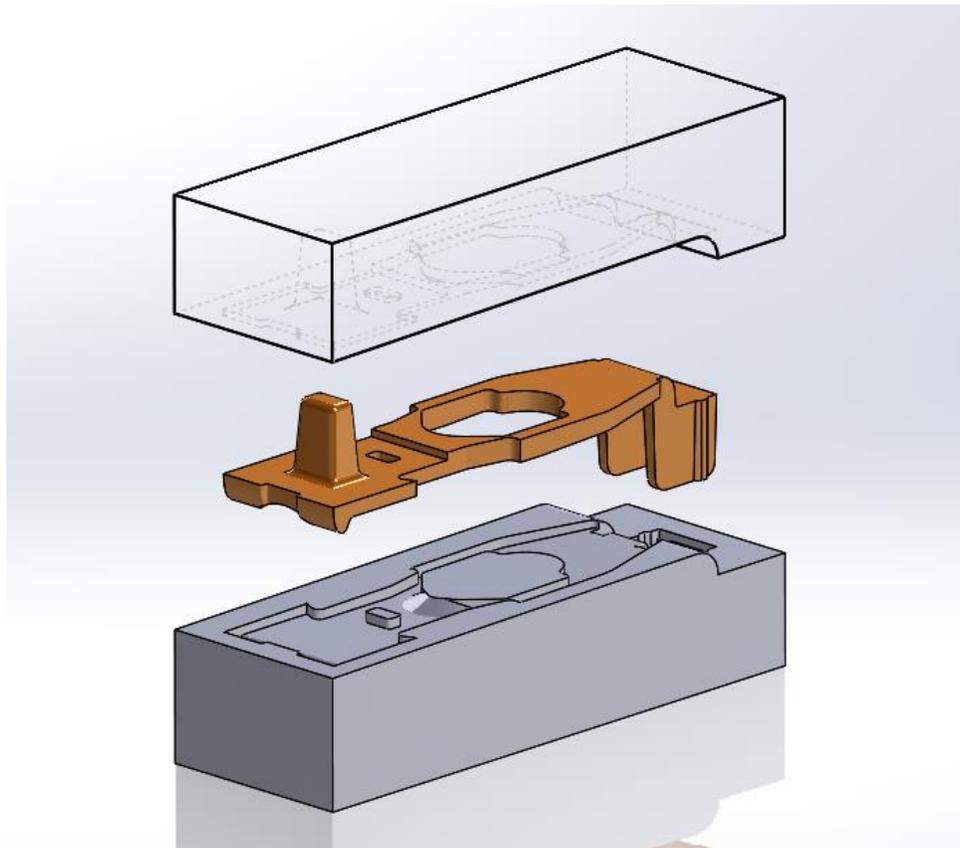


Mold Design: Creating a Cavity

TITLE:	Mold Design: Creating a Cavity
DATE:	December 2014
SUBJECT:	Solidworks Cavity Feature Usage
ABSTRACT:	Guide on using the Cavity Feature in Solidworks



This guide is written for SolidWorks 2014. The Mold Tools features suite has not changed between Solidworks 2013 and Solidworks 2014. You can use the Mold Tools features to create simple molds using the Cavity feature tool described in this document. The Cavity feature  will allow you to excavate material from an existing solid body without the need for complex surfacing features or other methods.

