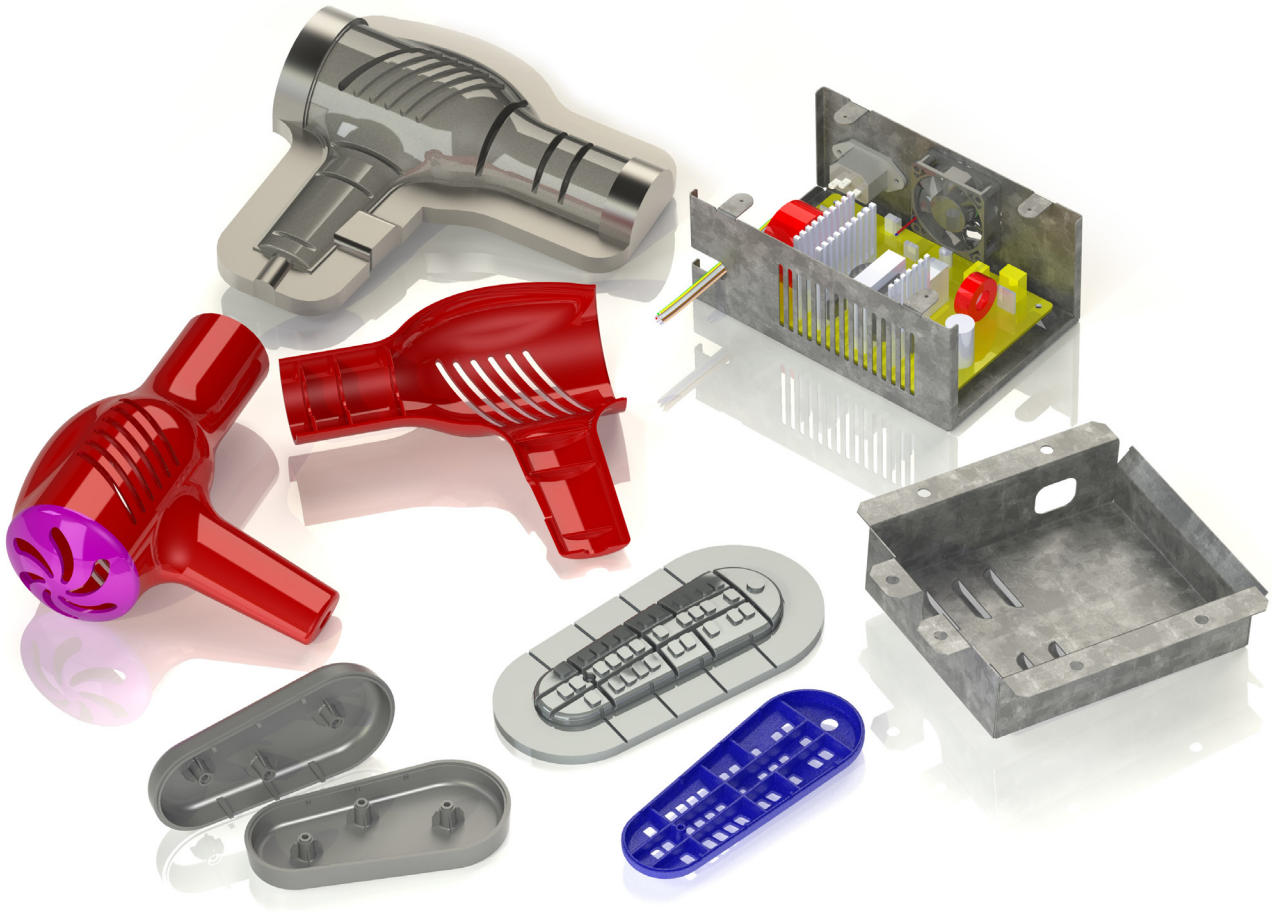
Beginner's Guide to SOLIDWORKS® 2019

Level II

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



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Chapter 1: Multi Body Parts, Editing and Other Tools

Multi body parts

SOLIDWORKS offers several 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 maybe 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 are the essence of the weldments environment (both covered in this book).

Contour Selection

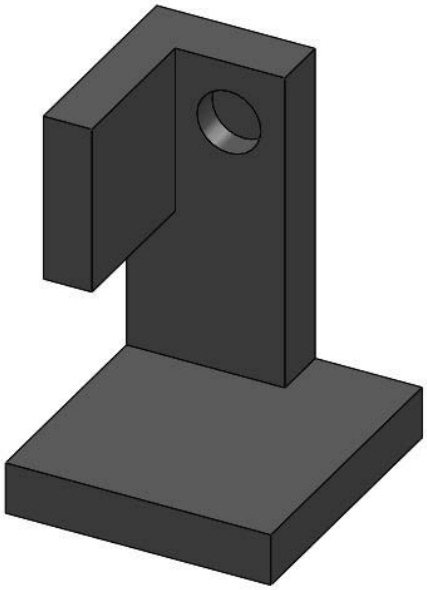
Besides working with single body parts, we have also worked with single and multiple contour open or closed sketches. In this section, we'll cover how to use Contour selection to work with sketches that contain multiple entities sharing an endpoint and/or having intersecting lines. We’ll learn how to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

