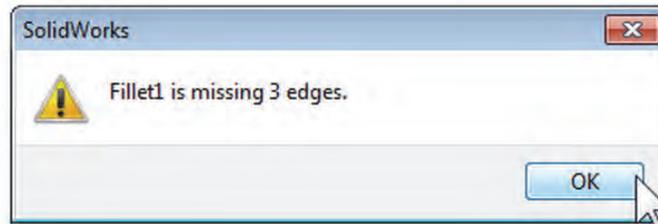
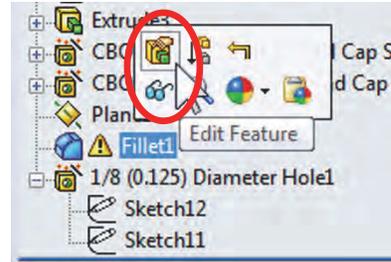
84. – Review the “**Parent/Child**” relationships again for “Boss-Extrude1” and “Plane2.” Now the only children of “Boss-Extrude1” are “Plane2” and “Fillet1”, and “Plane2” has no children. Close the dialog to continue.



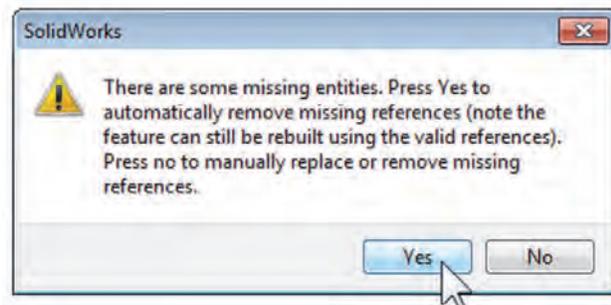
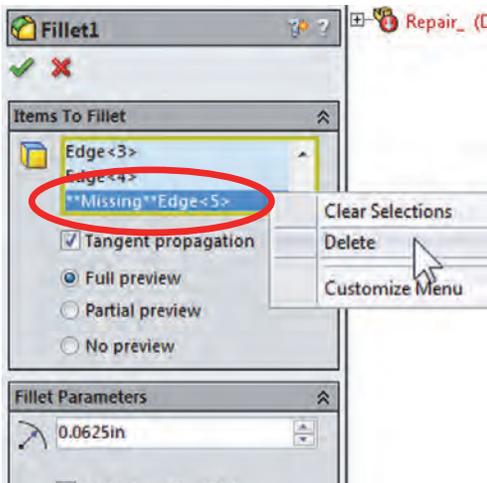
85. – Select “Fillet1” and check “What’s Wrong?” Note that the icon is different; in this case the error is a warning. The fillet was built but it’s missing one or more edges.



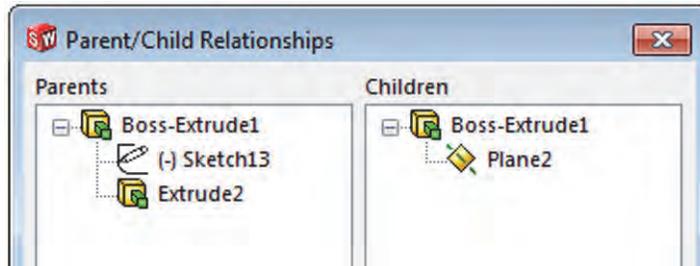
86. – Close the dialog and edit the “Fillet1” feature. When we edit the fillet we get a message letting us know that the fillet is missing three edges, this means that three edges that had been rounded, are no longer in the model, and the fillet command did not find them. Click OK to continue editing the fillet.



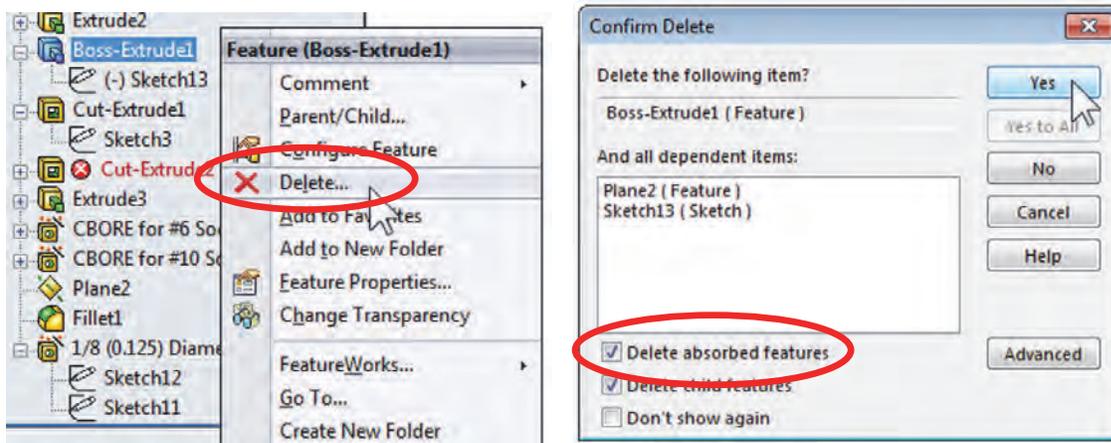
When we edit the fillet, the “Items to fillet” box is shown with a brown outline indicating that the fillet is missing one or more edges. In this case we can scroll down the list and delete the missing edges manually, or click OK. If we don’t delete the missing edge references, we’ll be asked if we want to remove the missing items and continue. Delete the edges from the list, or click OK to continue and select “Yes” to remove the missing references.



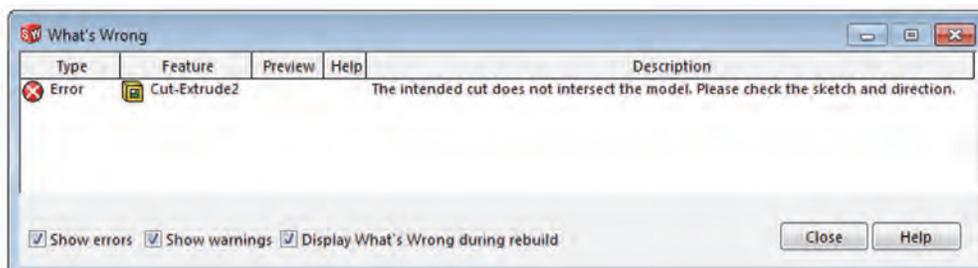
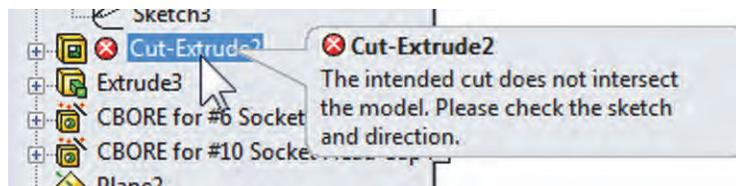
87. – After fixing “Fillet1” check the “Parent/Child” relations of “Boss-Extrude1.” Since “Fillet1” is no longer a child feature of the “Boss-Extrude1” feature we can delete it.



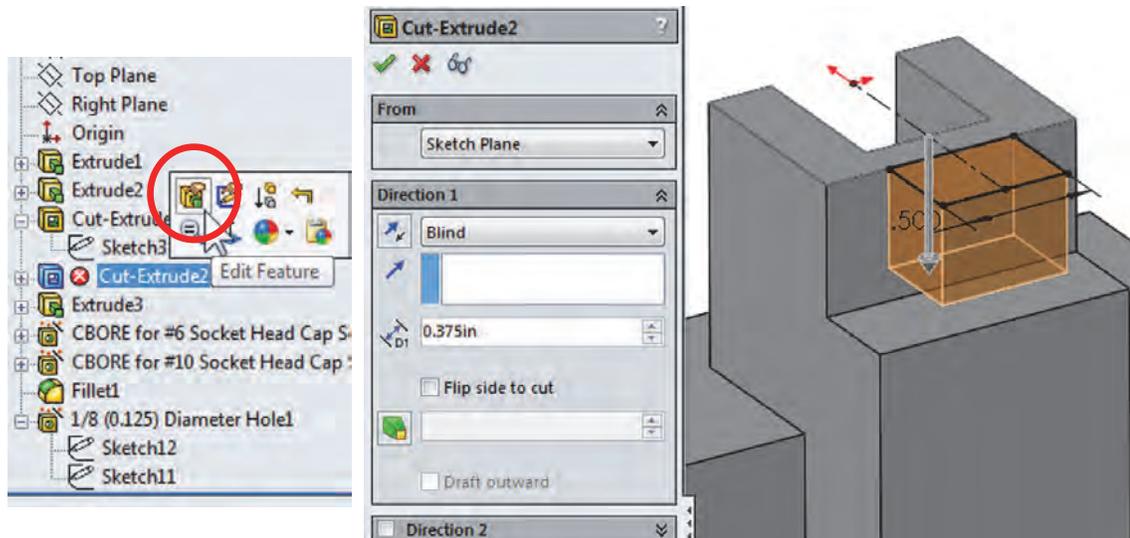
88. – Select “Boss-Extrude1” in the FeatureManager and delete it. This time the confirmation dialog only lists “Plane2”, which we don’t need (or want now) and it’s OK to delete it. Be sure to activate the option “Also delete absorbed features” to delete its sketch, too.



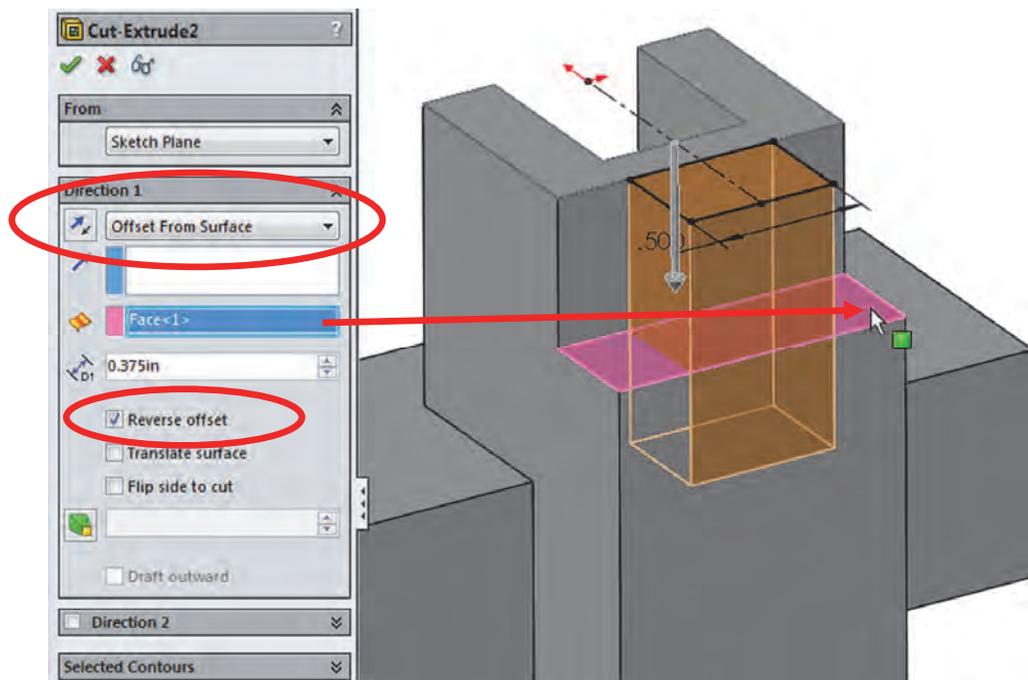
89. – The next error to fix is “Cut-Extrude2.” The error we see using the “What’s Wrong?” command means that the feature is not cutting the model.



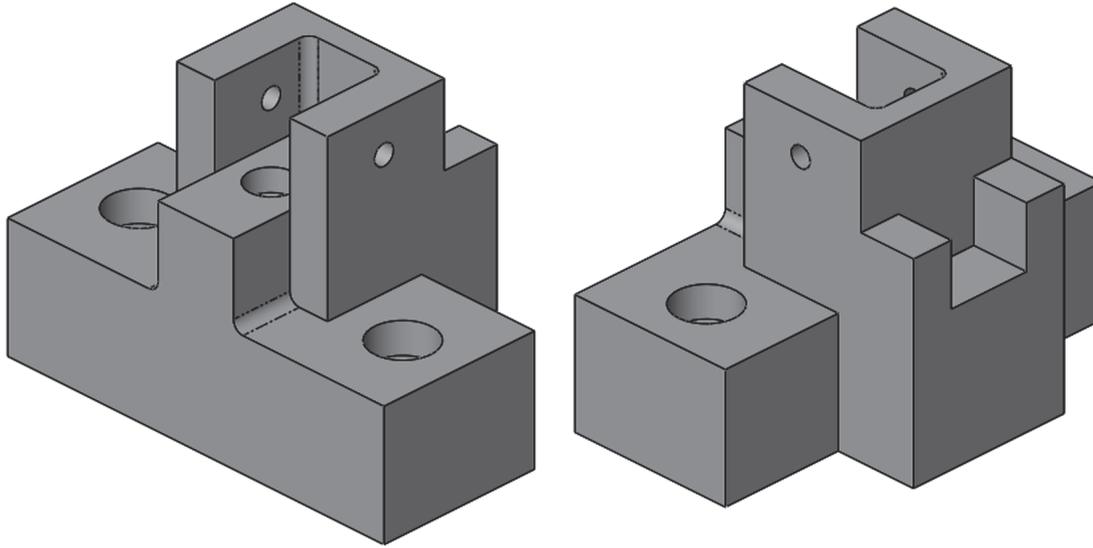
90. – Close the “**What’s Wrong**” dialog and Edit the “*Cut-Extrude2*” feature. In the preview we can see that the cut is not going deep enough and we need it to make a 0.375” cut into the lower step.