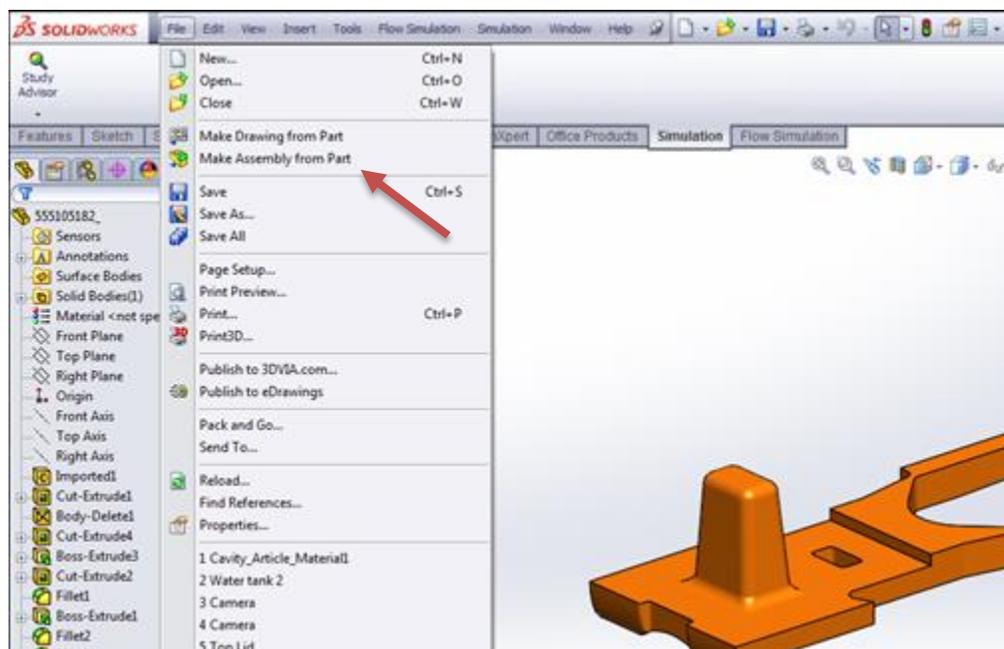
What you'll need

To create a mold using the Cavity feature , you will need the following:

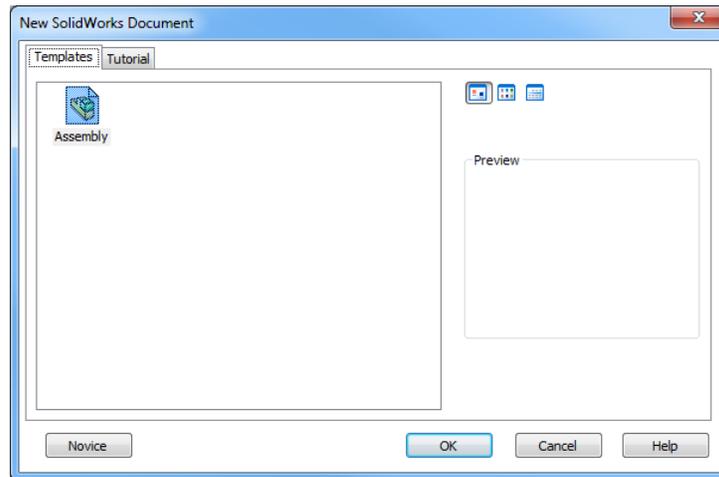
- Parent part(s) to create the mold from
- The foundation material that will contribute solid body mass to make the mold
- An assembly which the Cavity will be created in

Creating the Cavity Assembly

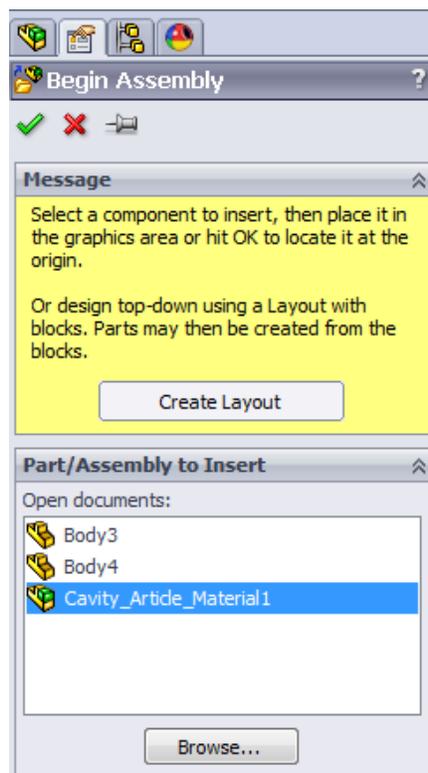
Insert the parent part(s) into an assembly. There are a number of ways to do this. Here, we show one method by **Clicking** on **File > Make Assembly from Part**.