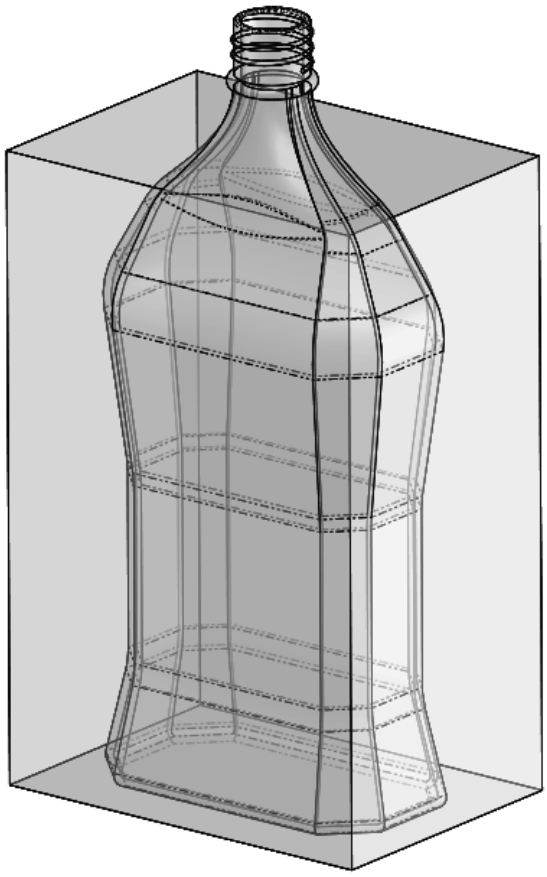
Part Editing

Another important topic when modeling in SOLIDWORKS is editing parts to fix an error or make a change to a component. 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 in both part and sketch.

Equations

The 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.

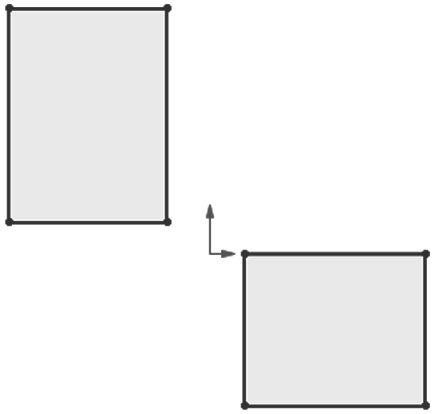




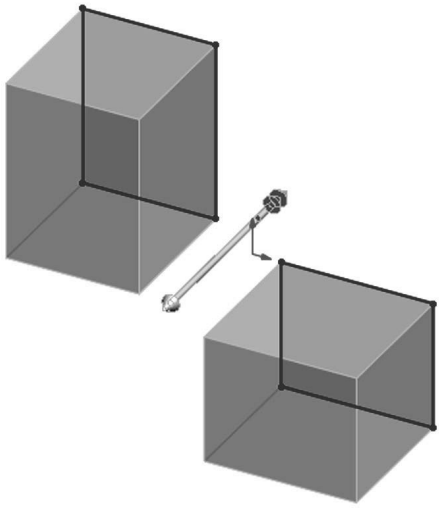
Multi Body Parts

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 the general concept of multi body parts, and then we'll look at different ways to use them.

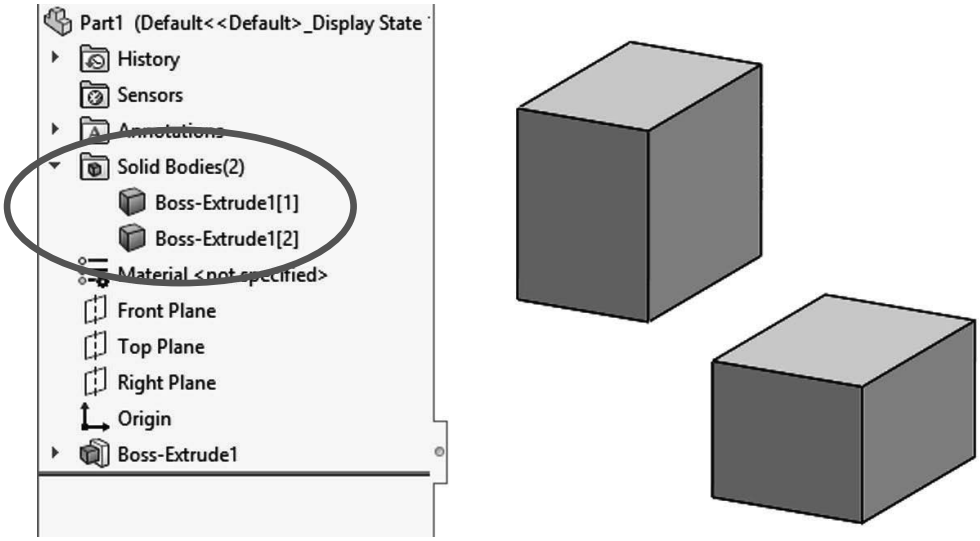
1.1. - Multi body parts are made by adding material that is not connected to the current solid body, in other words, that is purposely not merged with it. A different way to create a multi body part is by splitting an existing part into multiple bodies (this technique will be shown later). To show how a multi body part works, make a new part, and add a sketch to the *"Front Plane"* as shown. Do not worry about dimensions in this example, we are just illustrating the concept of how multi bodies work.



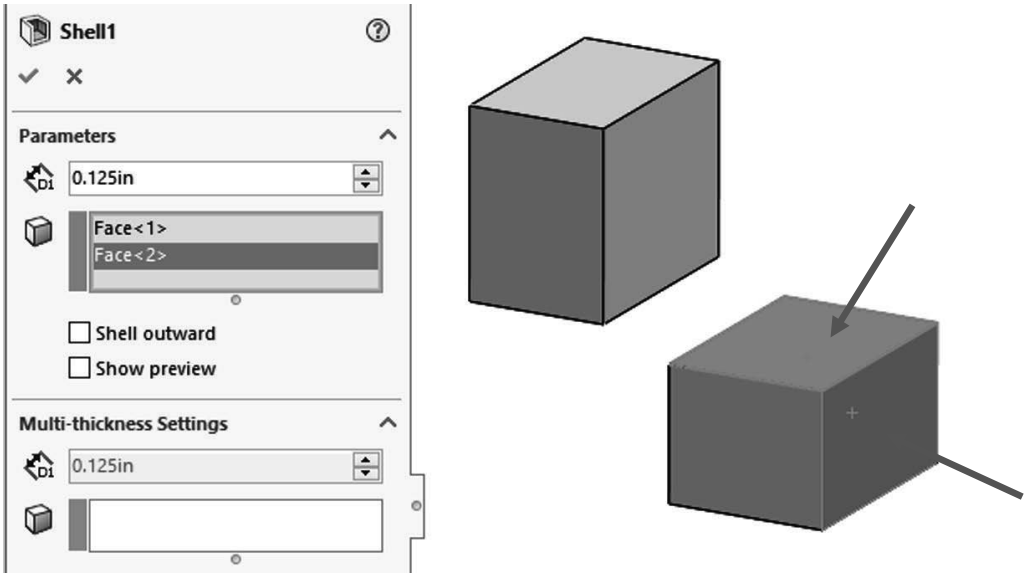
1.2. - Select the "**Extruded Boss**" command. Notice that we are not given a warning about having two separate bodies; it just works. Extrude the sketch any size that looks similar to the following image (dimensions are not important at this time) and click **OK** to finish. Making two or more extruded/revolved/swept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