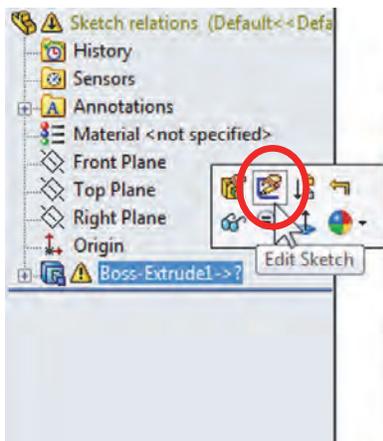
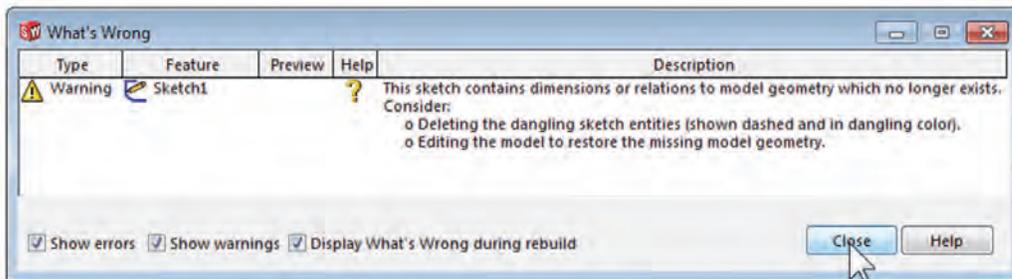
91. – Change the cut’s end condition to “**Offset from Surface**” and select the lower step’s face as indicated. This end condition will make the cut the offset distance going to either side of the selected face; if needed, turn on the “Reverse offset” checkbox to cut into the part as shown. Click OK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



The finished part fully repaired looks like this. Save and close the part.



92. – When working in a sketch we can have different errors, so we decided to show them with a different part. Open the part 'Sketch Relations.sldprt' from the included files. Like the part before, we have a number of errors in this part's sketch. After opening the file close the "What's Wrong" dialog and edit the "Boss-Extrude1" sketch. We'll show the use of diagnostic tools to help us correct these errors.



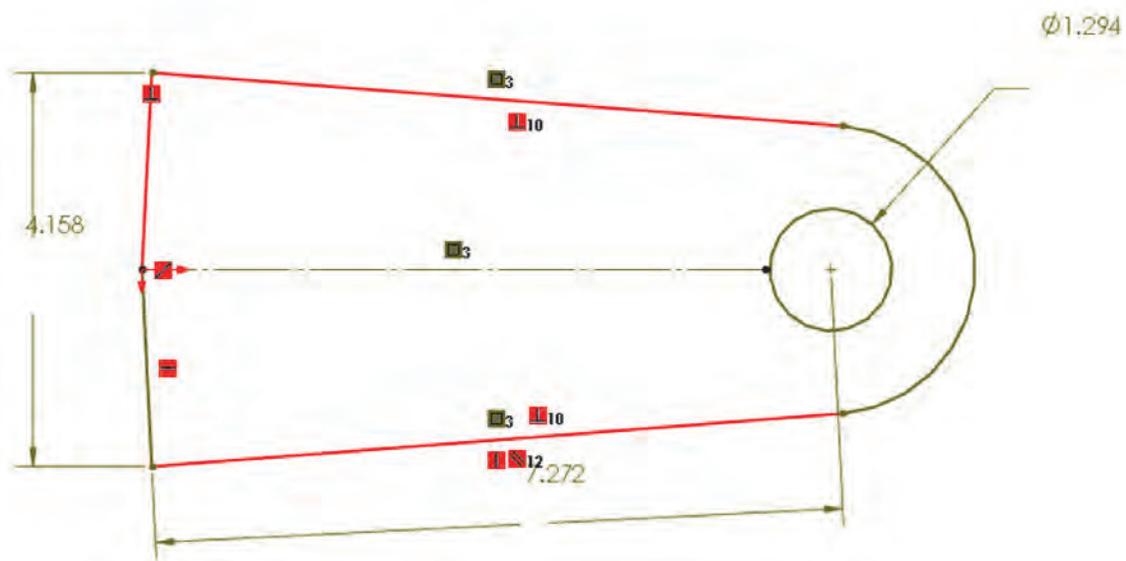
In a sketch we can have many types of geometric relations. What we are going to focus on is when relations are not solved correctly and generate warnings and errors.

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 any of the following states:

State	Sketch Color	Description
Under defined	Blue	Not enough relations to fully define sketch
Fully Defined	Black	Enough information to fully define sketch
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 that reference other component's geometry added in an assembly.
Locked *	Can be any color	'Frozen' <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 referenced by the sketch is deleted or modified, or when the user adds conflicting relations and/or dimensions to the sketch elements.



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 also a good tool to identify and/or delete relations, especially “External”, “In context”, “Locked” or “Broken”, if needed.

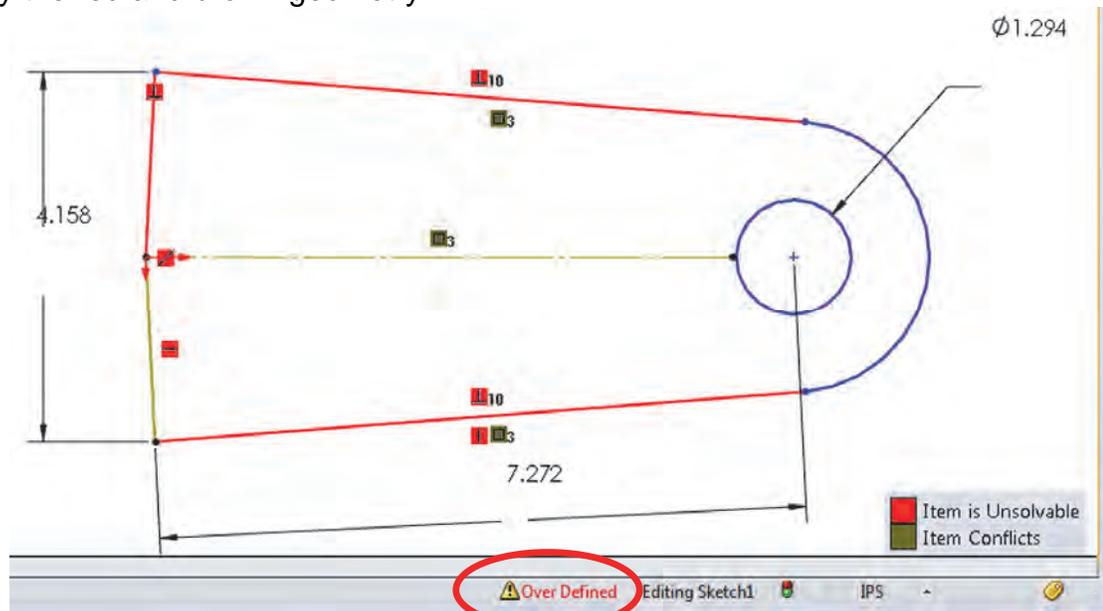
95. – Using this tool we’ll remove the “**In Context**” relations, as we don’t want our sketch to have relations to other part’s geometry. An “**In Context**” relation is added when we modify a sketch inside of an assembly and add references to other components. By deleting this relation, our part will no longer be referencing geometry outside the part (more on this in the Top Down Design section).

In the drop down list select “**Defined In Context.**” In our sketch we only have one 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.

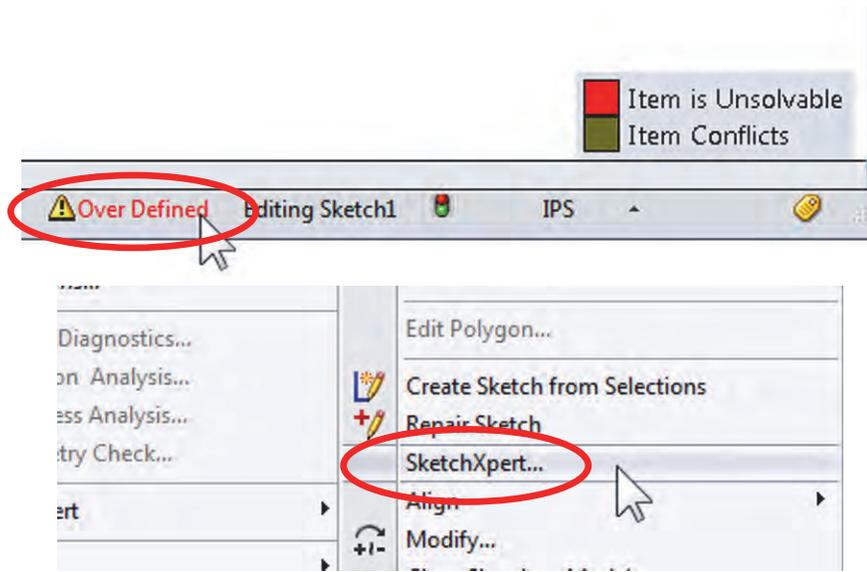
*More information will be provided later about relations in the context of an assembly in the “**Top Down Assembly**” lesson.



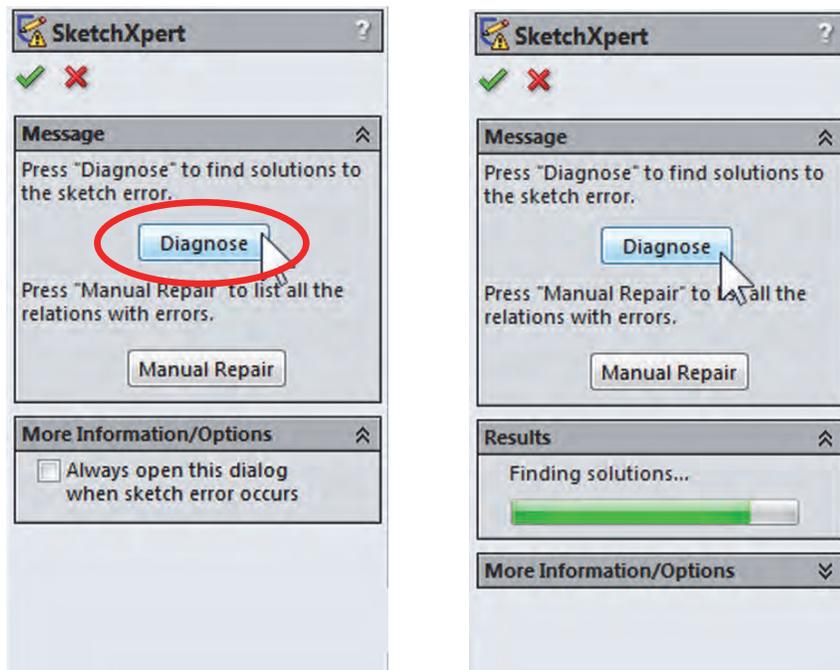
96. – After deleting this relation, our sketch is still in an undesired state, we have both unsolvable and over defined geometry, as we can see in the status bar and by the red and brown geometry.



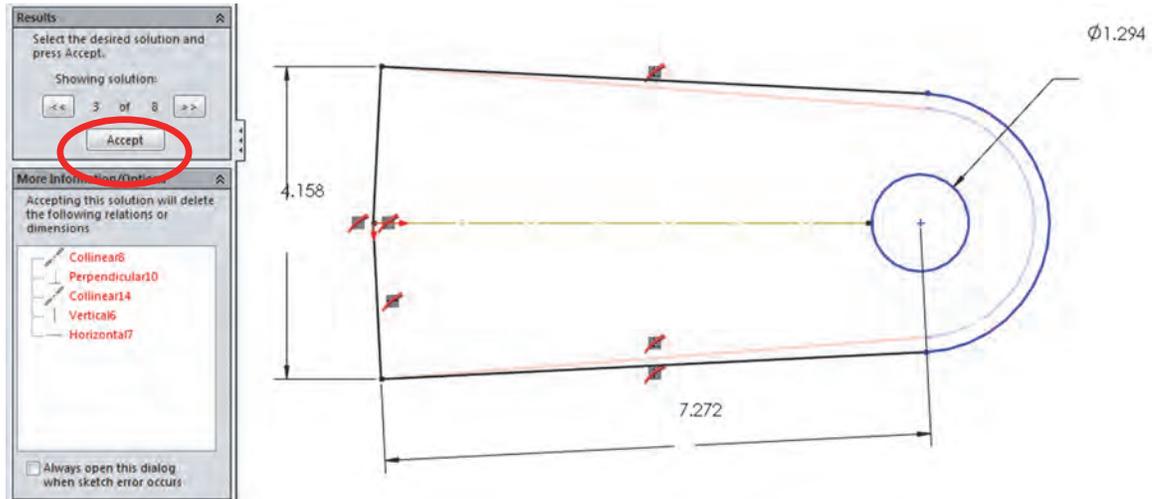
97. – The conflicting and over defined relations can be fixed using this tool, but it would take a long time; this tool is a good option when we have only a few relations that need to be fixed, or to find the relation that is causing problems. In this case, the sketch has so many errors that we'll fix them using **SketchXpert**, which is a great tool to quickly review multiple possible solutions. To activate it click in the “**Over Defined**” message in the status bar, or use the menu “**Tools, Sketch Tools, SketchXpert.**”



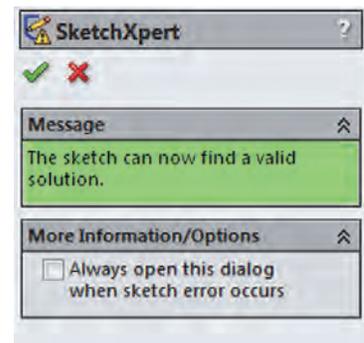
98. – In the **SketchXpert** dialog select “Diagnose” to automatically analyze the sketch and evaluate possible solutions.



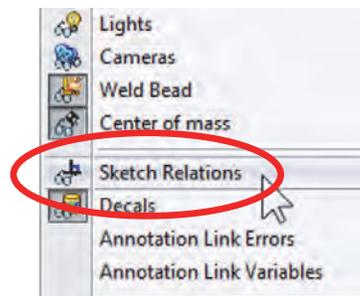
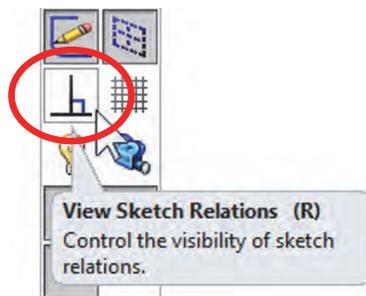
99. – SketchXpert quickly diagnoses the sketch and offers possible valid solutions. We can view each option by advancing in the “Results” box. In the “More Information/Options” box we can see the relations and/or dimensions that would be deleted if we accept the solution displayed. Scroll until you see the following solution and click “Accept” when done. This is not the optimum result, but it’s the closest one that doesn’t have an error.



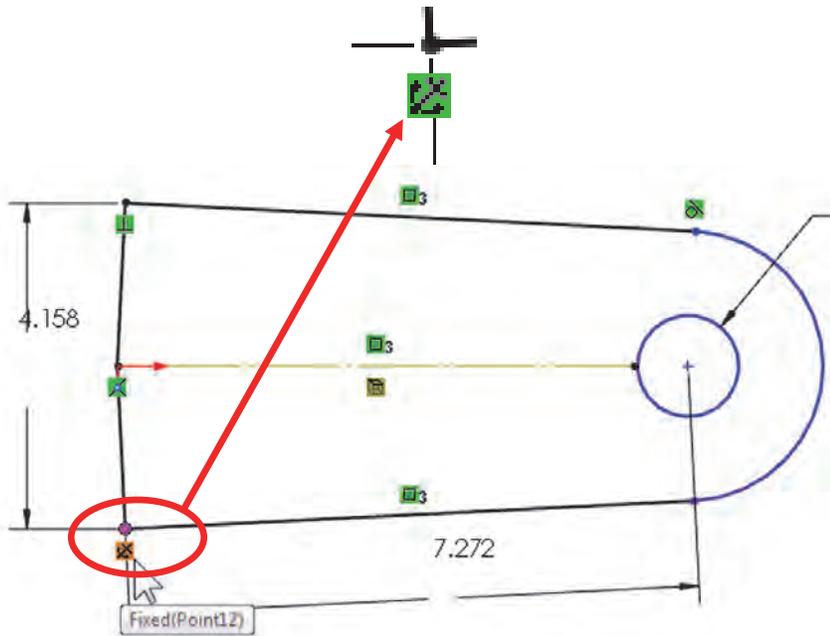
After accepting the solution we see the message “The sketch can now find a valid solution” with a green background to let us know that our sketch is error free. This does not mean that the sketch is what we want; it only says that there are no errors in it. Click OK to finish and continue.



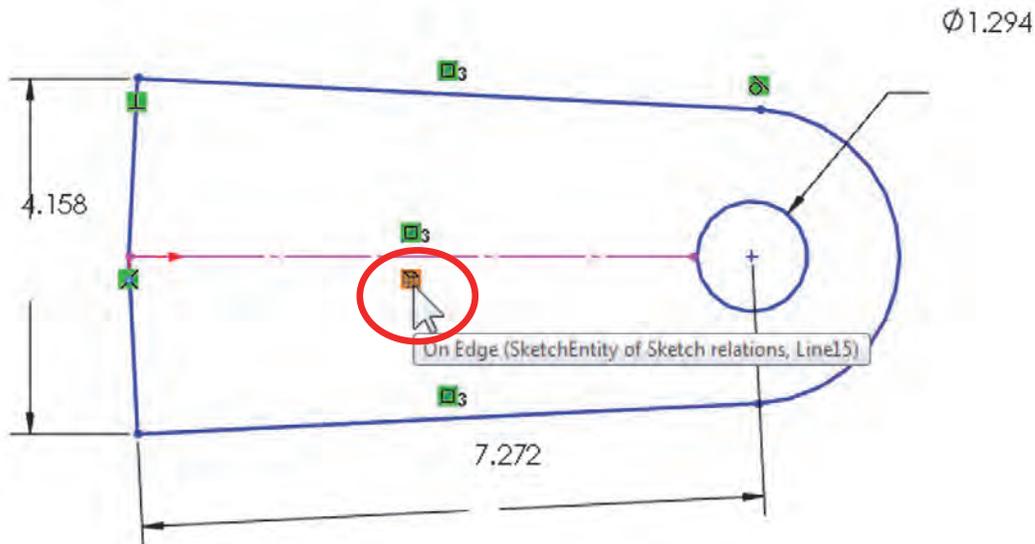
100. – We want to make the lines on the left to be collinear and horizontal (Remember the short red arrow at the origin indicates the horizontal direction in the sketch; here it is shown sideways to save space). What we need to do is to find what relations are keeping the lines fully defined, delete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

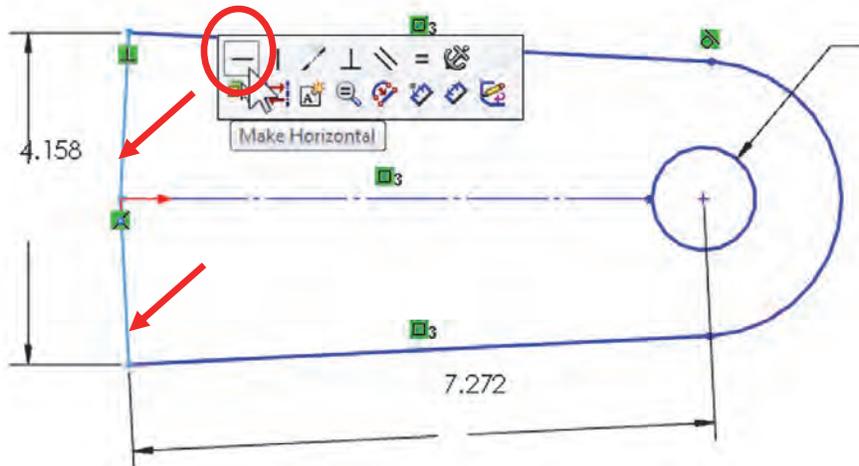


101. – We can see that there are only three relations in the two lines that we are interested in: a “Perpendicular,” a “Coincident” and a “Fixed” relation. The “Fixed” relation is the equivalent of arbitrarily constraining an element in space. Think of it as putting a nail in a geometric element and hammering it in. Read: brute force. Select the “Fixed” geometric relation in the screen and delete it.

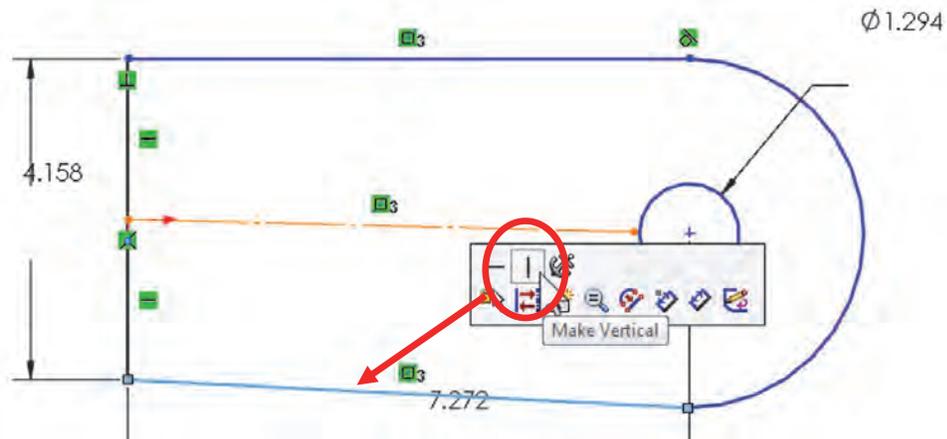


102. – If we pay attention, we’ll see that we have a relation colored brown/gold in the centerline. This relation is “On Edge”, this is the type of relation created when we use the “Convert Entities” command in a sketch. The brown/gold color means it is “dangling”; in other words, it lost the reference it was converted from, and we have to delete it. Delete the dangling relation and make both lines on the left “Horizontal”. (Remember the sketch is rotated 90° in the picture.)

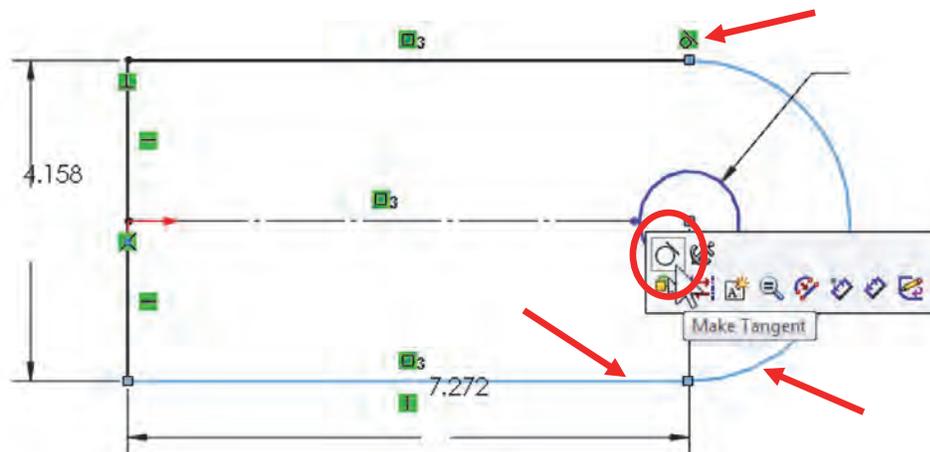




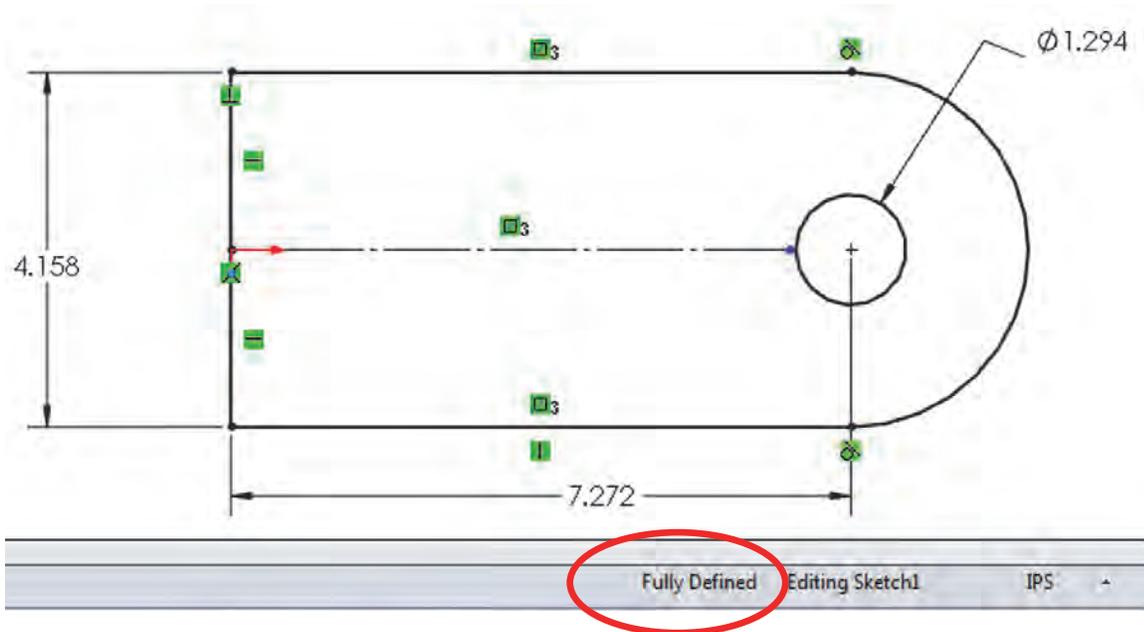
103. – Make one of the long lines **“Vertical;”** the other two lines will also become vertical as they have a **“Symmetric”** relation between them about the centerline.



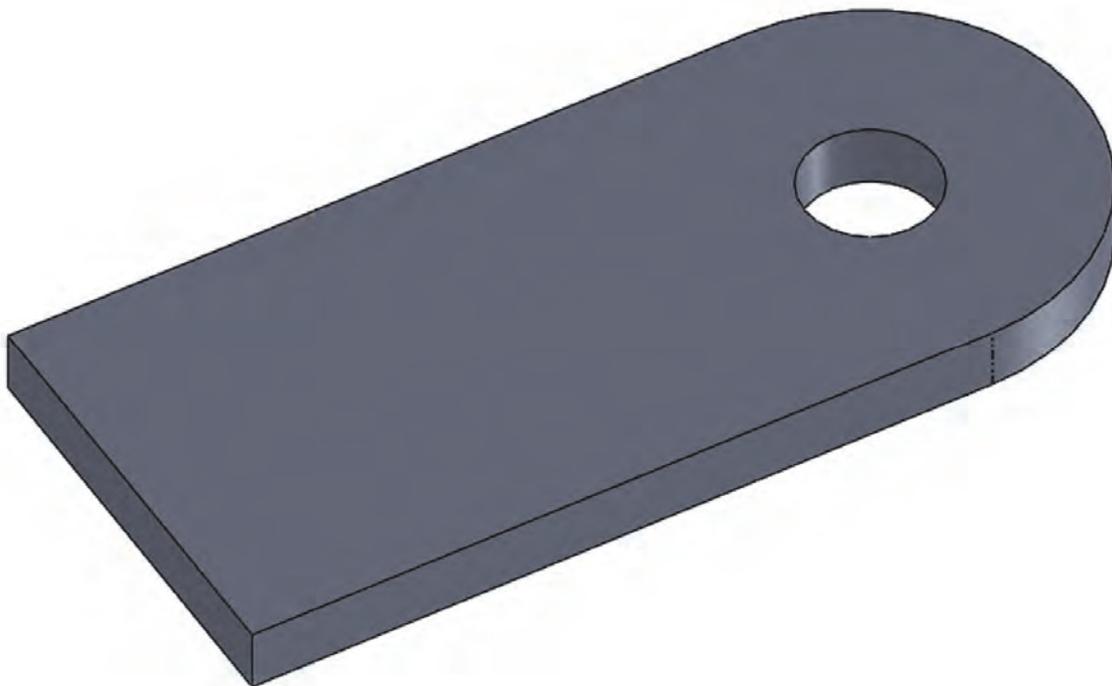
104. – To complete the changes and fully define the sketch we need to add a **“Tangent”** relation between the lower long line and the arc Press and hold the **“Ctrl”** key and select both entities; after releasing the **“Ctrl”** key select **“Tangent”** from the pop-up toolbar. Note that the other side of the arc is already tangent to the top long line. To see the effect of having an under defined relation, click and drag the blue endpoint to see the effect of not having the tangent relation.



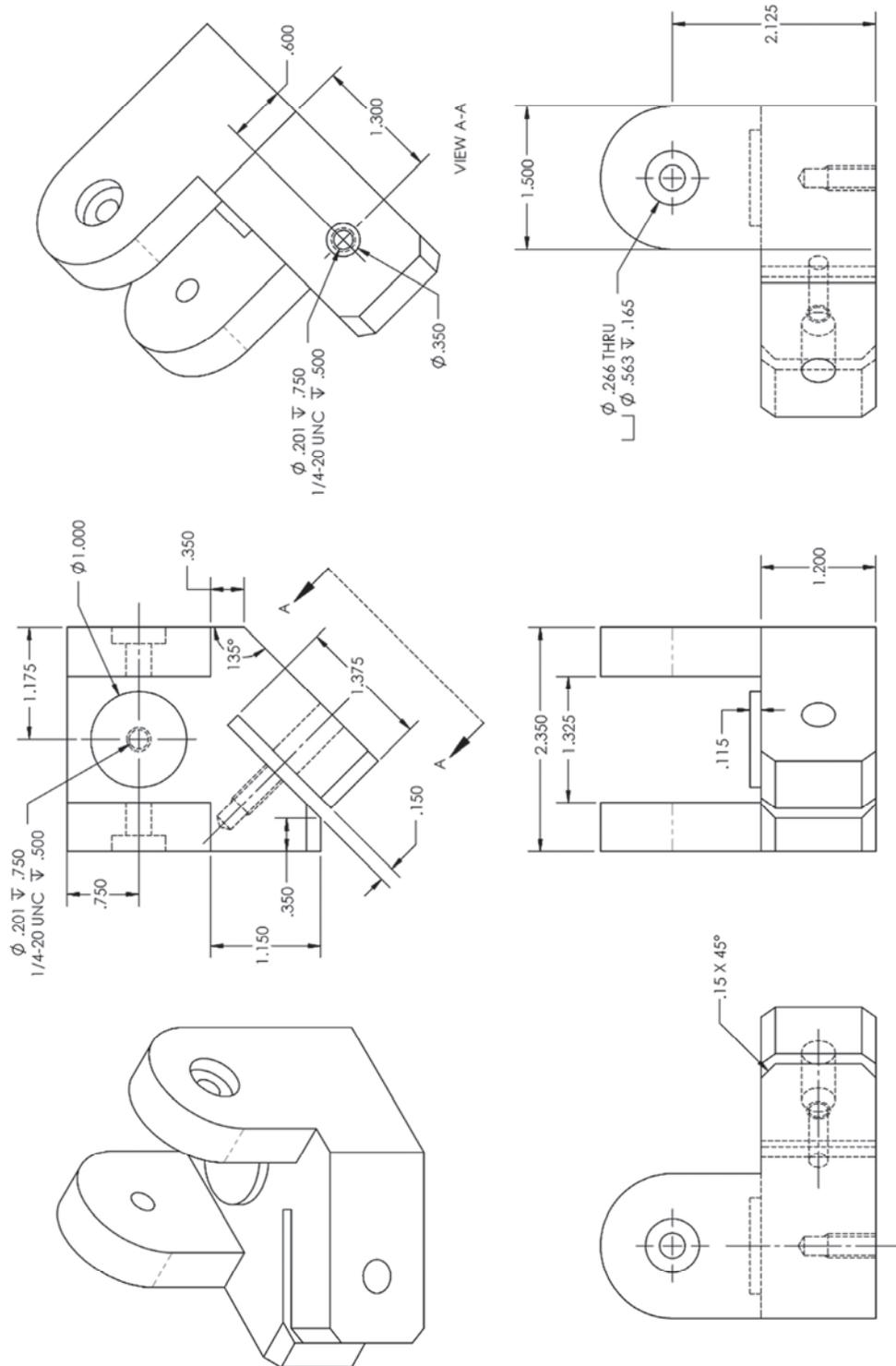
After adding these relations we can see the status bar indicates that our sketch is “Fully Defined.”



105. – Now that we are finished repairing the sketch, exit the sketch to finish. Save and close the file.



Exercise: Open the file 'Model Repair Exercise.sldprt'. After opening it and rebuilding it, a list of errors will be displayed; however, the part has so many problems that no geometry will be available. Correct the errors in features and sketches. Use the following image of the finished part for reference. A high resolution image of this exercise is included for your convenience.



Notes:

Equations

One way to help us maintain design intent in our models is by using equations. Equations are commonly used to evenly space features, add or remove instances to a pattern's count, change dimensions when other features are modified, etc. All equations in SolidWorks have the following format:

Variable = Expression
Where:

Variable is the dimension/value to change (dependent value.)

Expression is the algebraic combination of other dimensions and values that will define the value of **variable**.

For example, if we have a part of length “L” where we want to evenly space a pattern of “N” number of holes spaced a dimension “S,” the **variable** dimension will be “S,” because that’s the value we want to change when the number “N” and/or length “L” values change. Our equation would look like:

$$S = L / N$$

A good practice when working with equations in SolidWorks is to rename dimensions and features, so instead of having an equation that reads:

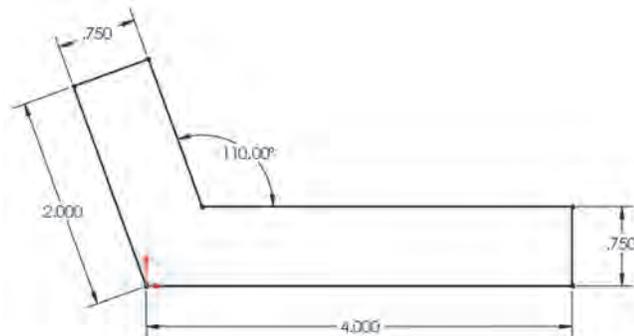
$$\text{“D1@LinearPattern1”} = \text{“D3@Sketch1”} / \text{“D2@LinearPattern1”}$$

It would look like this:

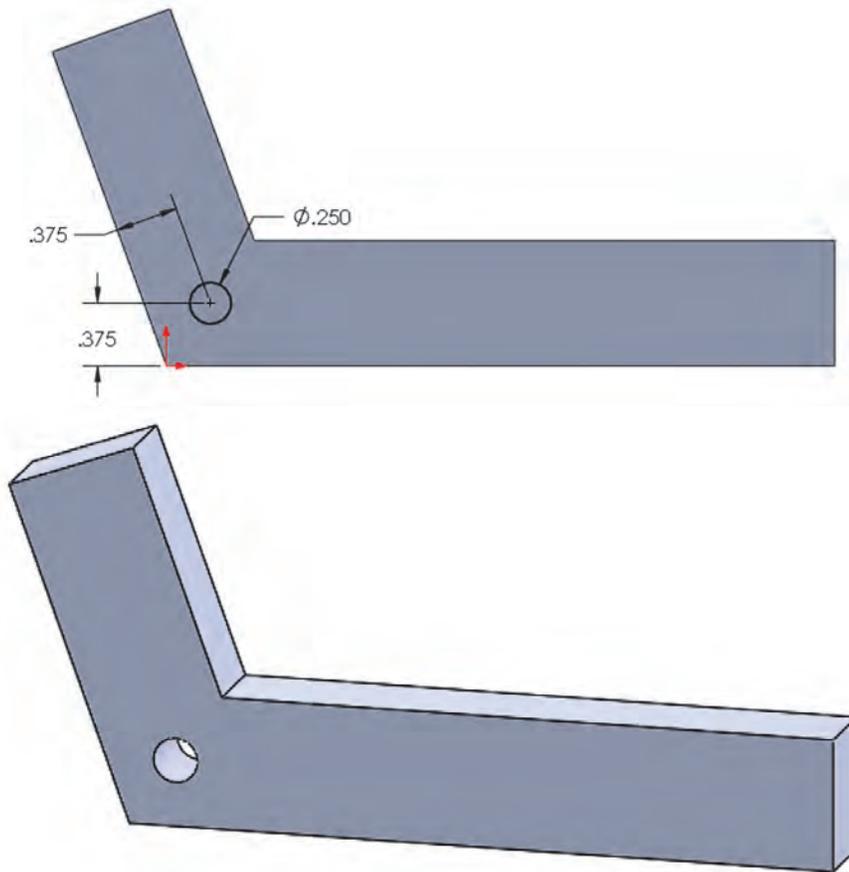
$$\text{“Spacing@Holes”} = \text{“Length@Base”} / \text{“Number@Holes”}$$

This is more descriptive and easier to understand. If we have one, maybe two, equations in a part it may not be a problem to keep track of them, but with more equations it becomes more difficult to manage and modify them if needed.

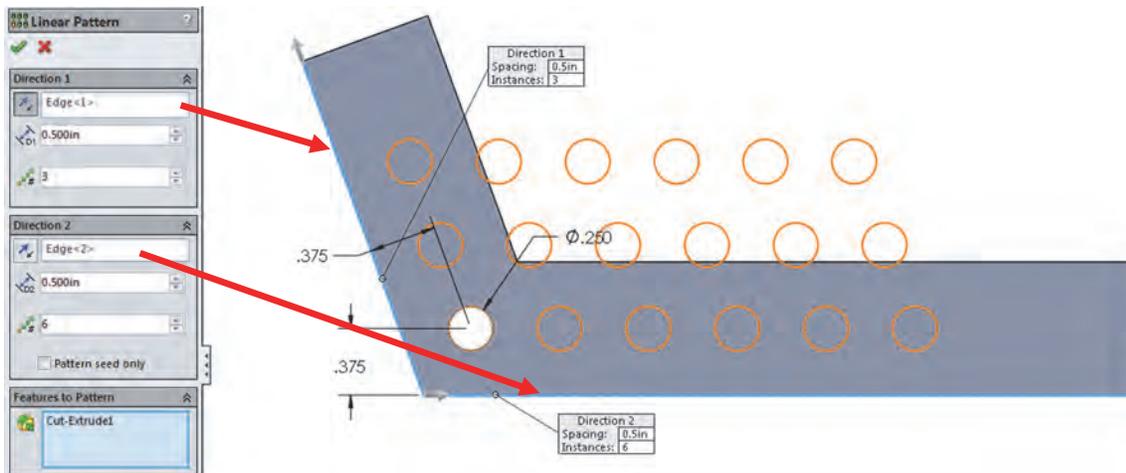
106. – To show how equations work, we’ll make a simple part, rename its features and dimensions, and finally add an equation. Draw the following sketch (any plane will do) and extrude it 0.5”. Dimensions are in inches.



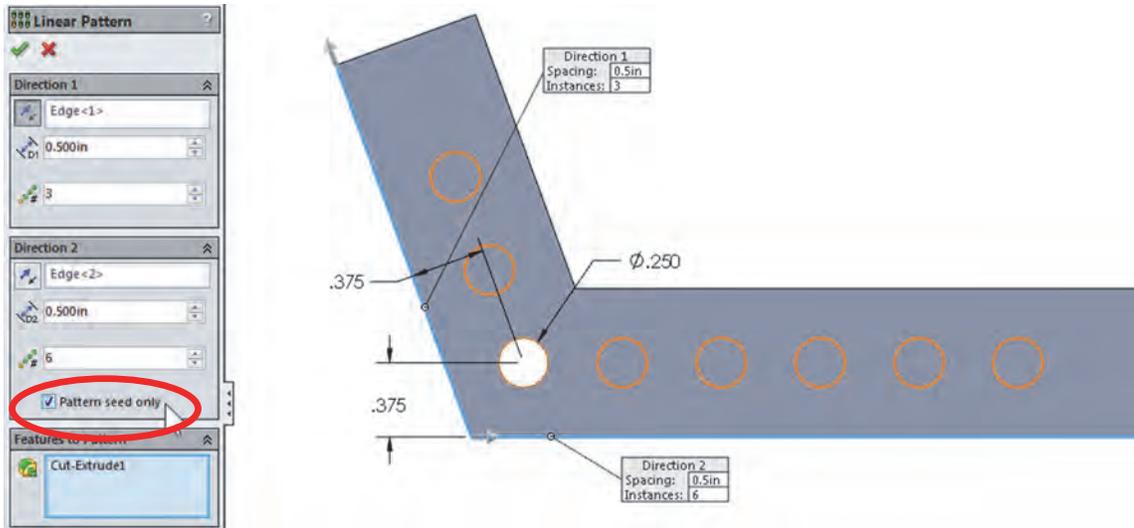
107. – Add a through hole in the corner...



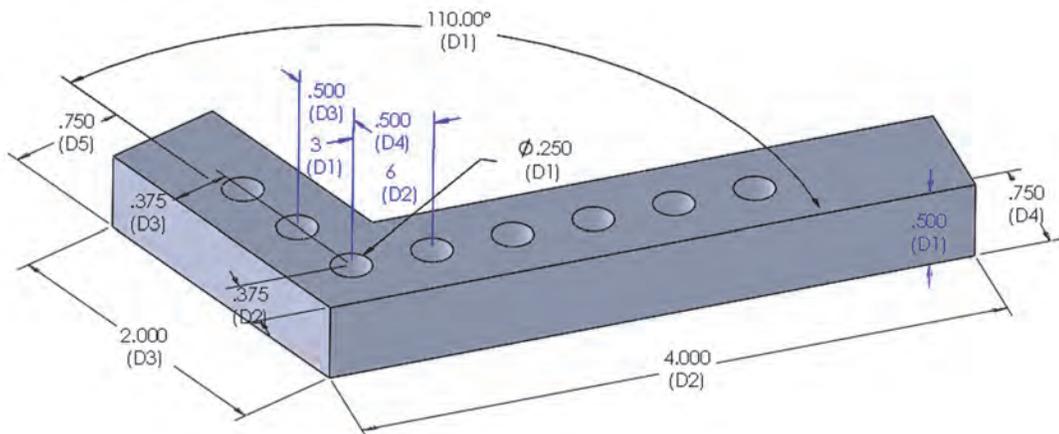
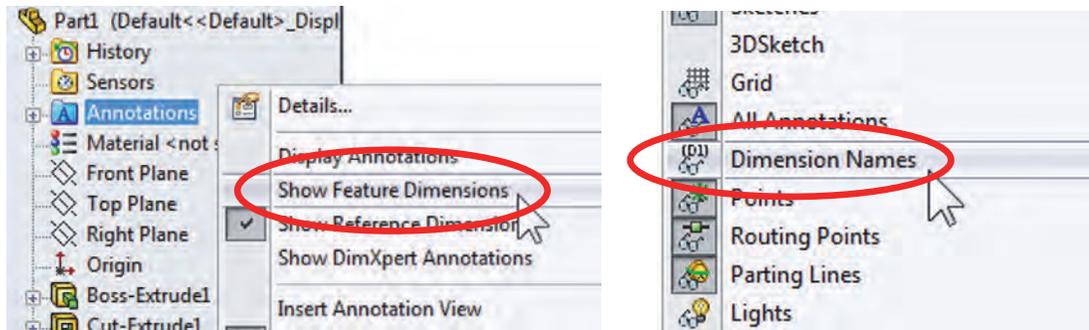
108. – Add a linear pattern. In this case the pattern will be made in two directions. Select a diagonal edge for Direction 1, three copies spaced 0.5", and a horizontal edge for Direction 2, six copies spaced 0.5" (don't finish it yet).



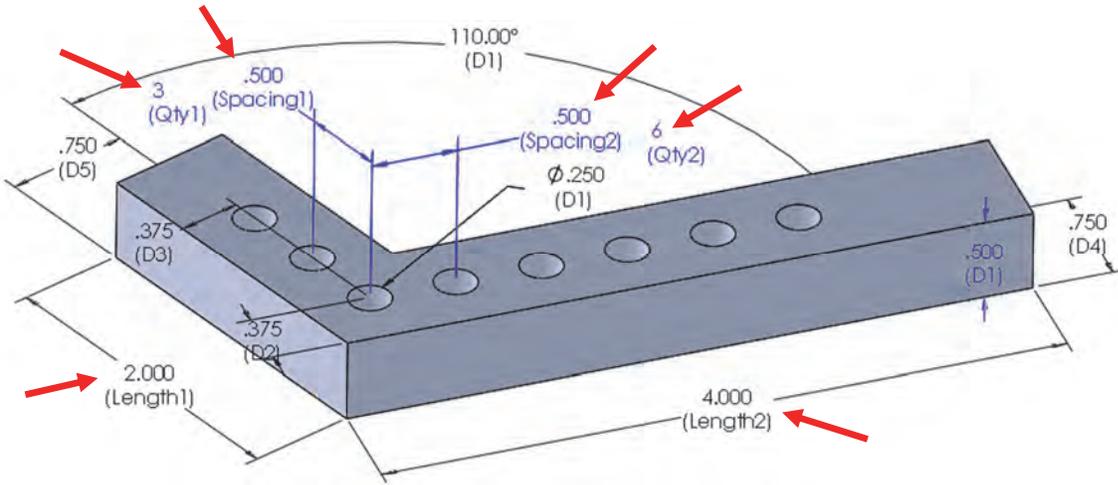
109. – Since we need a single row of copies in each direction, check the option **“Pattern seed only”** under the “Direction 2” options box. This option allows us to only copy the original hole, not the copies of the copy. Click OK to finish.



110. – Now we need to rename the dimensions. To show feature dimensions, right-mouse-click in the **“Annotations”** folder and select **“Show feature dimensions”**; to view dimension names, activate the menu **“View, Dimension Names.”**



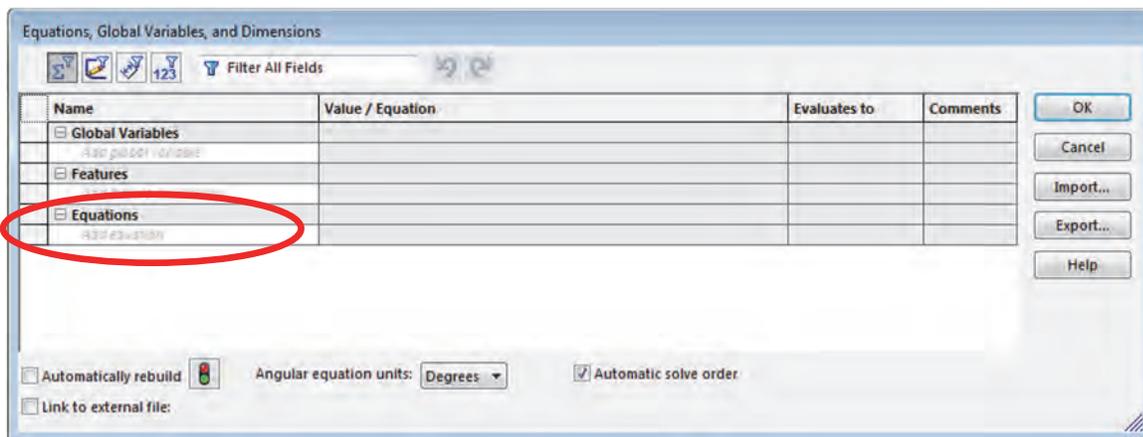
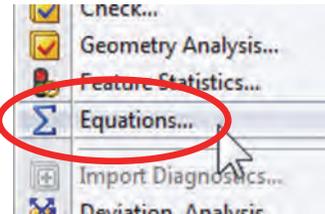
111. – Rename the dimensions for the side's length and linear pattern as shown; this way when we add the equations, it will be easier to identify the dimensions. Select each dimension and type the name in its PropertyManager.



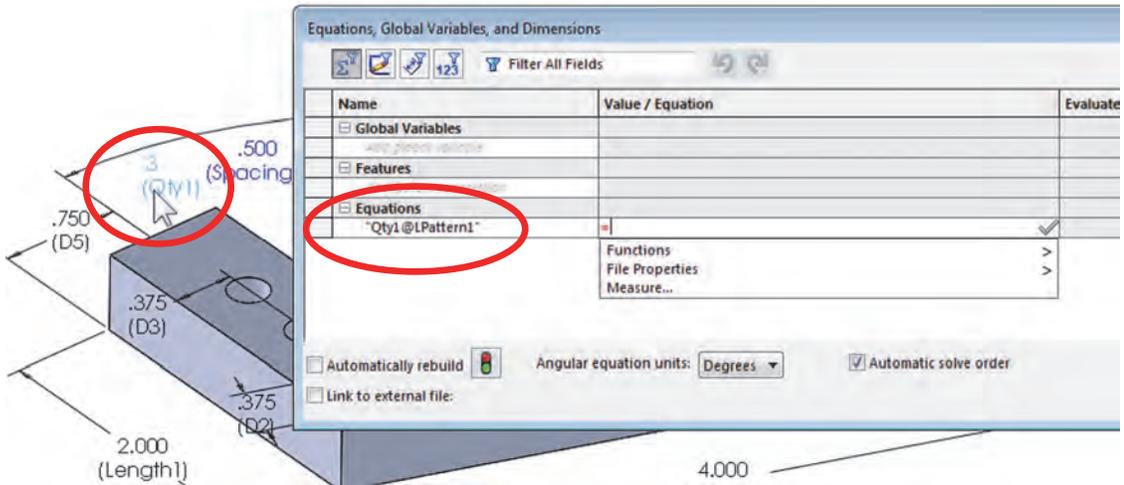
112. – Our design intent is to have a pattern of holes equally spaced to fill each side and update accordingly if the length or spacing changes. To accomplish this, we'll need two equations to calculate the quantity of holes (one for each direction). Our “dependent” dimensions will be “Qty1” and “Qty2,” and the driving dimensions will be “Length1,” “Length2,” “Spacing1” and “Spacing2.” Based on this, our equation's general format has to be:

$$\text{Qty1} = \text{Length1} / \text{Spacing1} \quad \text{Qty2} = \text{Length2} / \text{Spacing2}$$

113. – There are two ways to add equations; we'll learn both, one method with each equation. For the first one, select the menu “**Tools, Equations.**” We are immediately presented with the Equations dialog box. Here we can add equations, define Global Variables to use as constants, and set feature's suppression states.

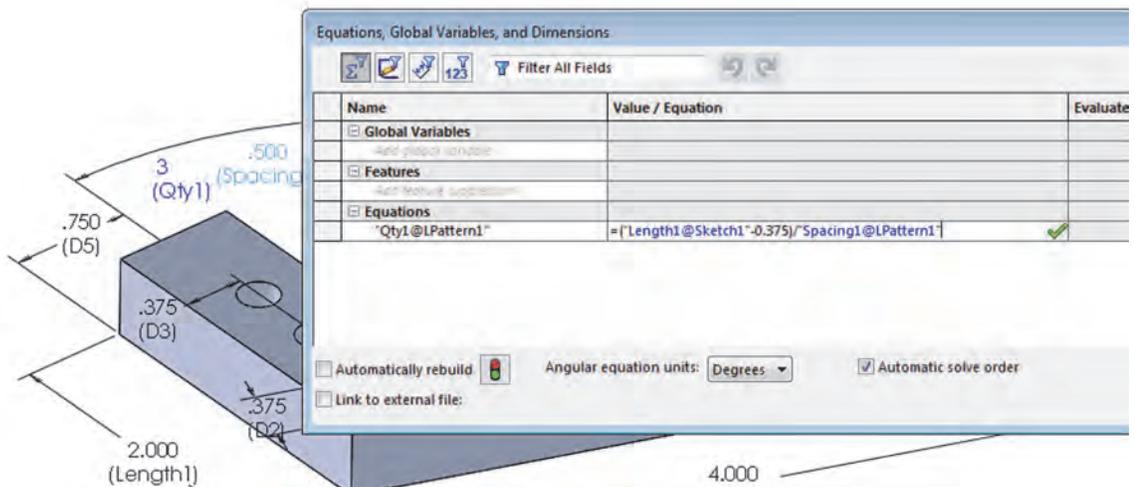


114. – To create the first equation click in the “Add equation” field in the Equations list; this is where the dependent variable will be listed. We can either type a dimension’s name, or select it in the screen. Select the dimension “Qty1” in the screen to add its full name “Qty1 @LPattern1” in the “Add Equation” field.

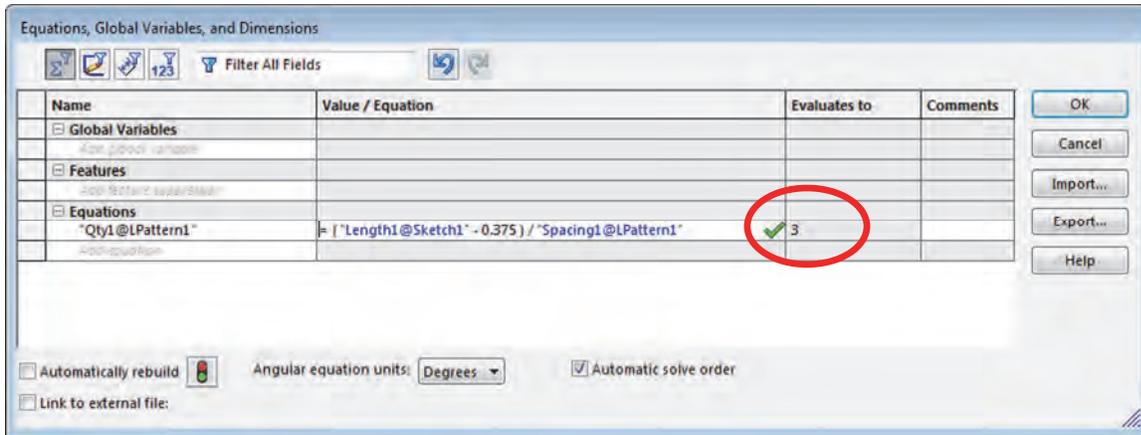


115. – After selecting the “Qty1” dimension, the equal sign is automatically shown in the Value/Equation field, waiting for us to type the rest of the equation to evaluate “Qty1.” In order for our equation to calculate the correct number or holes, we have to subtract a small distance from the length in order to have a minimum space at the end. Otherwise we may get an extra hole at the end. After the equal sign, open a parenthesis, select the “Length1” dimension (its full name will be copied), type ‘- 0.375’, close the parenthesis, type ‘/’ to divide and finally select the “Spacing1” dimension. The green checkmark next to the equation indicates that it is a valid expression. Our first completed equation looks like:

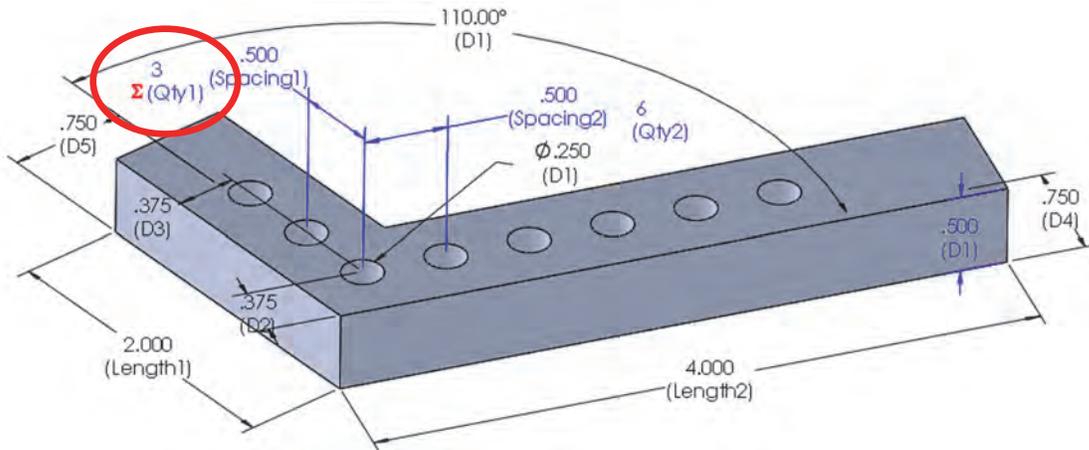
$$\text{“Qty1@LPattern1”} = (\text{“Length1@Sketch1”} - 0.375) / \text{“Spacing1@LPattern1”}$$



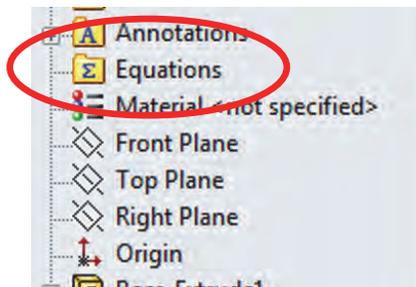
116. – In the “Equations” dialog we see the first equation is added, and after selecting the “Evaluates to” column the value is calculated. The value for the “Qty1” dimension evaluates to 3. Click OK to finish and continue.



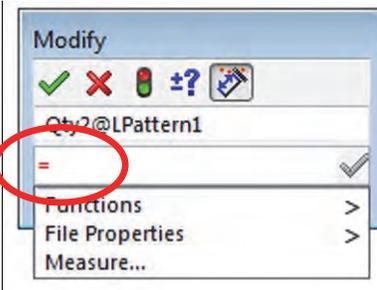
117. – After adding the equation, we see a red Σ symbol next to the dimension, this indicates that its value is driven by an equation, and cannot be changed directly as other dimensions. It can only be changed when the dimensions driving it (*Length1* and *Spacing1*) change.



After adding an equation to a model, a folder named “Equations” is automatically added to the FeatureManager.

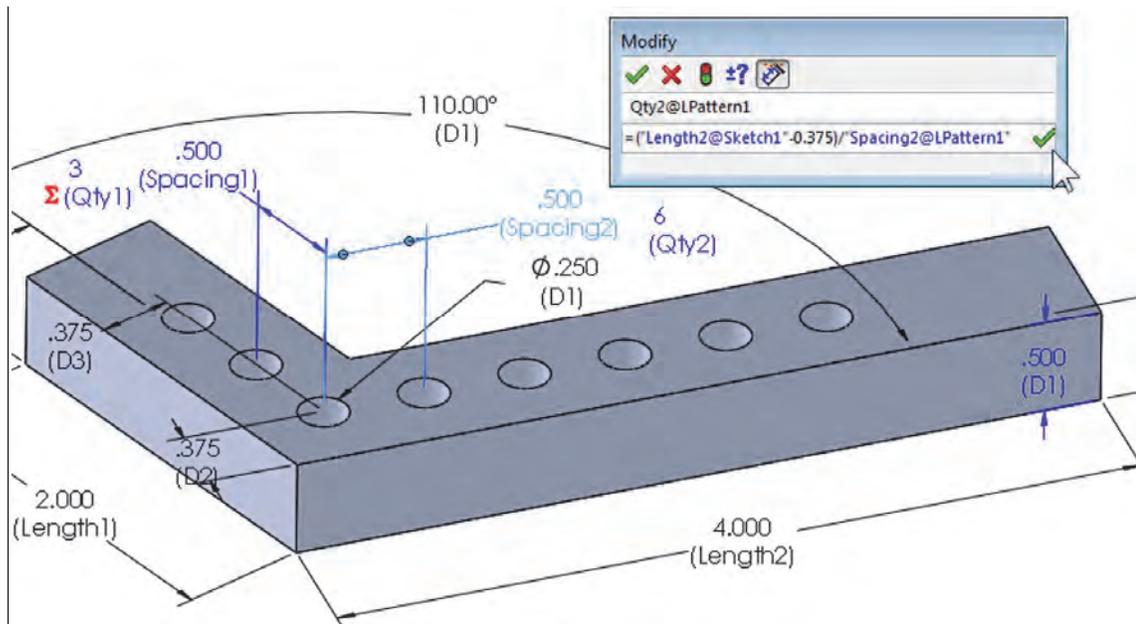


118. – To add the second equation we'll use a different approach. Double click in the "Qty2" dimension as if to change its value. In the second line, where we enter the dimension's value, type '=' (equal sign); now we are ready to enter the equation just as we did previously, but using the Length2 and Spacing2 dimensions. Remember we can select a dimension to copy its name into the equation.

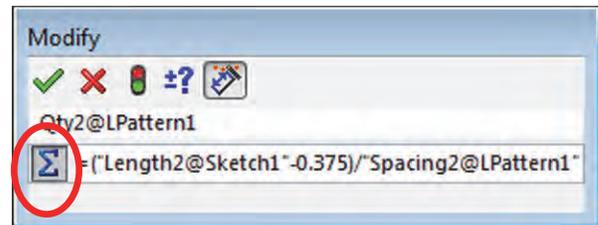


119. – Just as we did with the first equation, we'll open a parenthesis immediately after the equal sign, click in the "Length2" dimension, enter '- 0.375', close the parenthesis, add '/' and click in the "Spacing2" dimension. The completed second equation entered in the Modify dialog box looks like:

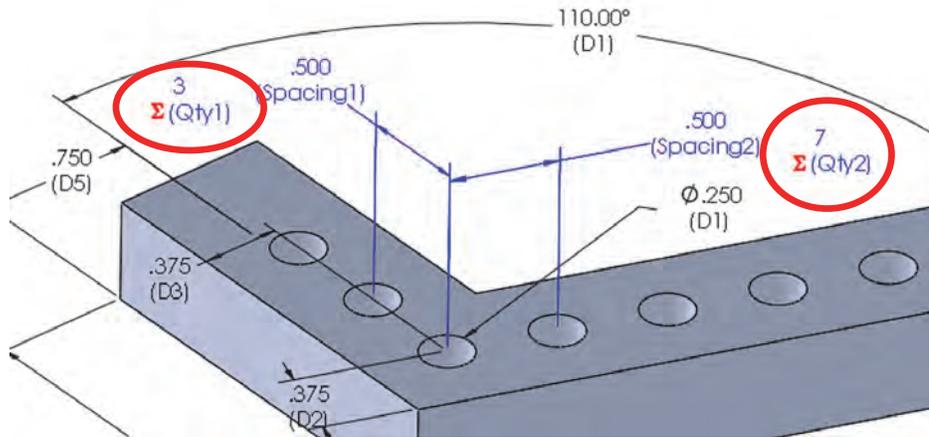
$$= ("Length2 @Sketch1" - 0.375) / "Spacing2 @LPattern1"$$



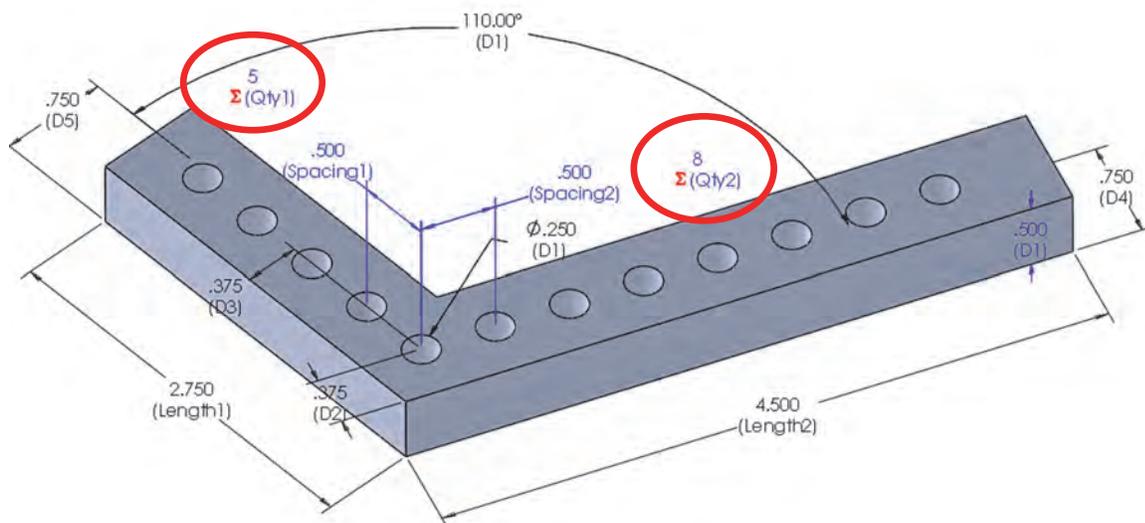
After clicking the green checkmark at the right side, the equation is created and the Σ symbol is added. Click OK to finish adding the equation.



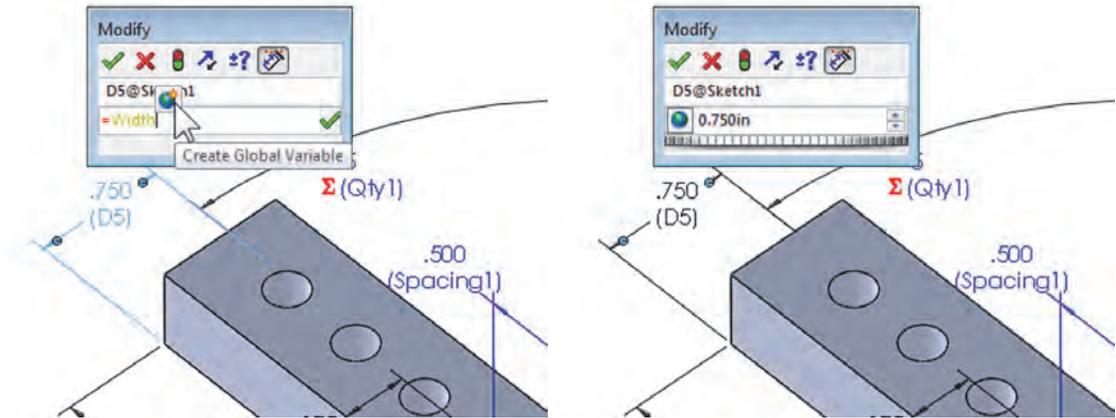
Now “Qty2” evaluates to 7. Both “Qty1” and “Qty2” have the red Σ symbol added letting us know that these dimensions are equation driven.



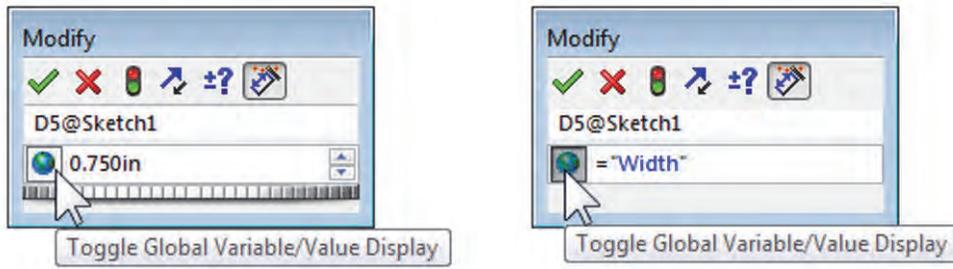
120. – To test the equations, change “Length1” to 2.75” and “Length2” to 4.5”. Rebuild the model. Now we have 5 holes in the left side and 8 holes in the right side.



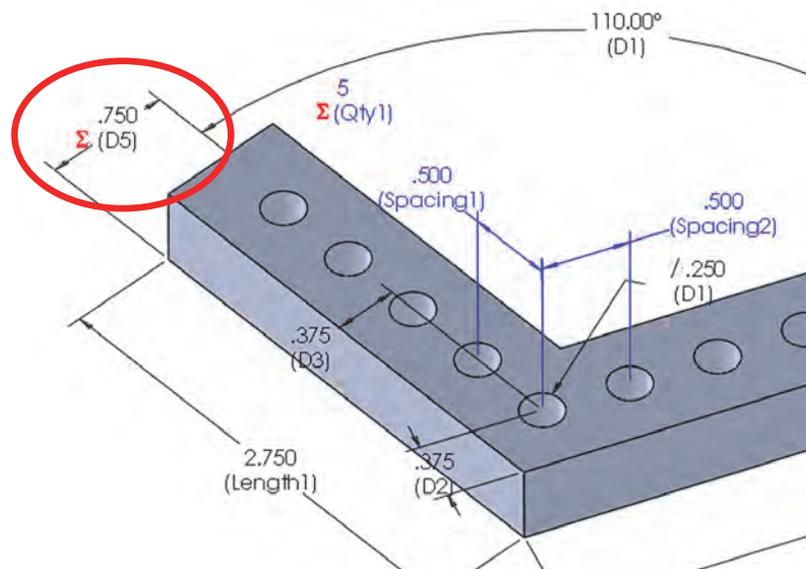
121. – Another way to maintain design intent is by using **Global Variables**. A Global Variable is essentially a constant with a name; after we create a variable it can be used in equations. Double click in one of the 0.75” dimensions and enter an ‘=’ (equal sign) followed by the variable name “Width.” After entering the name, a Global Variable icon is displayed. Click in the icon to create a new Global Variable and at the same time make the dimension’s value equal to it. Click OK to continue.



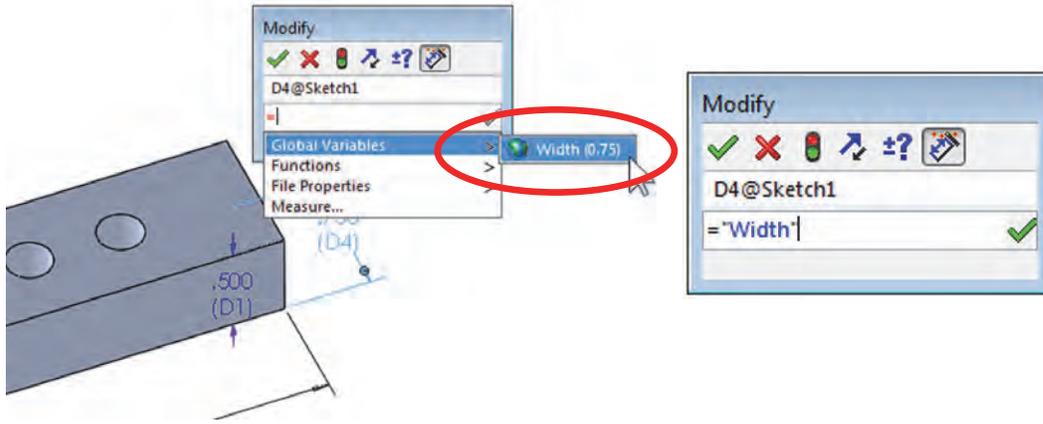
We can switch from viewing the dimension's value or the Global Variable name using the toggle button on the left side of the Modify dialog box.



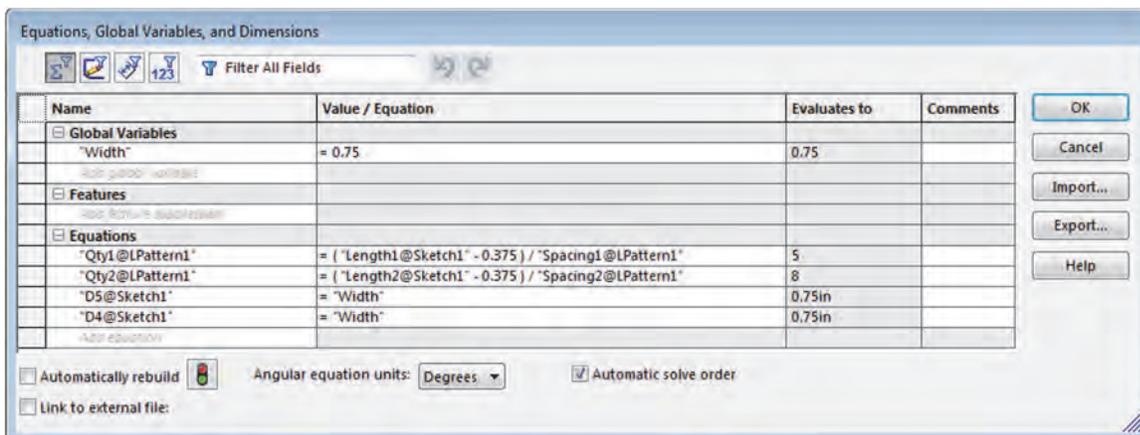
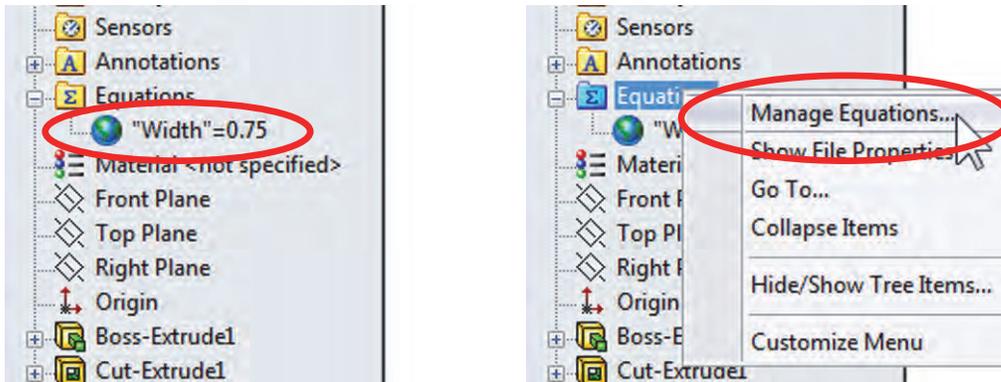
122. – After creating the Global Variable, the value is displayed with the red Σ symbol letting us know that the value is now driven by an equation. What happened was that the Global Variable was created, and at the same time an equation was added making this dimension equal to the “Width” variable.



123. – To illustrate how to make a dimension equal to a Global Variable, double click in the other 0.75" dimension, enter an '=' (equal sign), and from the drop down menu select "Global Variables, Width" and click OK to finish.



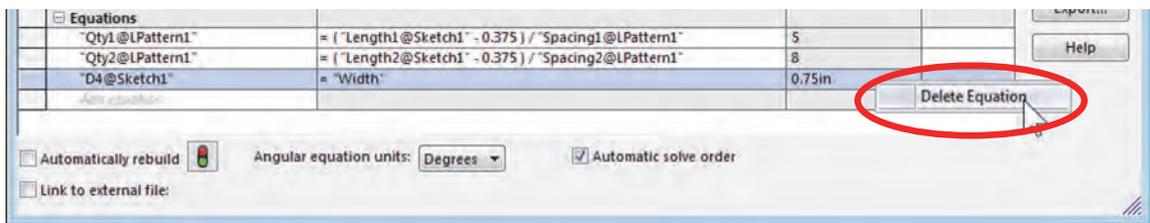
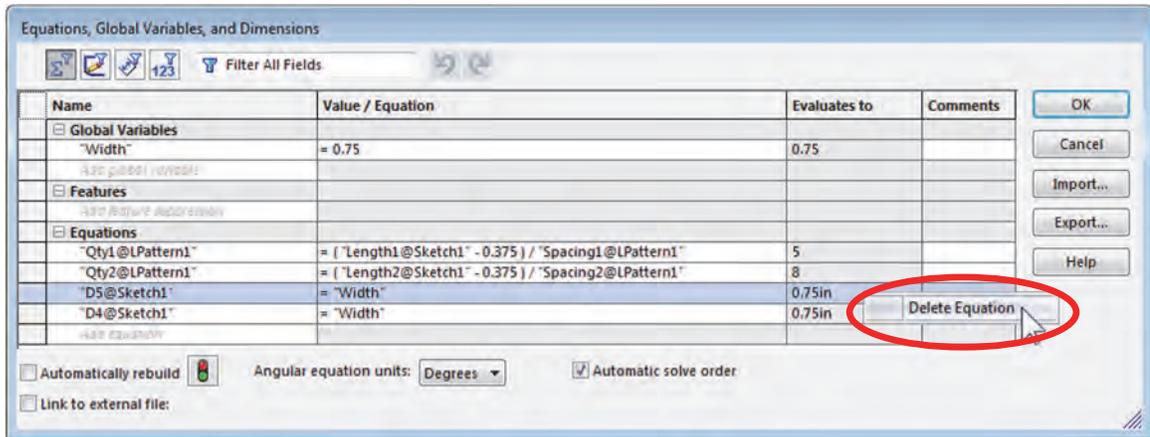
What we just did was to create a new equation making this dimension equal to the Global Variable "Width." Global Variables are automatically listed under the "Equations" folder in the FeatureManager and in the "Equations" dialog box. To change a Global Variable or an equation, right mouse click in the "Equations" folder and select "Manage Equations."



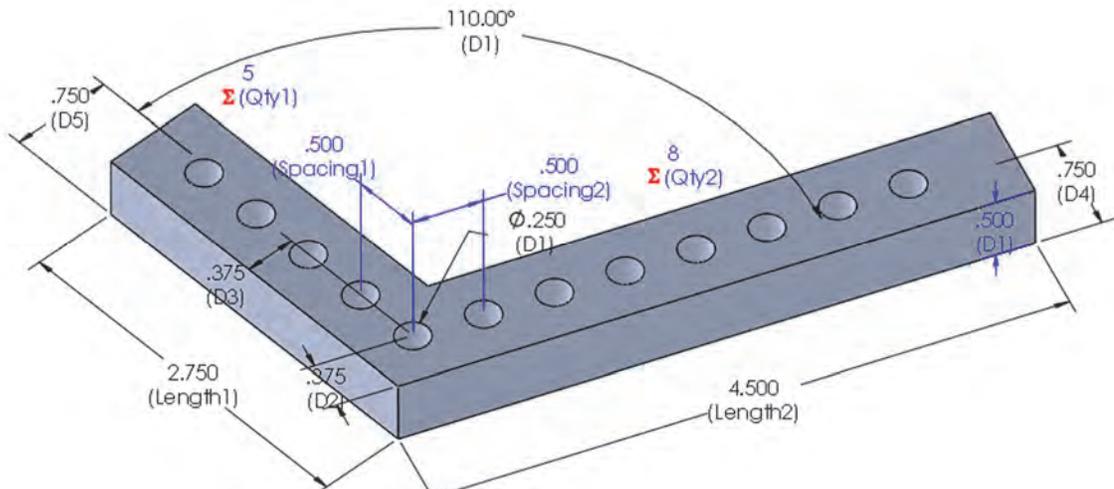


In equations we can't change the dependent dimensions directly ("Qty1" and "Qty2"), only the referenced dimensions ("Length1," "Length2," "Spacing1" and "Spacing2"). Global Variables allow us to update multiple dimensions at the same time, including when used in equations.

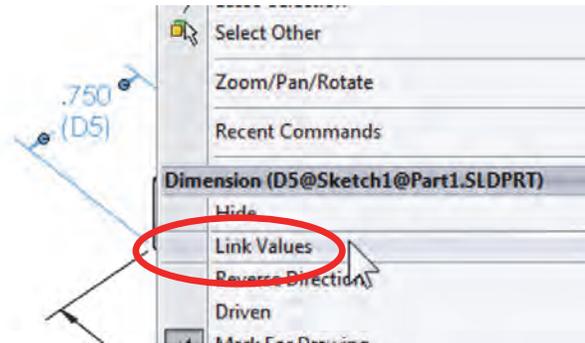
124. – In the "Equations" dialog we can Add, Edit, or Delete. To delete an equation, right-mouse-click in the equation, and select the option from the menu. Delete the last two equations in the list to learn a different way to make multiple dimensions equal to the same value.



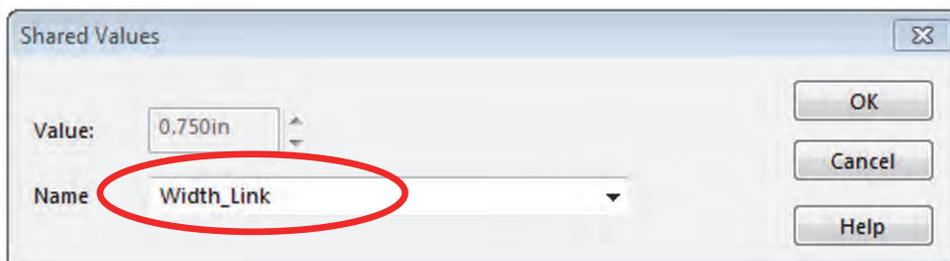
125. – Now our list only shows two equations. Click OK to close the equations dialog. After deleting the two previous equations, the dimensions are no longer preceded by the red Σ .



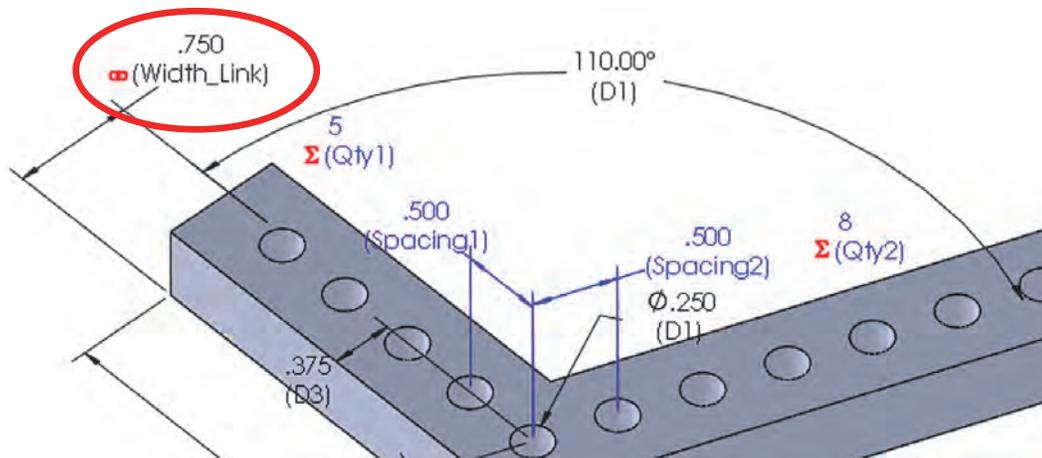
126. – A different way to make multiple dimensions equal to each other is by using the “**Link Values**” command. It is similar to making a dimension equal to a global variable as we did before; the difference is that we don’t have to create a variable, and unlike using an equation where we need to change the Global Variable in the equations editor to modify its value, by linking values we can change it directly in any linked dimension. Right-mouse-click in one of the width dimensions, and select “**Link Values**.”



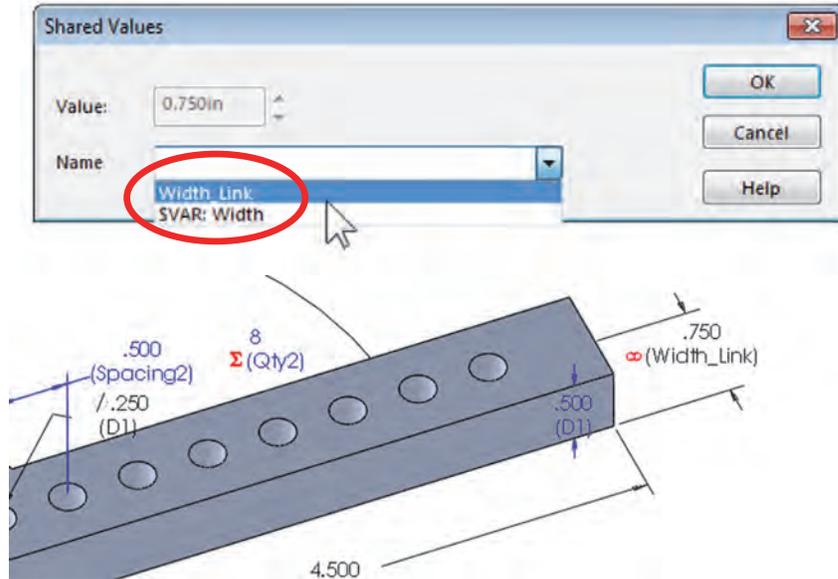
127. – In the “Shared Values” window enter ‘*Width_Link*’ in the name field. (Do not use ‘*Width*’; the name was already used.) Click OK to continue. Note that we cannot change the dimension’s value.



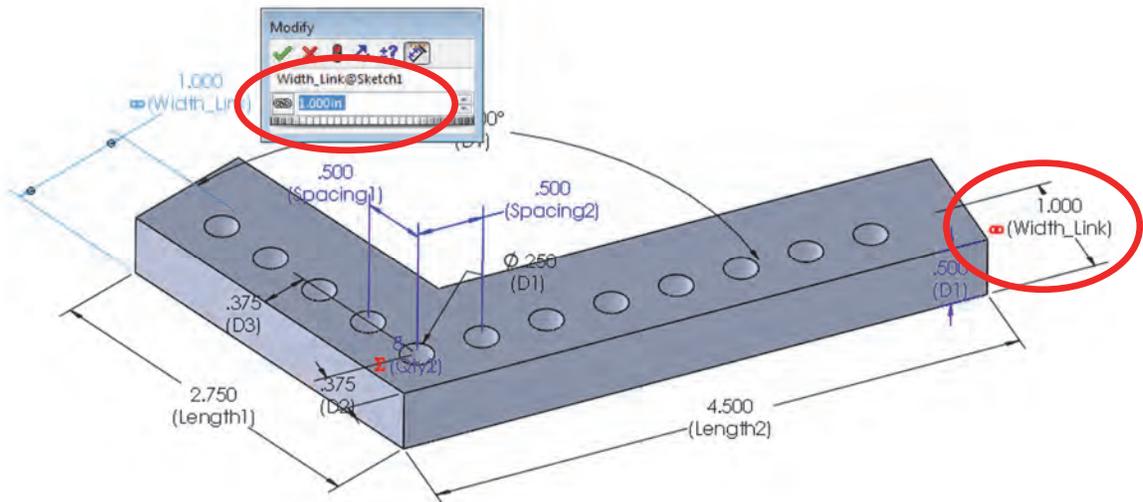
128. – After linking the dimension, its name is changed to “*Width_Link*” and a chain-link icon is added letting us know it is linked to a named value.



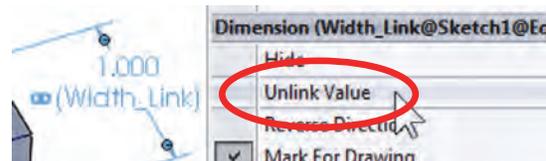
129. – To link the second dimension, right-mouse-click in it and select “**Link Values**” as we did for the first dimension, but instead of entering a name, select “*Width_Link*” from the drop-down list and click OK. Notice the previously made “*Width*” global variable is also listed as an option formatted as “\$VAR: *var_name*”



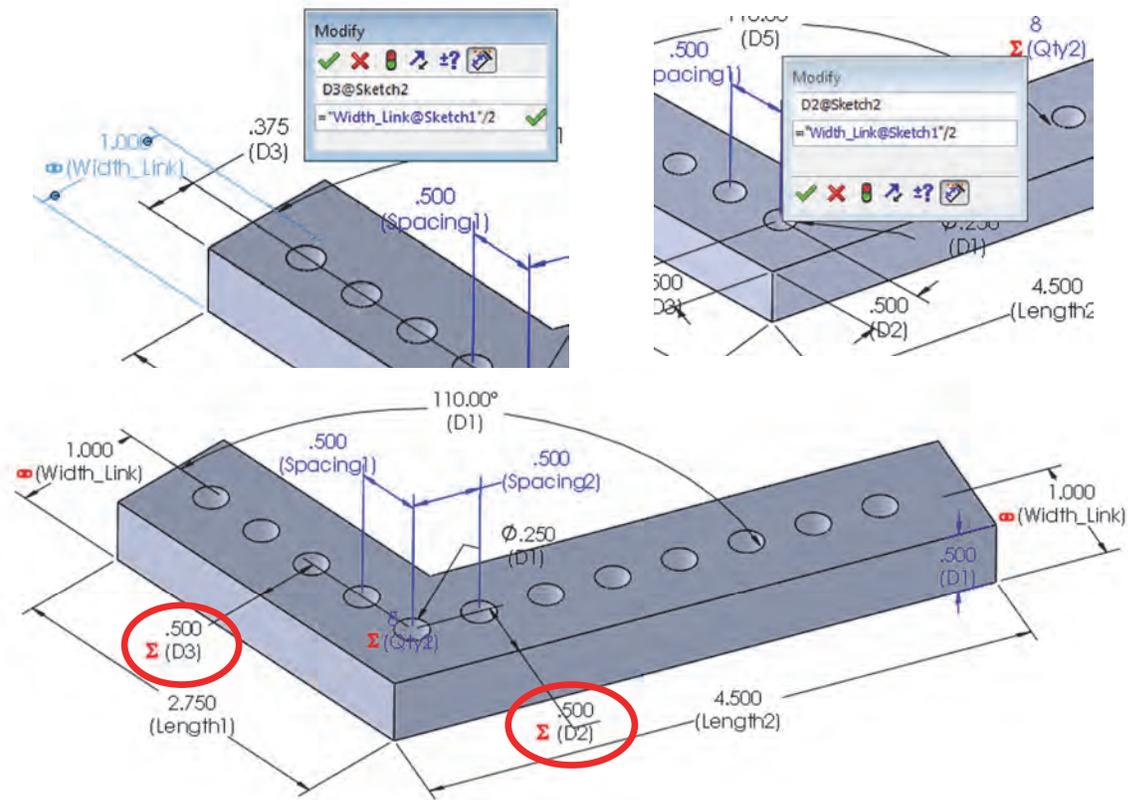
130. – Change the value of either linked dimension to 1” and rebuild the part to see the other dimension update at the same time. Note the chain-link icon preceding the value alerting us that the dimension is linked to other dimensions.



To un-link a value right-mouse-click in it and select “**Unlink Value**” from the menu.



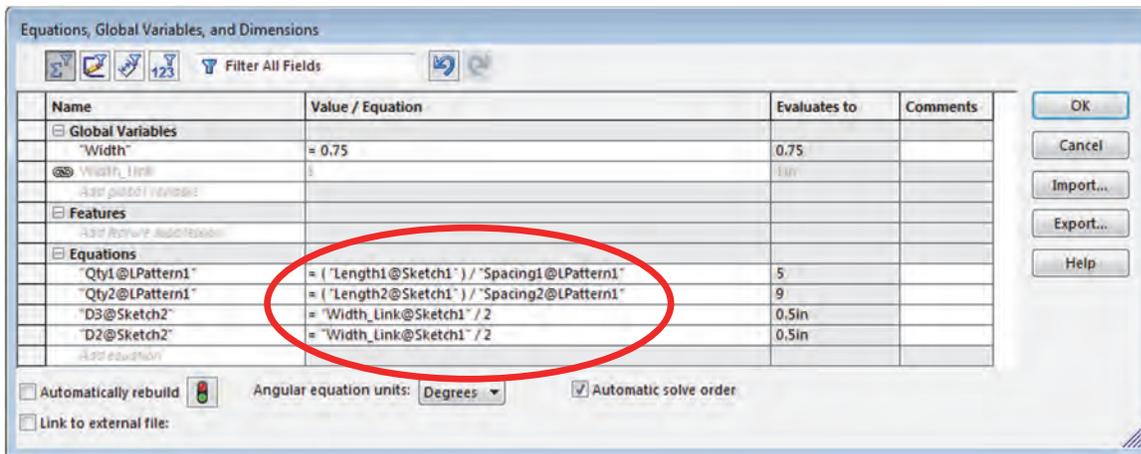
131. – Add two new equations to center the holes about the “*Width_Link*” dimension. Optionally rename the dimensions. Select the dimension either in the screen or from the drop-down list. After rebuilding the part, the holes will be centered.



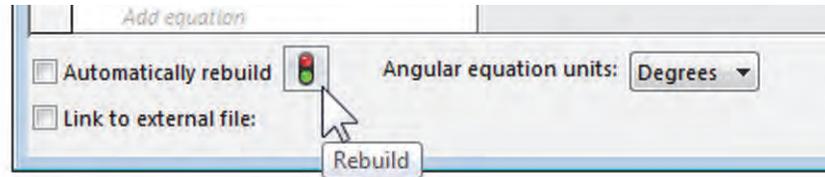
132. – To illustrate the effect of not subtracting 0.375” from the length and show how to edit equations, remove the ‘- 0.375’ portion from the two equations (it doesn’t matter if we leave the parenthesis or not). Our equations should now read:

$$\text{“Qty1@LPattern1”} = (\text{“Length1@Sketch1”}) / \text{“Spacing1@LPattern1”}$$

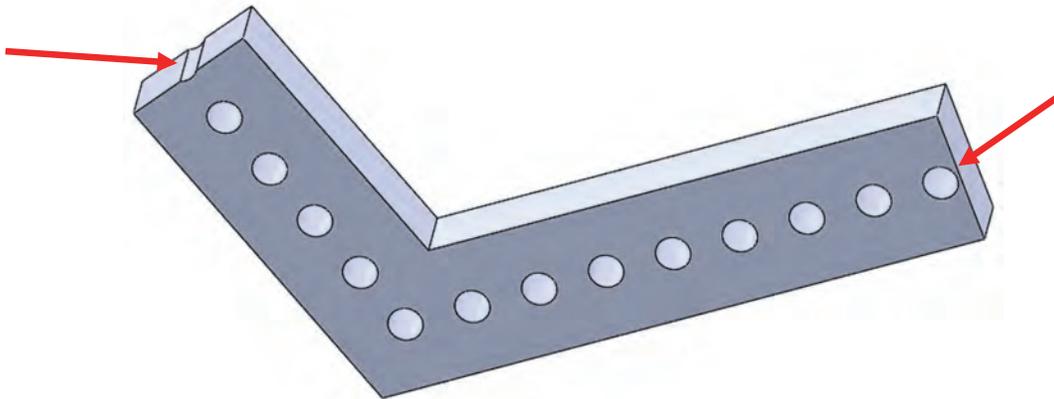
$$\text{“Qty2@LPattern1”} = (\text{“Length2@Sketch1”}) / \text{“Spacing2@LPattern1”}$$



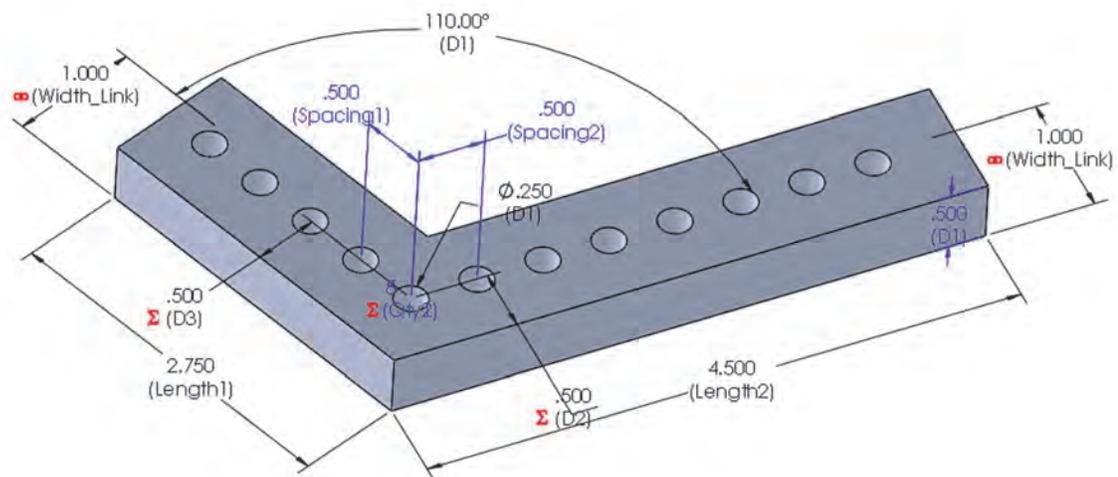
133. – Now the equations evaluate to 5 and 9 respectively. We can see the effects of the changes by rebuilding the model at the bottom of the dialog, or having them automatically rebuild as we make changes to them. Click OK to finish and rebuild the model to see the effect of the change.



What happened with this change is that a sixth instance is added that breaks the edge of the part in the short side and gets very close to the edge in the long side; the equation resolves to 5.5 and is rounded up to 6. That is the reason why we subtracted a distance of 0.375" from the length in the equations.

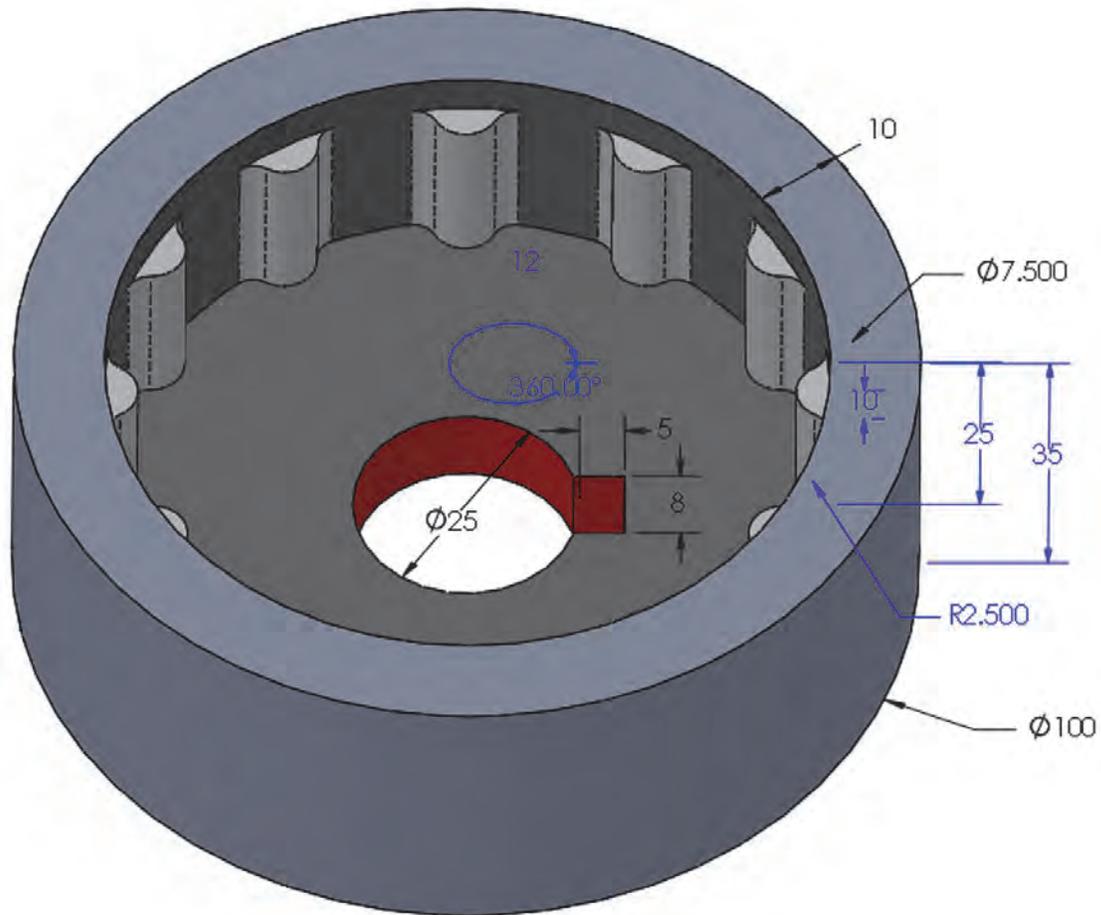


134. – Edit both equations again to subtract the 0.375" distance from the length, returning the equations to their original value. Save and close the part.



Exercise: Open the file *'Equations Exercise.sldprt'* from the included files and give all dimensions used in an equation a meaningful name. Add equations and use either Global Variables or Link Values to:

- Make the bottom thickness equal to the wall thickness (10mm dimension).
- Make the number of "Copies" equal to the outside diameter divided by 8.
- Make the "ShaftCut" feature diameter $\frac{1}{4}$ of the "Body" inside diameter.



In the SolidWorks equations, we can use almost any algebraic expression to evaluate values. As a general guide, you can write equations with the same algebraic format as you would in Excel formulas.