Select the desired Assembly template from the next dialogue window.

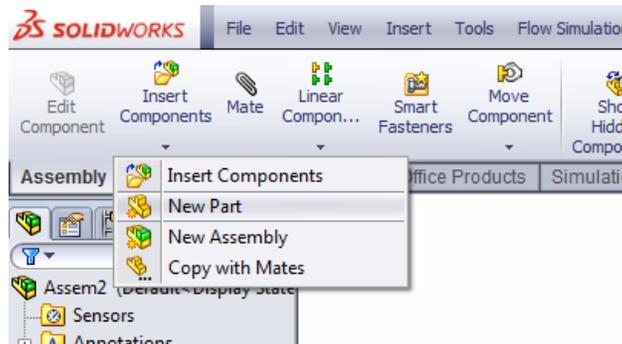


Select the Part(s) you will want to create the mold from in the Begin Assembly Property Manager. Before clicking anything, you can **Click OK**, and the parent part will be placed on the origin automatically. This will make it easy for you to create the mold body later on.

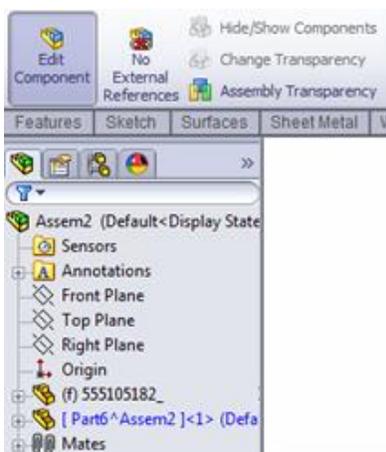


Creating the Base Material

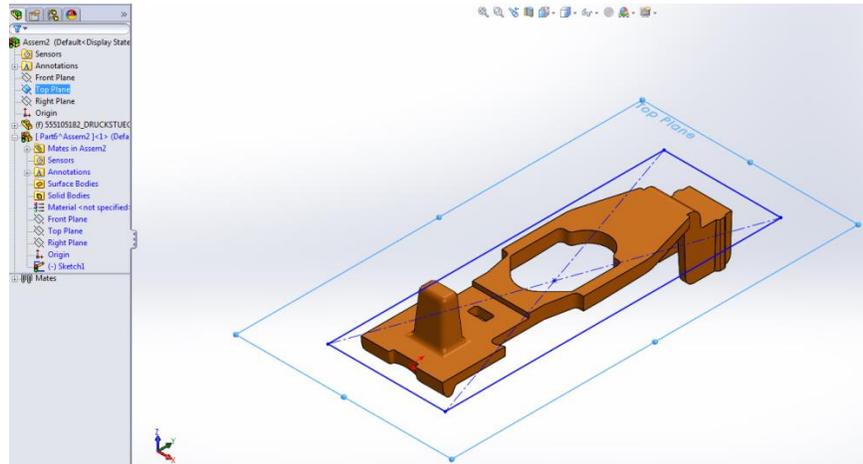
The component or assembly should now be at the Origin or location where you placed the component in the Graphics Area. Create the base material for the Cavity feature to be built from. To do this, **Select** the **Assembly** tab of the Command Manager (Default Settings), **click** on the drop down arrow below Insert Components and **select New Part**.



A new Part should appear in the **Feature Manager Tree**. It will simply be named "Part#" until it is given a new name. Begin working on the new Part by **selecting a Face or Plane** to begin the initial feature of the new Part. Once a Face or Plane has been selected, Solidworks will show **Edit Component** mode, where the component in the Feature Manager Tree is shown in blue.