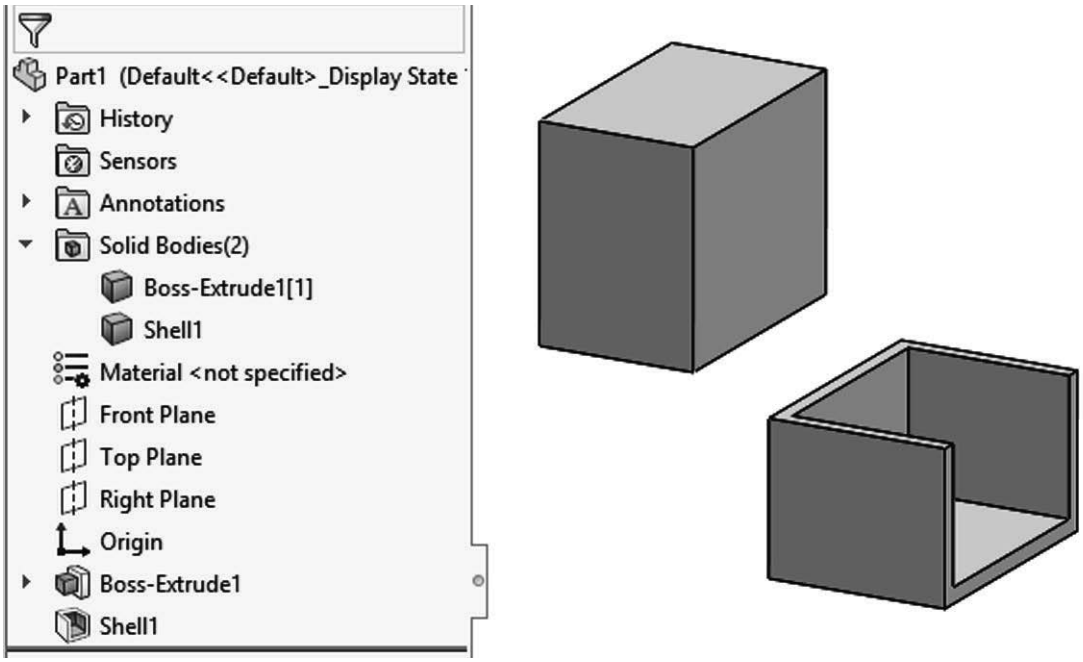


The first thing we notice is a new folder in the FeatureManager called "*Solid Bodies (2)."* This folder is automatically added when SOLIDWORKS detects multiple disjointed bodies in a part and lists the number of bodies found in the part (in this case 2). If we expand the folder, we can see the two bodies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

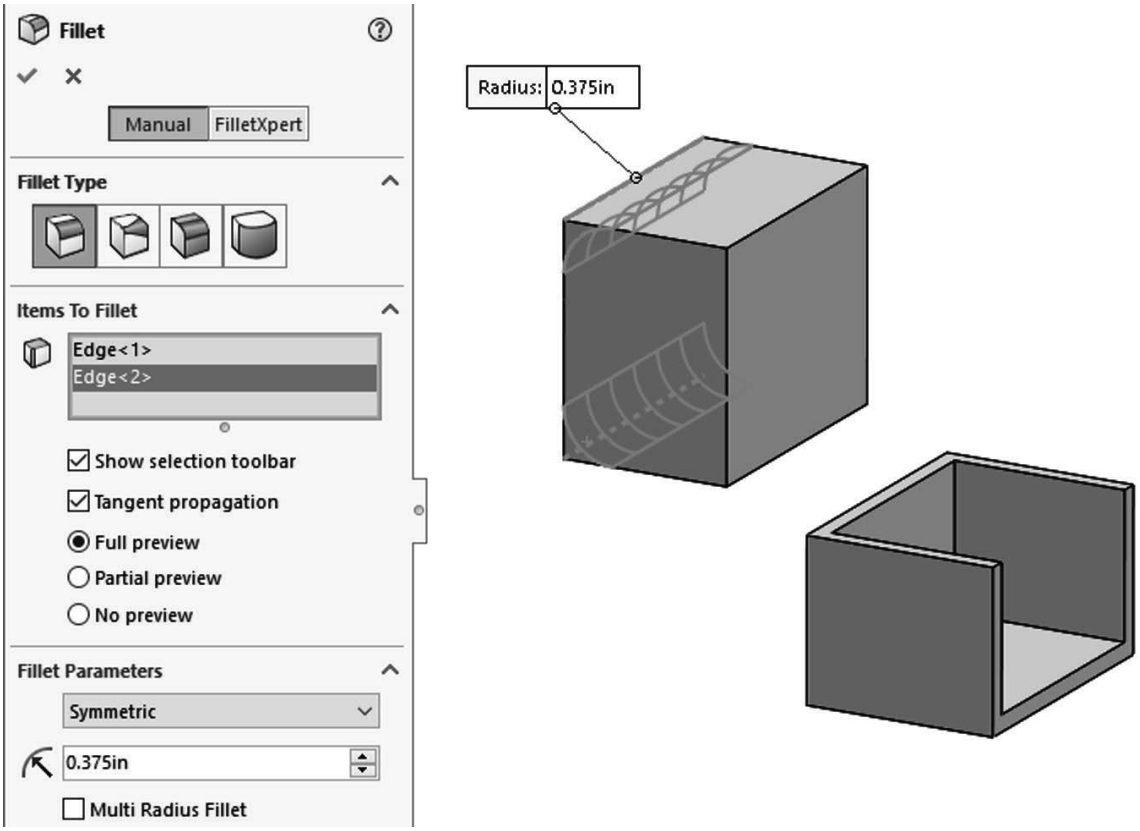


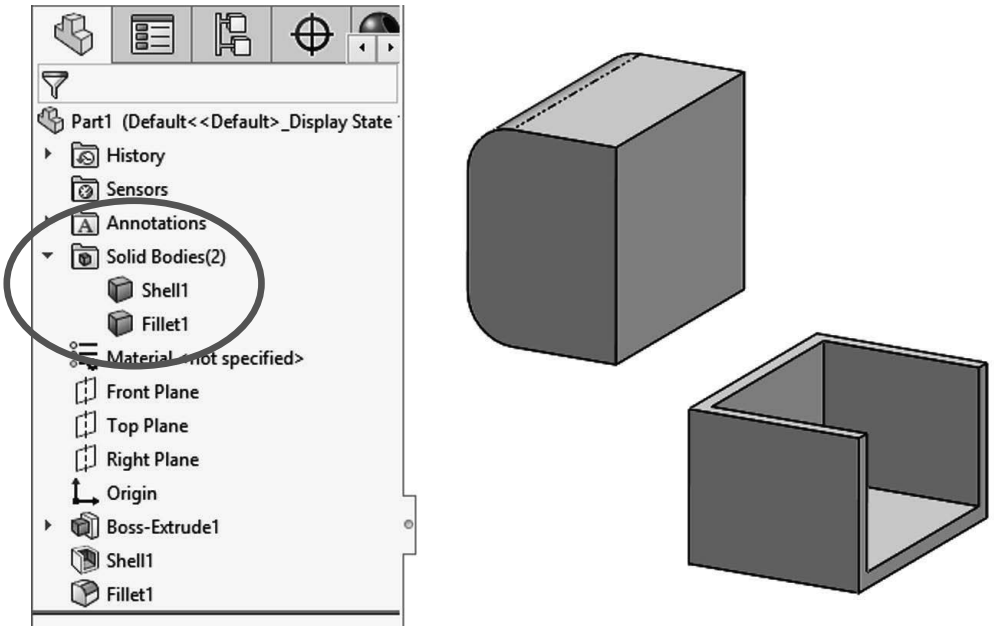
1.3. - The next step is to add a new feature. When working in a multi body part we can make 'local' operations to each body, in other words, a feature affecting only one solid body. Local operations are applied features like fillets, shells, chamfers, etc., which can only be applied to a single body at a time, for example we cannot fillet or shell both bodies at the same time. Select the "**Shell**" command from the Features toolbar and shell the bottom body removing the indicated faces.





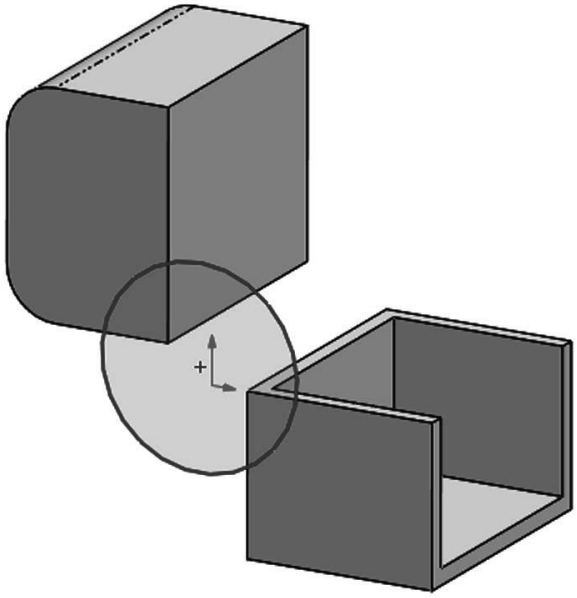
1.4. – Now we'll round a couple of corners in the other body. Select the **Fillet** command and round two corners of the top body as shown.





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 match the last feature applied to it.

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

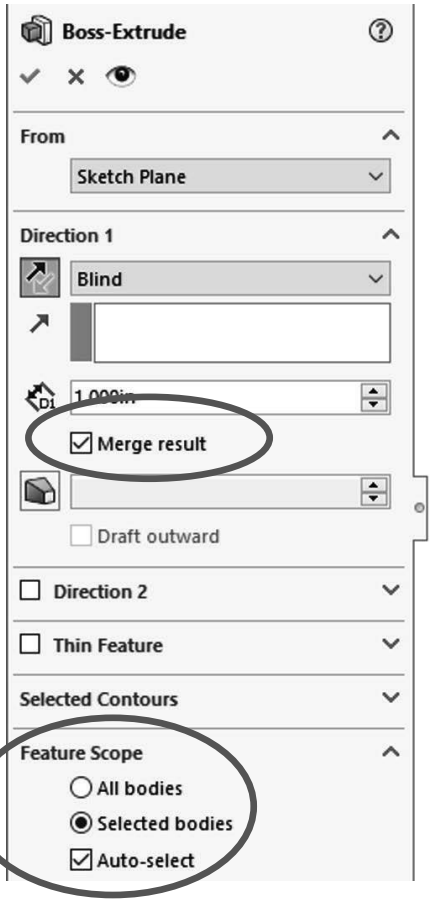


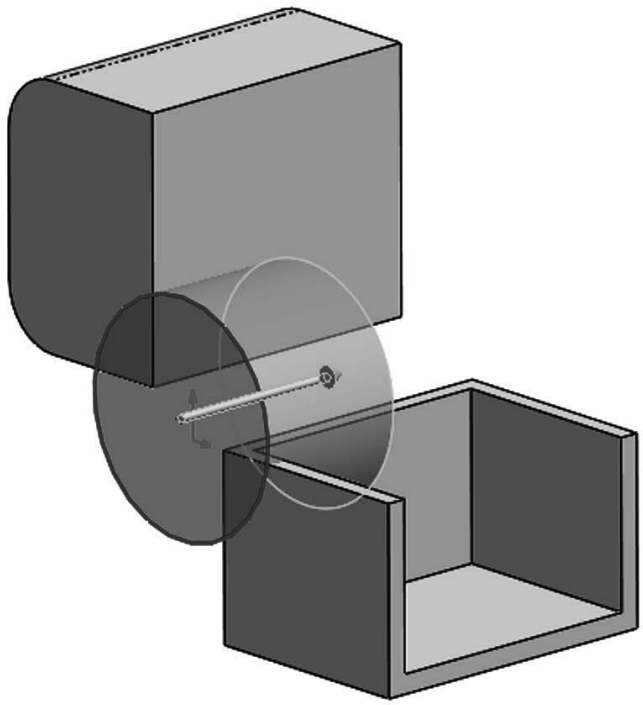
1.6. – Extrude the sketch *into* the existing bodies. (The same would happen if we extrude *away* from the bodies because they are touching the new extrusion.)

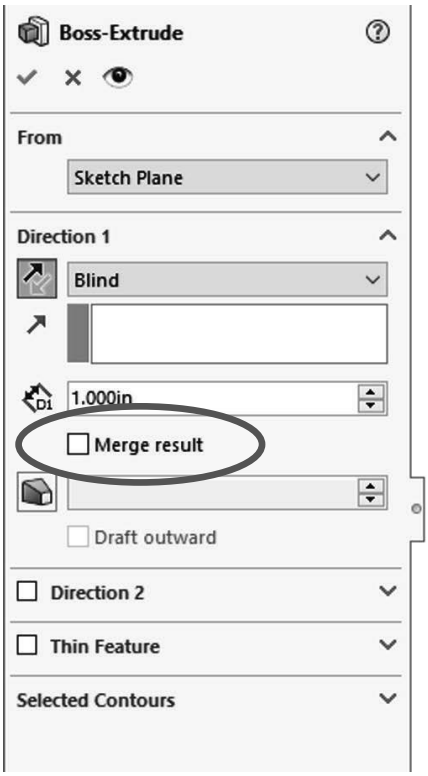
Notice the “Merge result” option in the Extrude command. It’s always been there (except when there are no solid bodies) and by default is always checked. When a part has multiple bodies, an additional option at the bottom called “Feature Scope” is shown. This is where we can select the existing bodies we want to merge the new extrusion with.

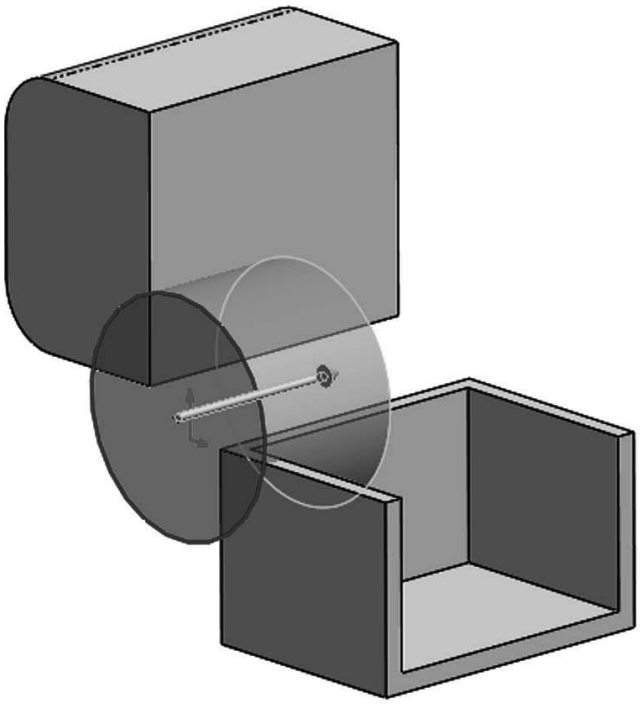
The “Merge result” options for the new extrusion automatically includes “All bodies” or manually “Selected bodies.” By default, the “Feature Scope” option is set to “Selected bodies” and “Auto-select.” These two options mean that, by default, the new feature will merge with any solid body it intersects or touches. If the “Merge result” option is un-checked, “Feature Scope” is automatically removed, and the new extrusion will not be merged with any solid body.

For this step uncheck the option “Merge Result” and click OK to finish.