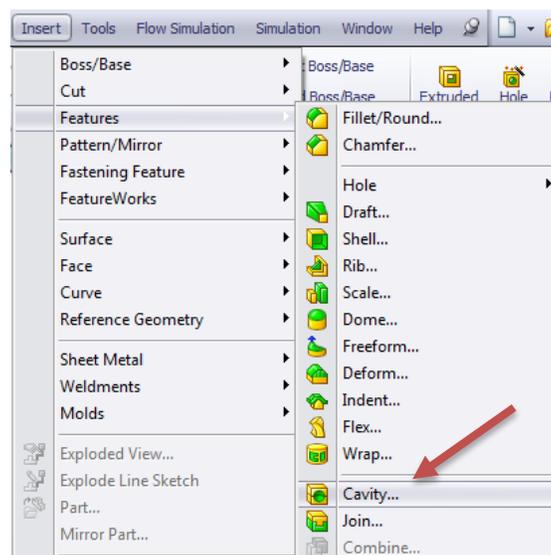
Create a closed contour that will define the Cavity base material Extrude feature and **select the Boss-Extrude** feature.



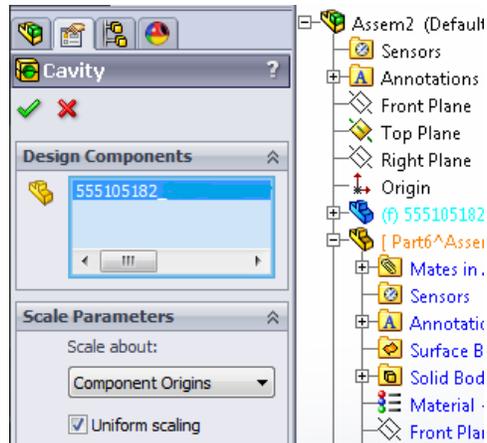
The Blind End Condition is chosen by default. **Change the extrusion depth** in Direction 1 (and Direction 2 if necessary) such that the entire body of the encapsulated component is covered.

Creating the Cavity Feature

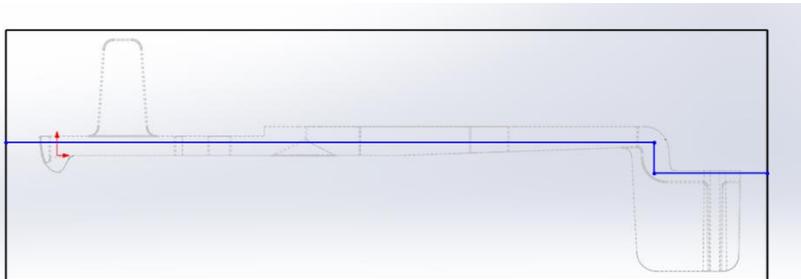
While still in Edit Component mode (within the newly created **Mold Base**), **select Insert > Features > Cavity** to create a new Cavity feature.



The **Cavity Feature Property Manager** should be displayed. **Select the component for the mold** to be created from (selecting from the Pop-out Feature Manager Tree is the simplest way to do this as the component typically will not be visible in the Graphics Area).



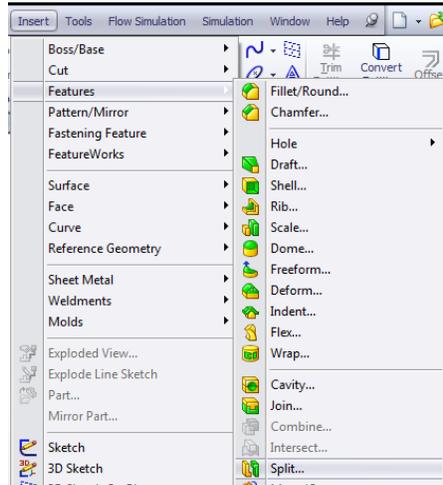
Internal to the Cavity Mold Base, the feature has created a cavity based on the outside contours of the inside component. In the image below displayed in **Hidden Lines Visible** mode shows the internal component and its outline within the Cavity Mold Base.