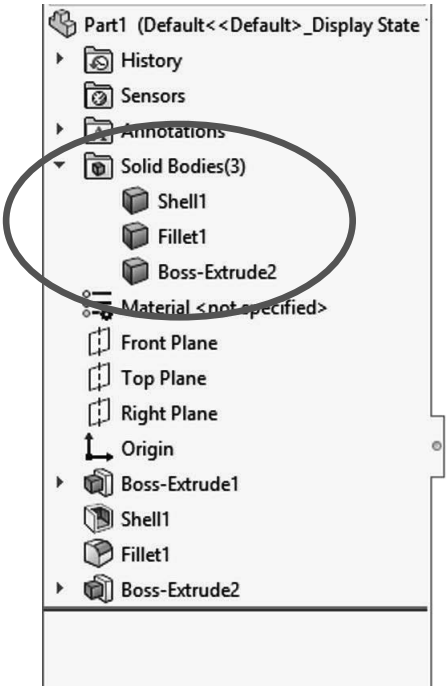


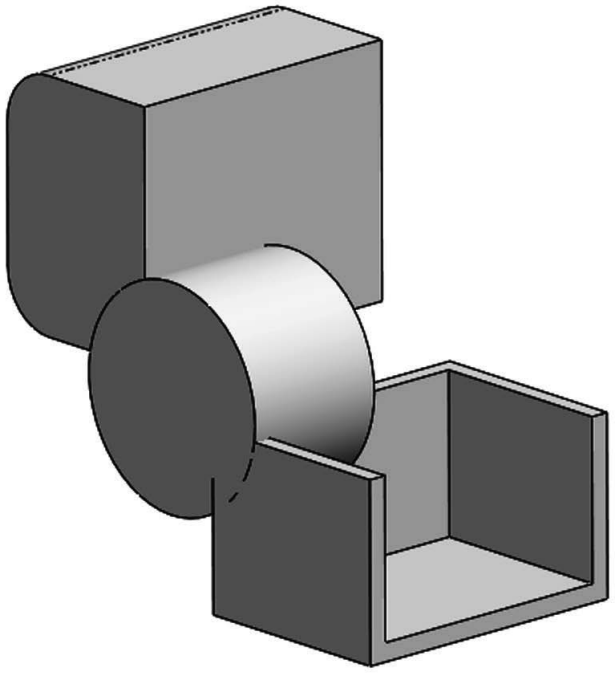




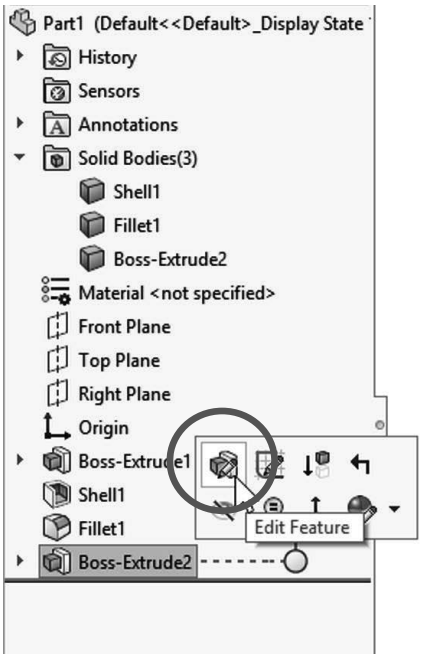


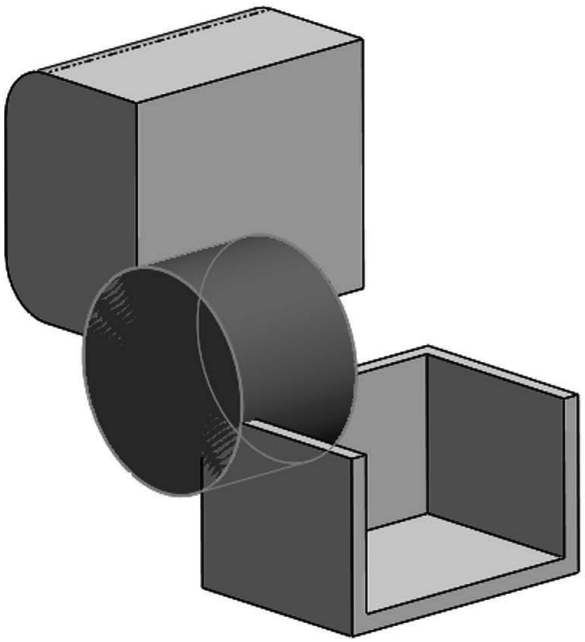
1.7. - After adding this extrusion we have three different bodies in our part. See how the different bodies' edges intersect each other. If the "Merge result" option had been checked, we would not see the overlapping edges as they would be merged into a single body.

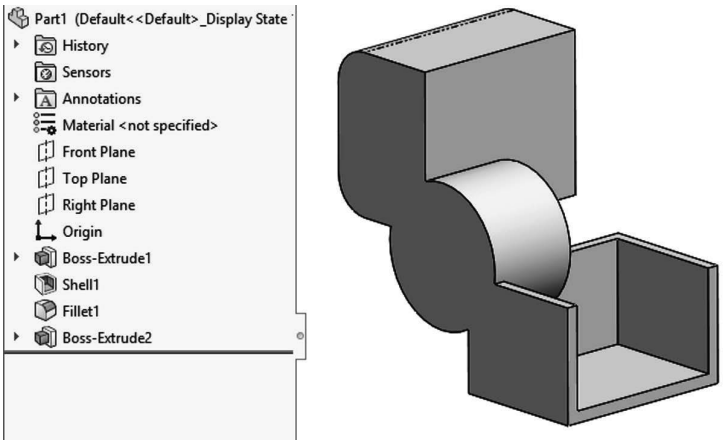


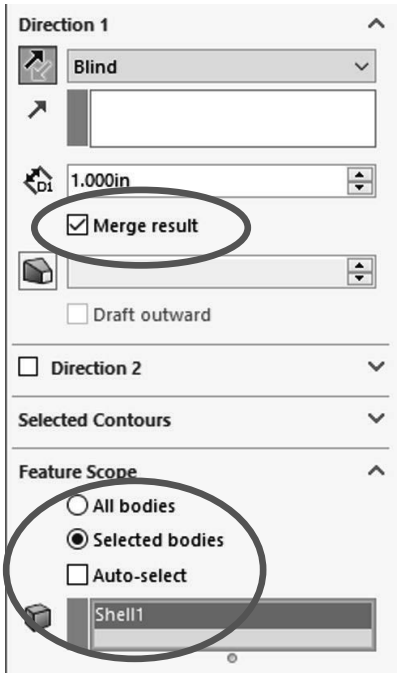
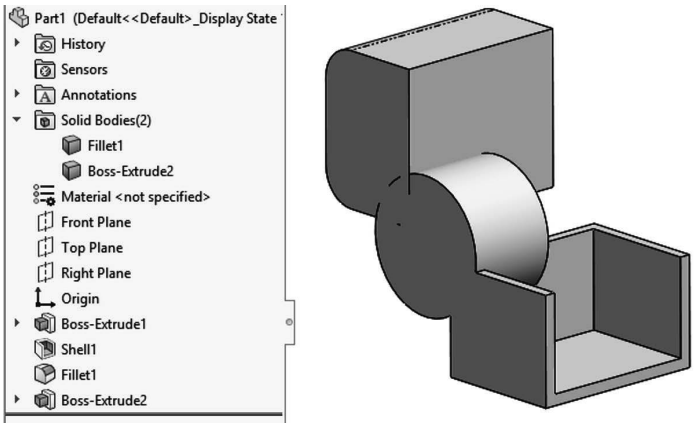


Edit the last extrusion to explore the effect of different “Merge Result” and “Feature Scope” combinations.

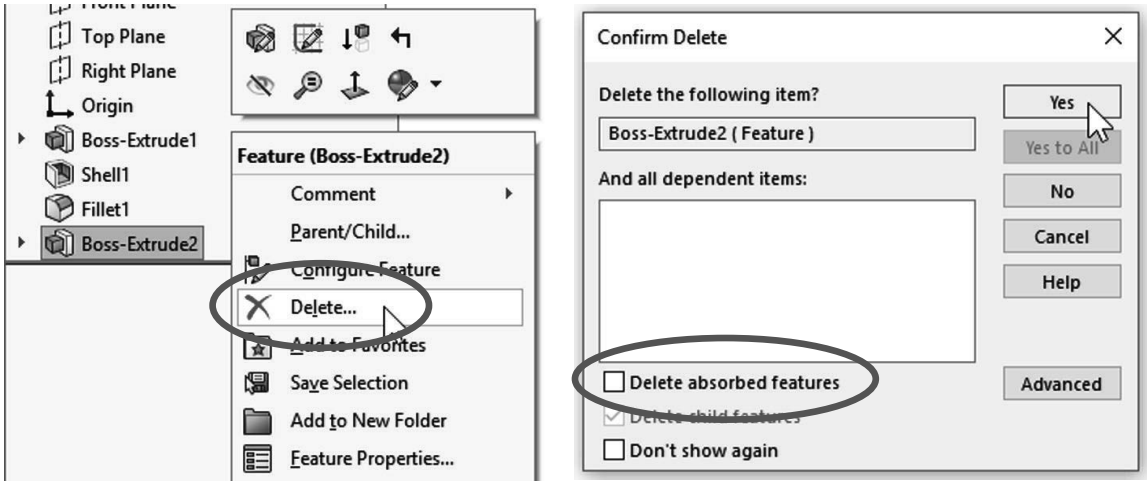




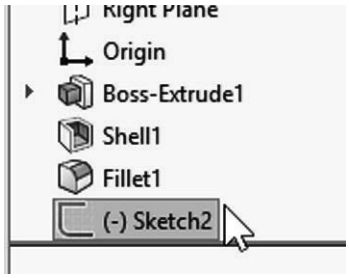
Combination:	Result:
Merge Result: Checked **Feature Scope**: Selected Bodies, Auto-Select 	Single body. By touching both of the existing bodies “Auto-select” merges with them, fusing them into a single solid body. The “Solid Bodies” folder is no longer visible as there is only one solid body in the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

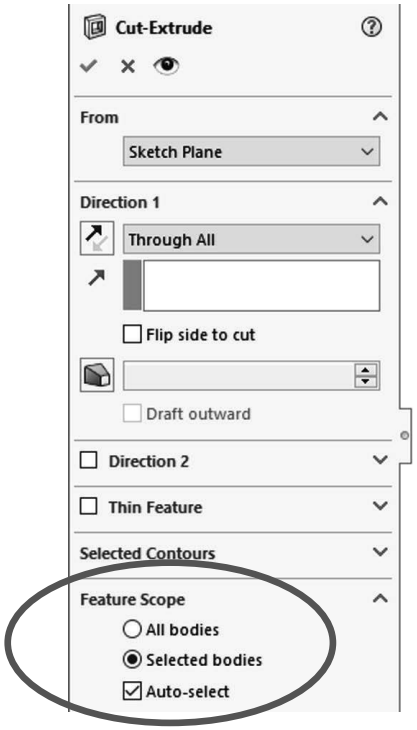
Merge result: Checked **Auto-select**: Unchecked, Add shell body to selection box 	Two bodies. We are telling the Boss-Extrude to merge only to the Shell body (or whichever body we pick). Notice the edges of the upper body and the Boss-Extrude overlap, as they are different bodies. 

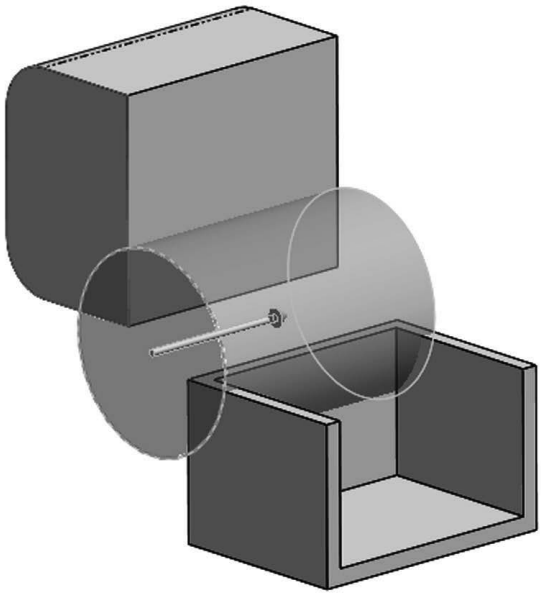
1.8. - The “Merge result“ option works the same way with any feature that adds or removes material to the part, including revolved boss, sweep, loft, etc. Now we’ll see how it works when material is removed. Delete the last Boss-Extrude feature and keep the sketch. When deleting the “Boss-Extrude,” make sure the option “Delete absorbed features” is unchecked; this way the boss will be deleted, but the sketch will be left behind.



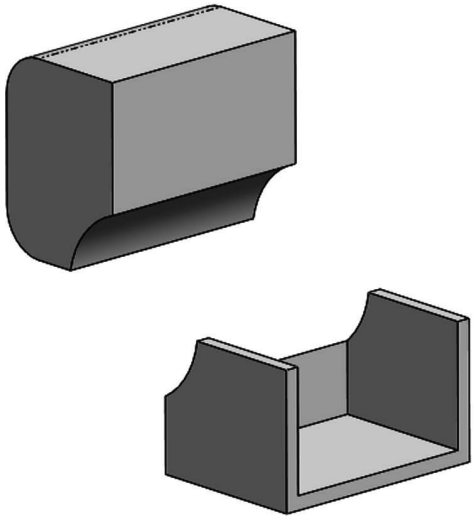
Select the sketch in the FeatureManager and make a Cut-Extrude feature using the “Through All” end condition. In this case the “Merge Result” option is replaced by “**Feature Scope**” with the same selection options: “All bodies” or “Selected Bodies” with/without “Auto-select.” Leave the “Auto-select” option checked and click **OK** to finish.



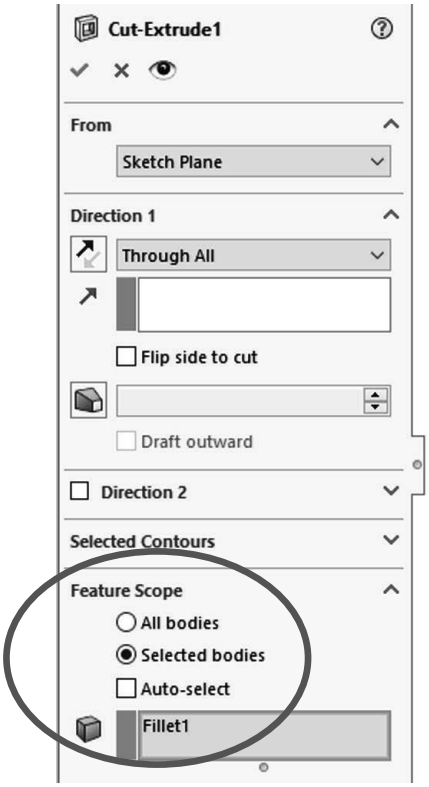


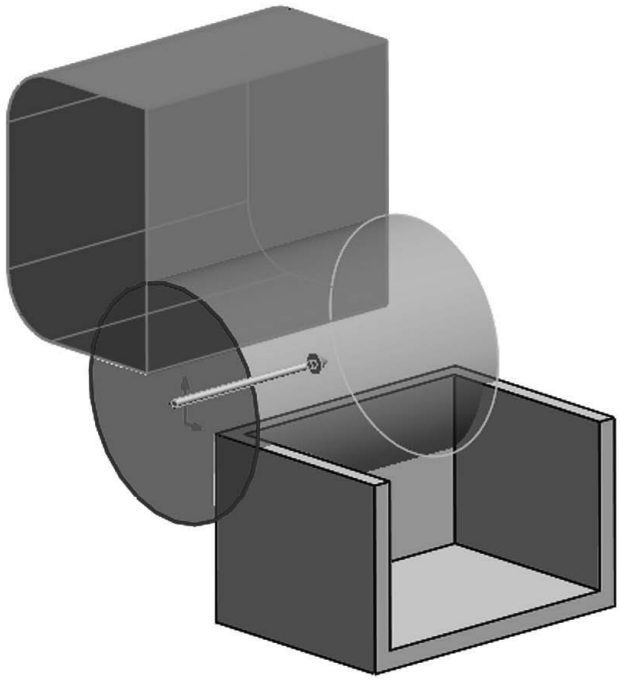
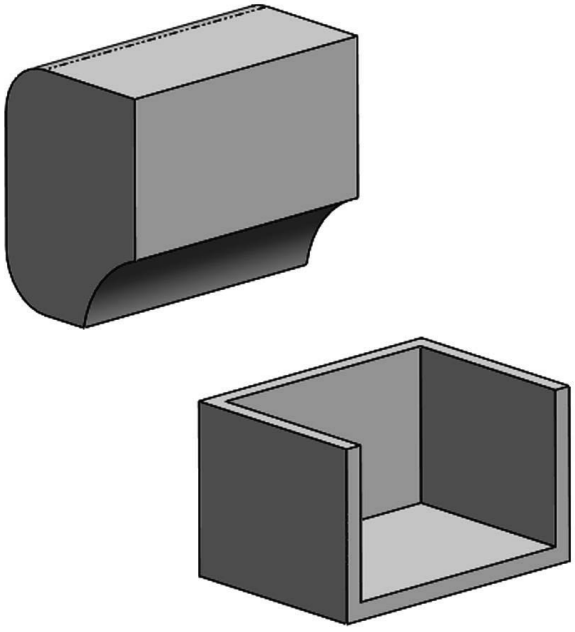


1.9. - What we end up with is the same two bodies we had before, but now they have a cut through them because both bodies were automatically selected by the Cut-Extrude command.



1.10. - Edit the "*Cut-Extrude1*" feature, turn the "Auto-select" option OFF in the "**Feature Scope**," and select the top body only. Click **OK** to finish. Now we are only cutting into the top solid, even if 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.