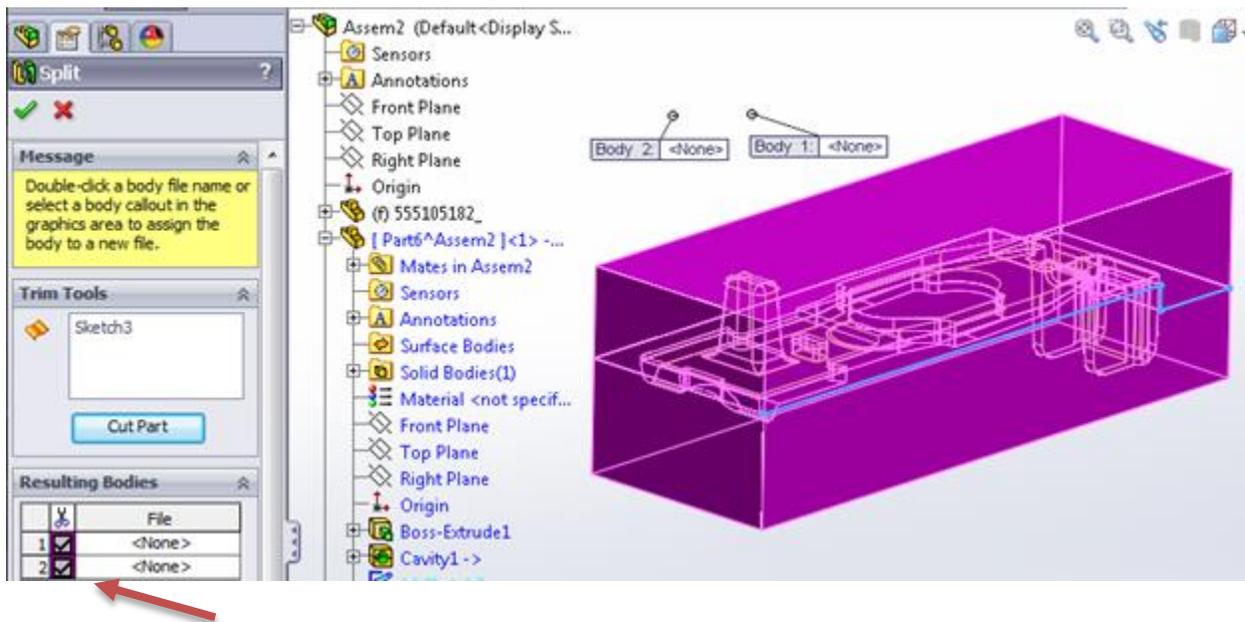
Create a sketch that adequately bisects the Mold Base for use in the following step. The sketch needs to pass completely through the Mold Base, end to end.

Separate the Cavity into Halves

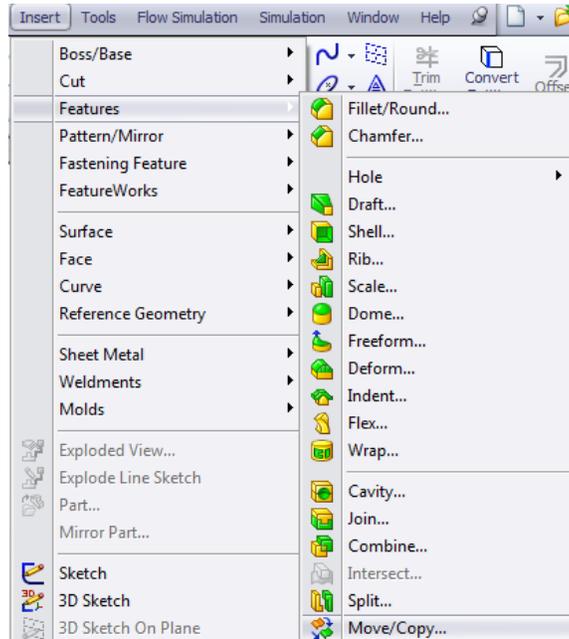
Use the **Split Feature** to separate the Cavity base material into two halves to create a mold. To find the Split feature, **click on Insert > Features > Split.**



In the **Split Property Manager**, use the newly created sketch that defines the parting line between the two halves as the cutting entity. **Click Cut Part** to cut the base material part. Also **check both boxes under Resulting Bodies** to have both halves saved as separate bodies.



The assembly will now contain two halves of the Mold Base, as well as the cavity component. Use the **Move/Copy** feature to separate the bodies of the Mold Base.



Here is an example of an assembly created using the Cavity feature:

