How to Draw a Coke Bottle Mold in SolidWorks?

Today I'll show you how to create a mold for a coke bottle in SolidWorks. We'll use the Cravity feature to do this. This tool can be very helpfull to subtract a shape from another shape. In this case we'll subtract the shape of a plastic bottle froma solid block (the mold). I will also […]

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Download this model of a Bottle **here** and Unzip it into your working map.

Open the file Bottle.SLDPRT

Go to: **File > New > Open > Bottle. SLDPRT**

NOTE: It's a bottle without an inner space to create the shape of the mold.

Now it’s time to draw the mold block

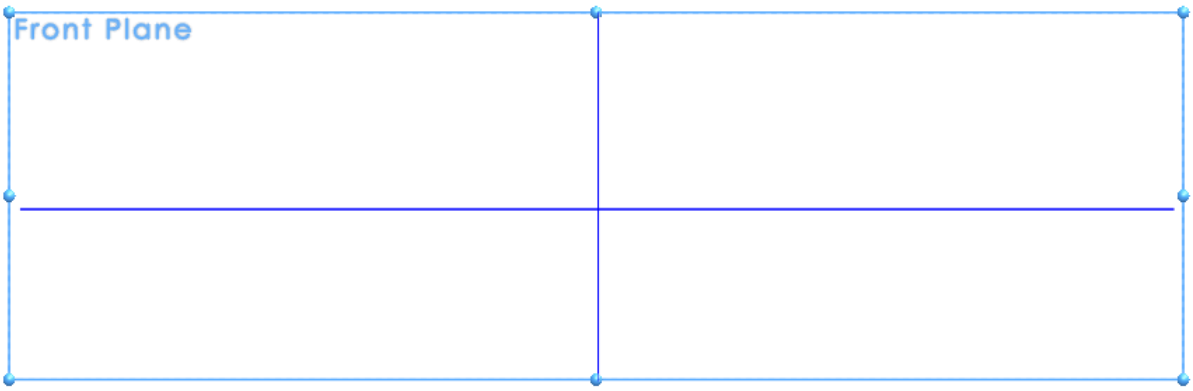
Open a new part with model units set to millimeters

Go to: **File > New > Part**

Create a 2D sketch

Select the Front Plane in the feature tree (menu at the left side) and create a sketch by clicking on the 2D Sketch icon

The display changes so the Front plane faces you.

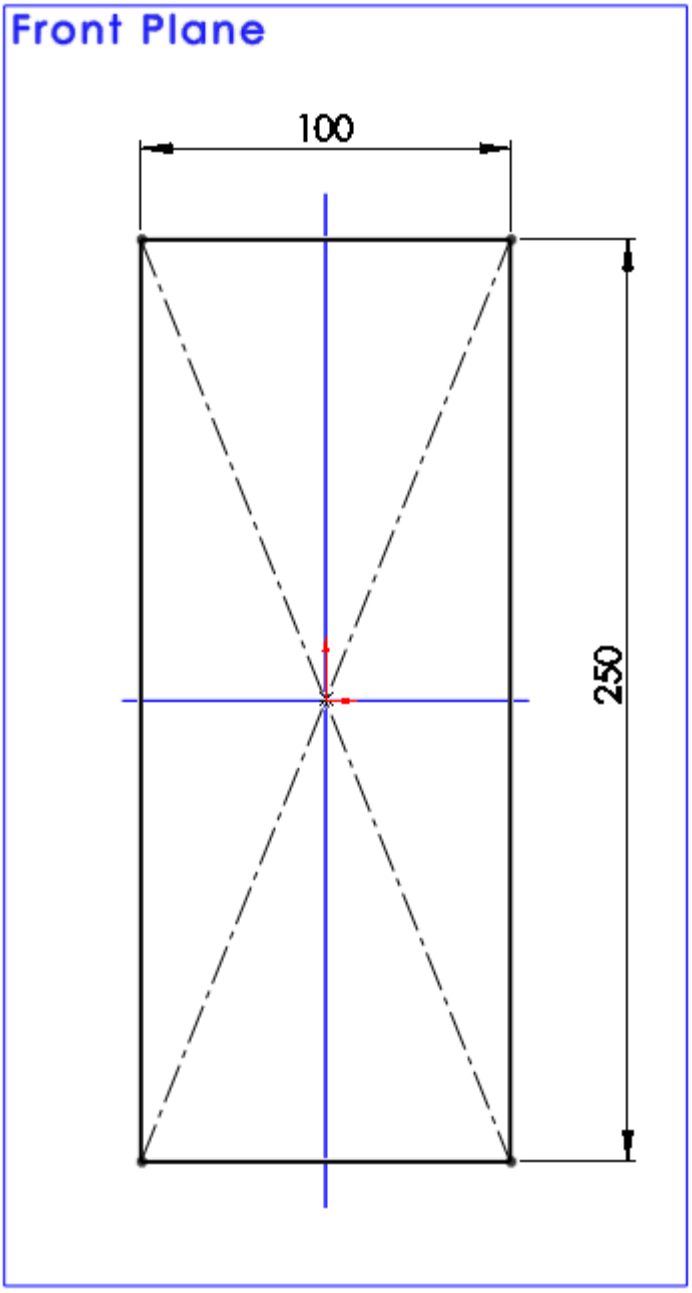


Draw a Center Rectangle

Go to **Tools > Sketch Entities > Center Rectangle** or click at the center rectangle icon

Create a sketch which starts at the Origin.

Change the dimensions of the rectangle into 100 and 250 mm by clicking at the dimension button



Create an Extruded Boss/Base

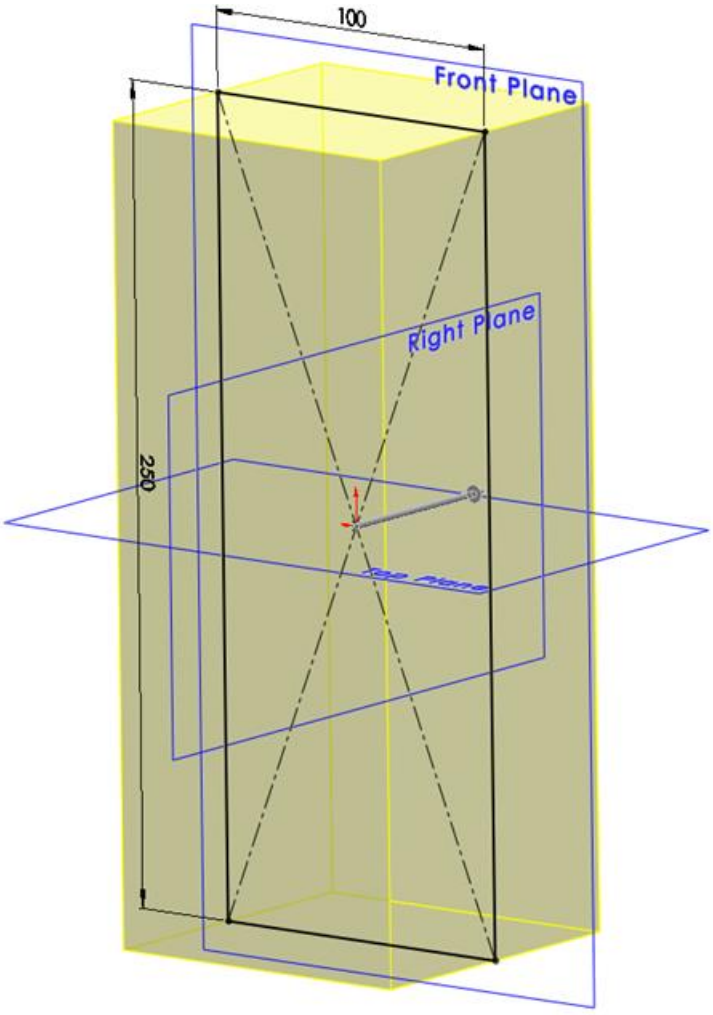
Go to: **Insert > Boss/Base > Extrude** or click at the Extrude icon

The **Boss-Extrude** menu appears

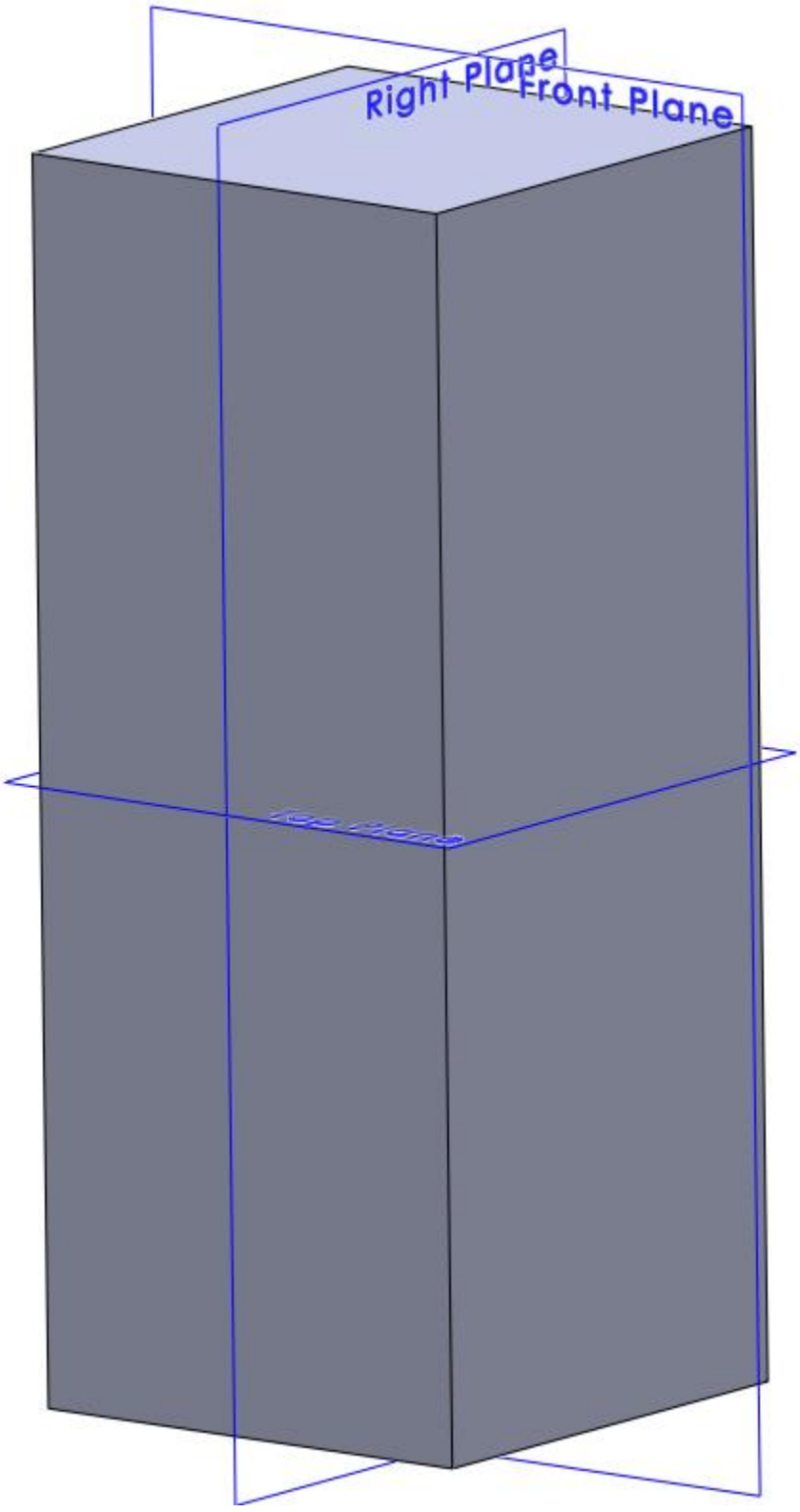
Direction 1 is set to **Midplane**

Change D1 into 100 mm

Click **OK**



Save the file with the following name: **Mold_Block.SLDPRT**



Create a new assembly

Go to **File > New > Assembly** or click at the Assembly icon 

Select the three planes in the feature tree and make them visible by clicking on the glasses. 

Insert the Mold_Block

Go to: **Insert > Component > Existing Part/Assembly** 

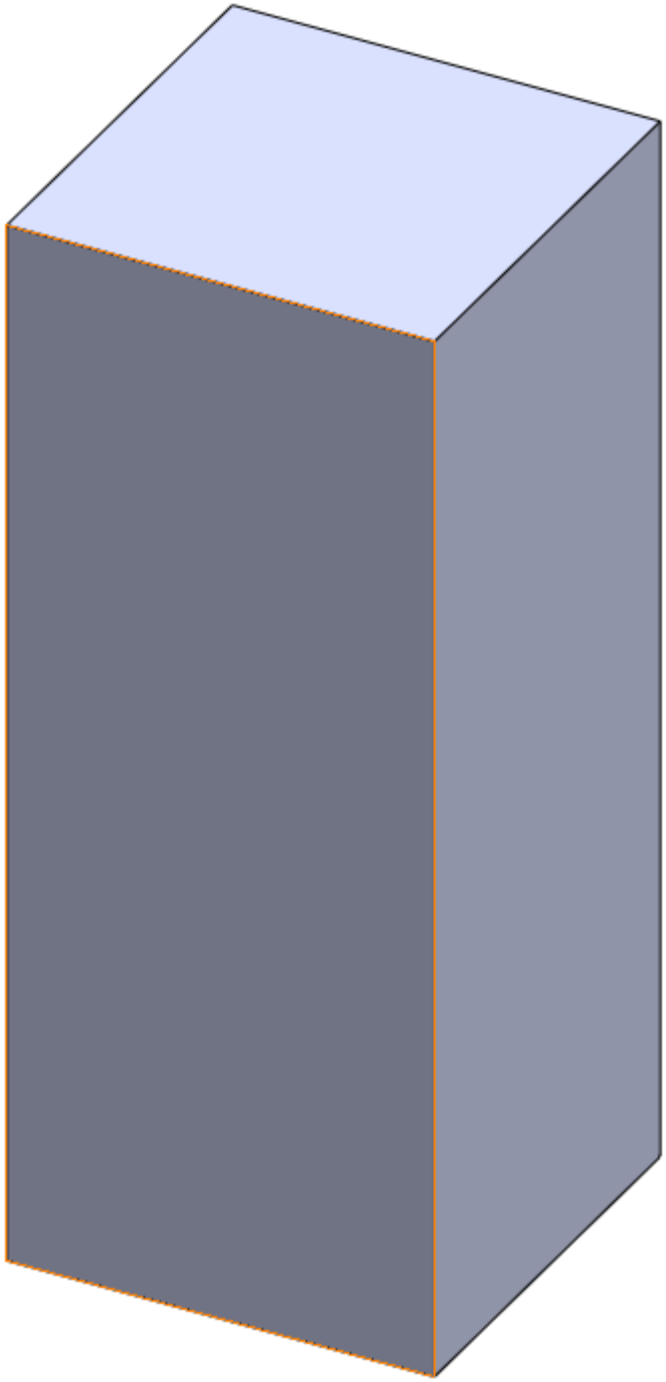
Click on the 

button.

Select the **Mold_Block.SLDPRT** in your working map.

Click OK 

The Mold_Block is fixed automatically in the assembly.



Insert the Coke bottle

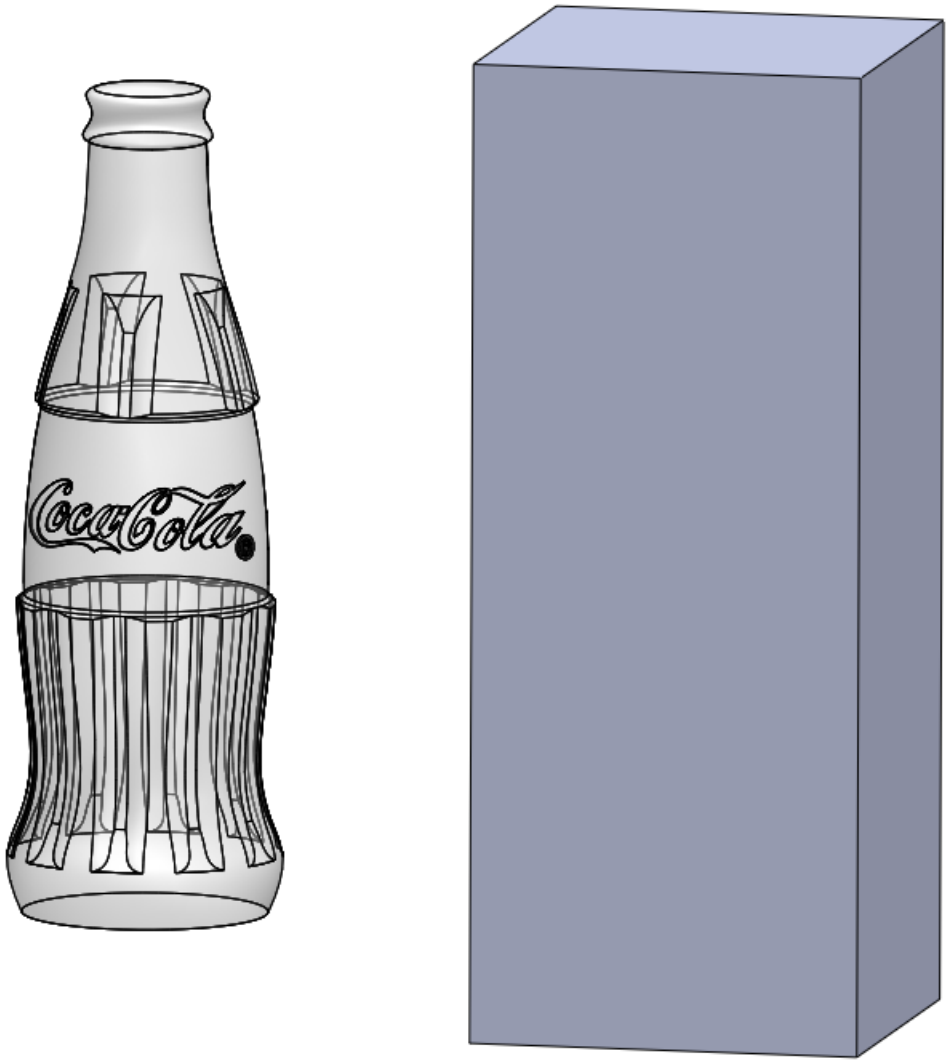
Go to: **Insert > Component > Existing Part/Assembly** 

Click on the 

button.

Select the **Bottle.SLDPRT** in your working map.

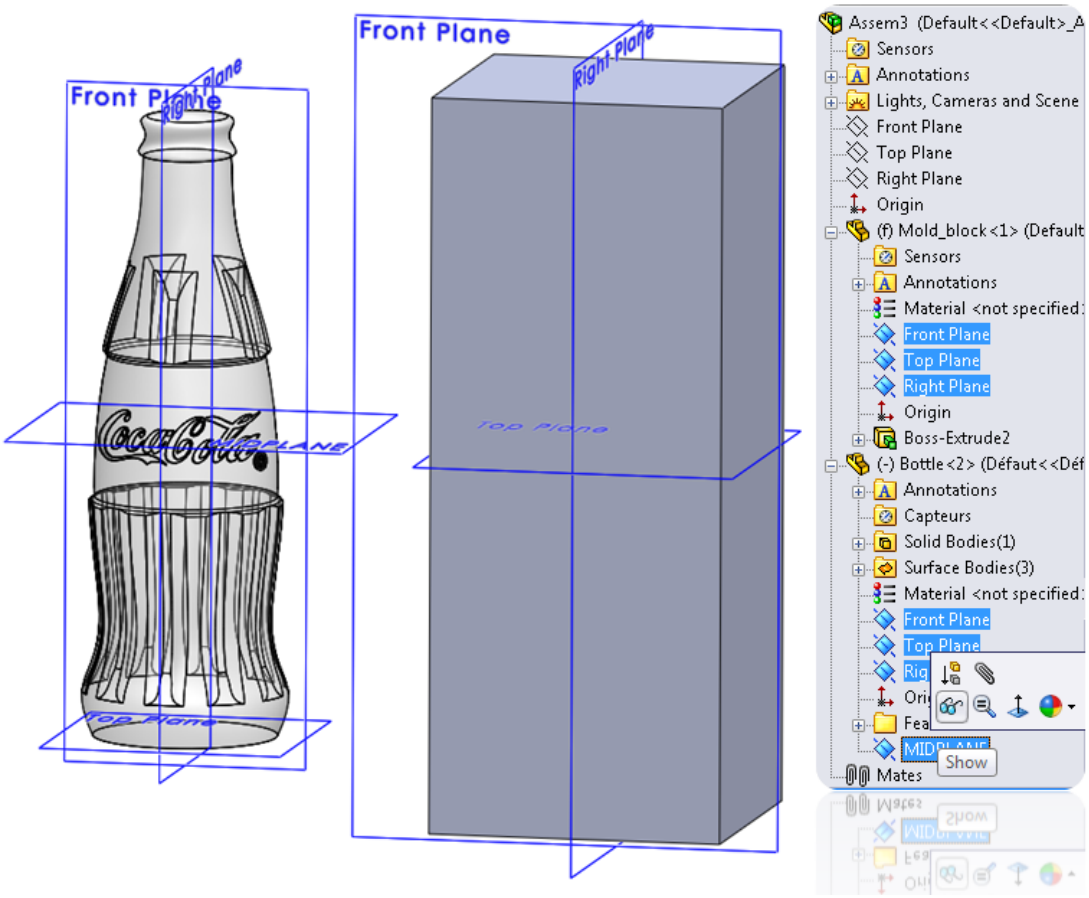
Place the model of the Bottle next to the Mold_Block as shown in the picture.



Unhide the Planes

Go to **View > Planes** or click at the Planes icon 

If the planes keep unvisible click at the Glasses in the Feature Tree to Unhide the planes as shown in the picture



Place the Bottle inside the Mold_Block

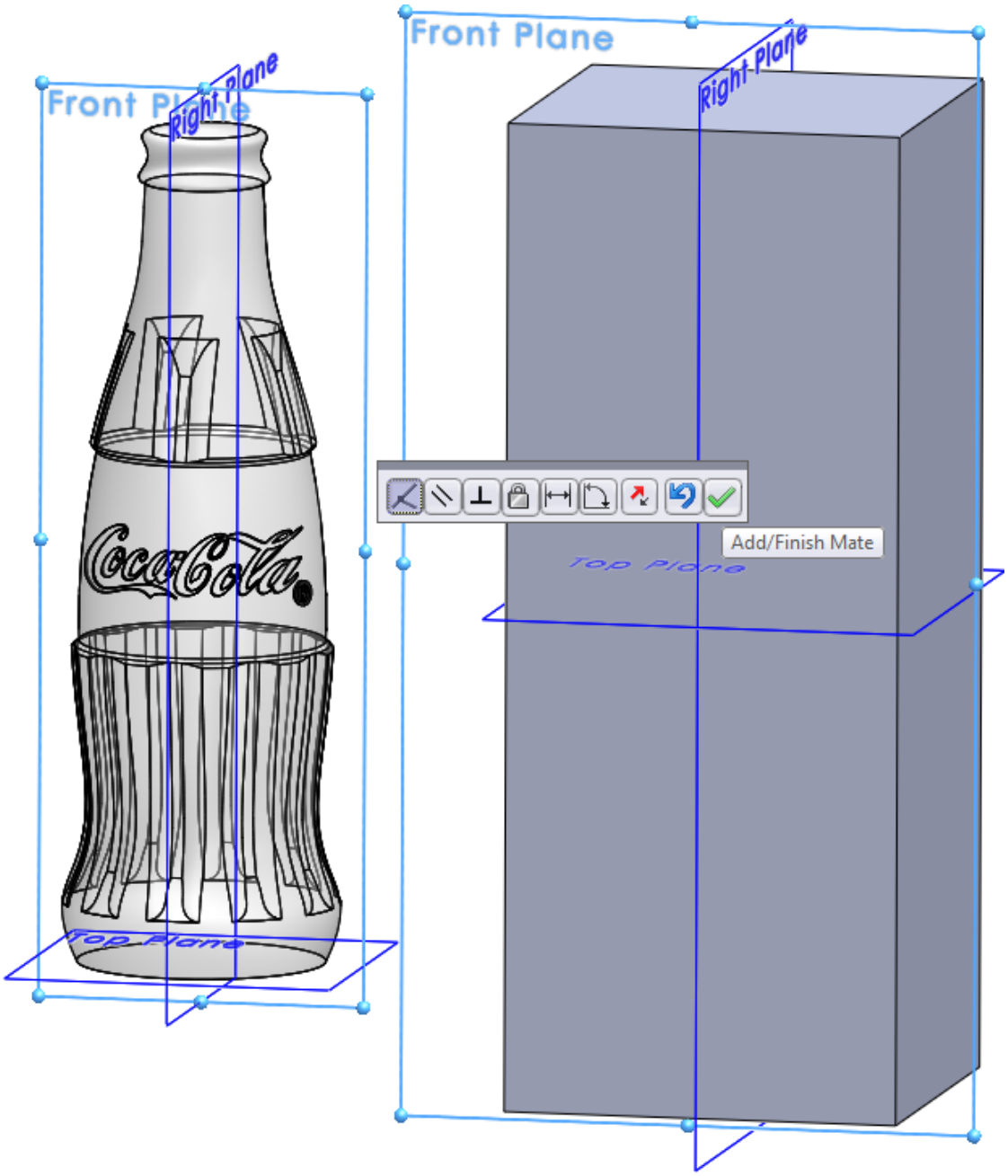
Create a coincident mate

Go to: **Insert > Mate**

Select the coincident option.

Select the **Front Plane** of the Mold Block and the **Front Plane** of the Bottle.

Click OK



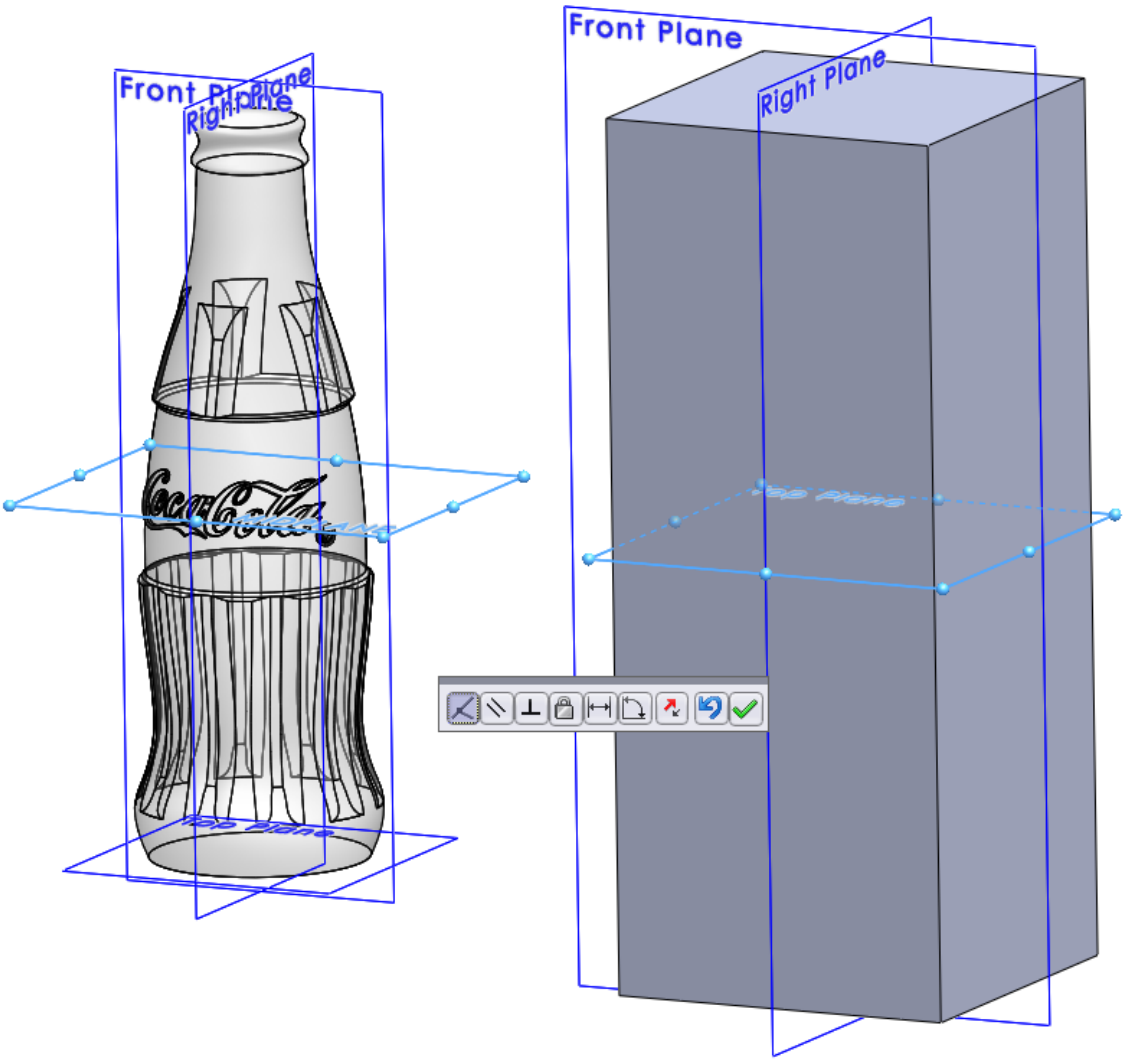
Create another coincident mate

Go to: **Insert > Mate**

Select the coincident option.

Select the **Top Plane** of the Mold Block and the **MIDPLANE** of the Bottle.

Click OK



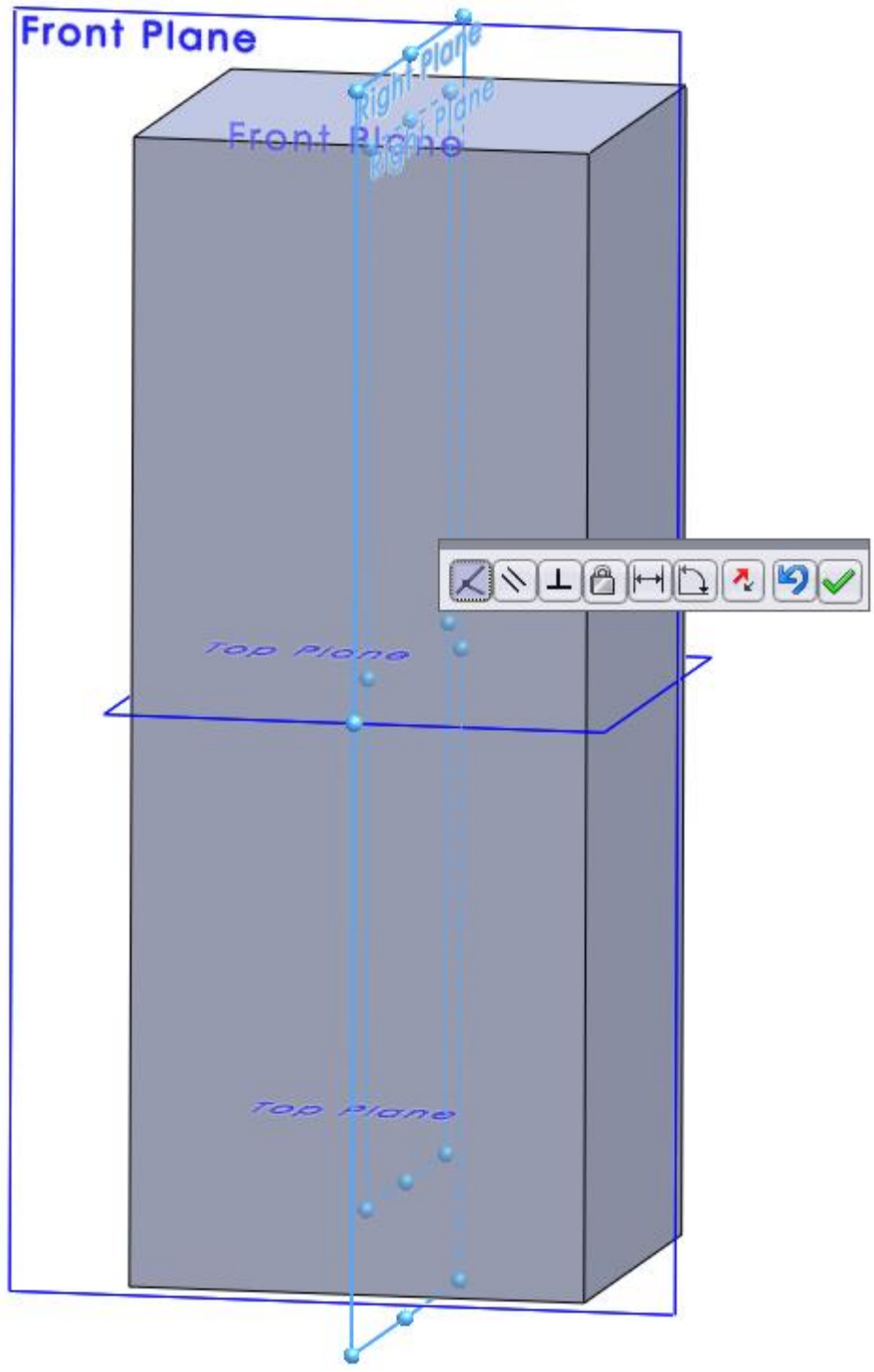
Create another coincident mate

Go to: **Insert > Mate**

Select the coincident option.

Select the **Right Plane** of the Mold Block and the **Right Plane** of the Bottle.

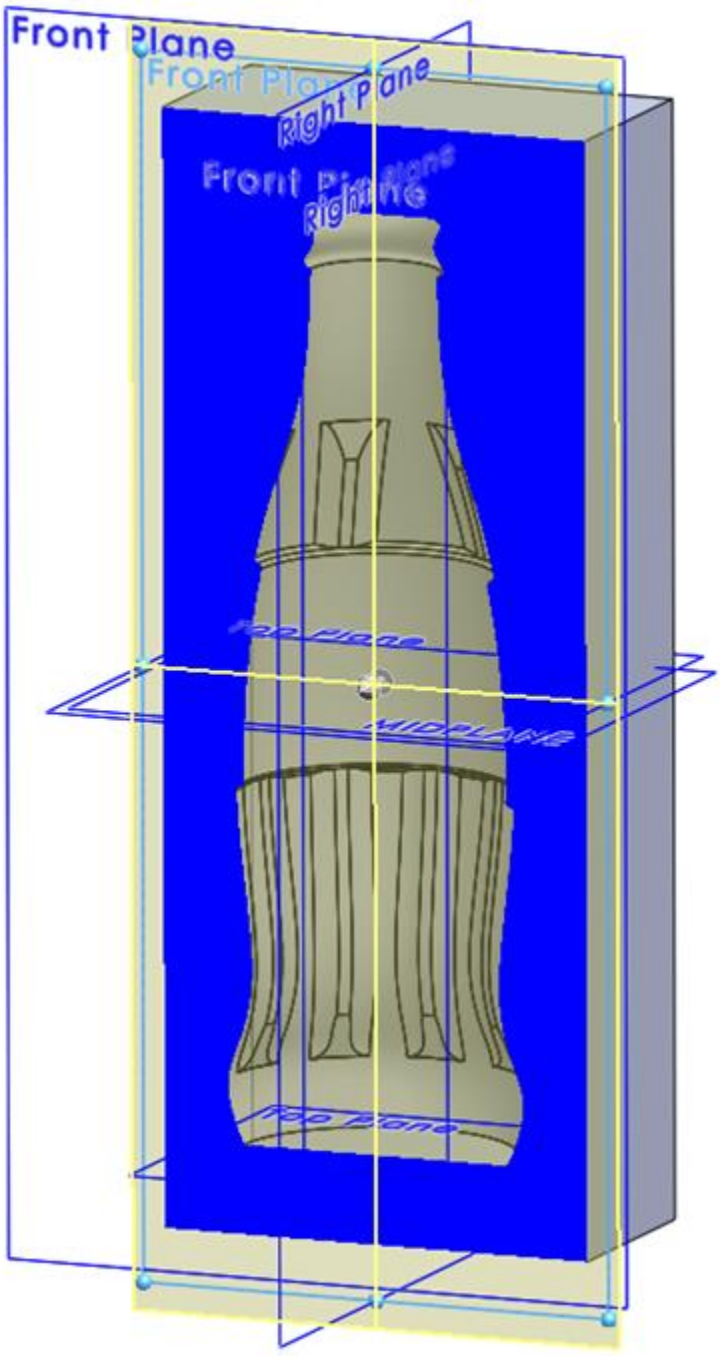
Click OK

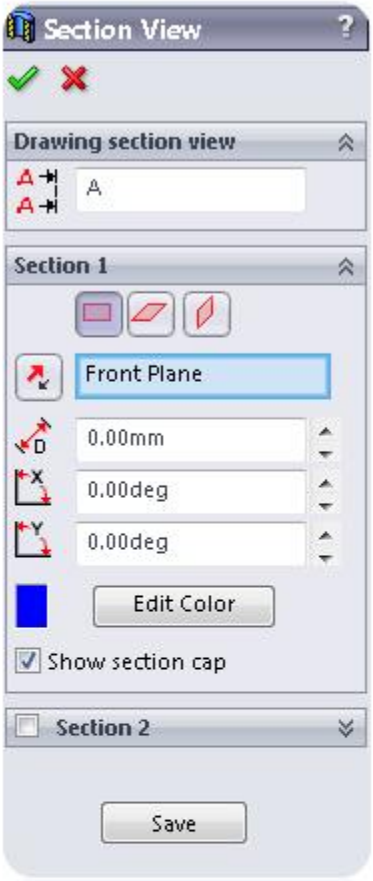


The two models are now fully defined.

Use the Section View to check if the Bottle is inside the Mold Block.

Go to **View > Display > Section View** or click at the Section View button



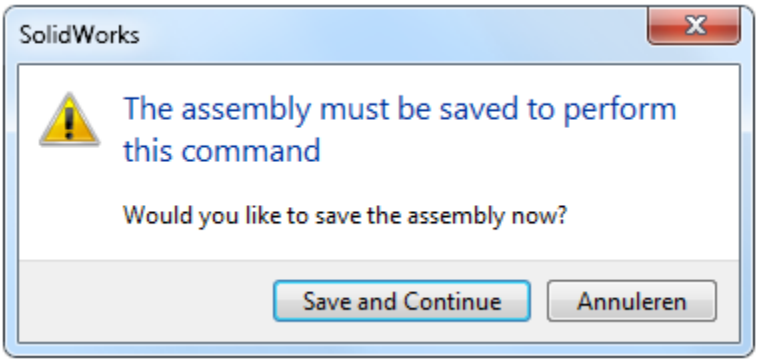


Now it’s time to subtract the model of the bottle from the Mold Block.

Click at the Mold_Block and click at the **Edit Part** icon as shown in the picture

You'll get the Following warning: **"The assembly must be saved to perform this command"**

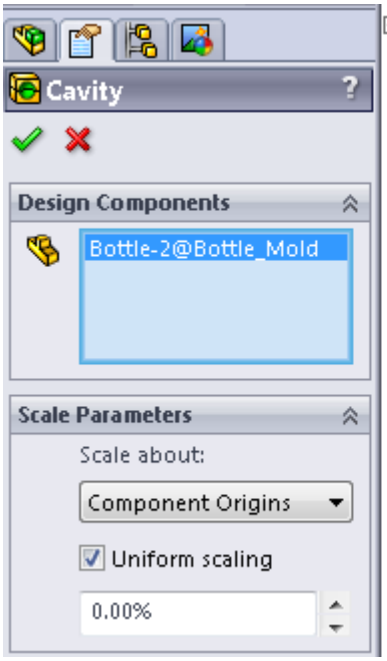
Click at **Save and Continue** to Save the assembly.

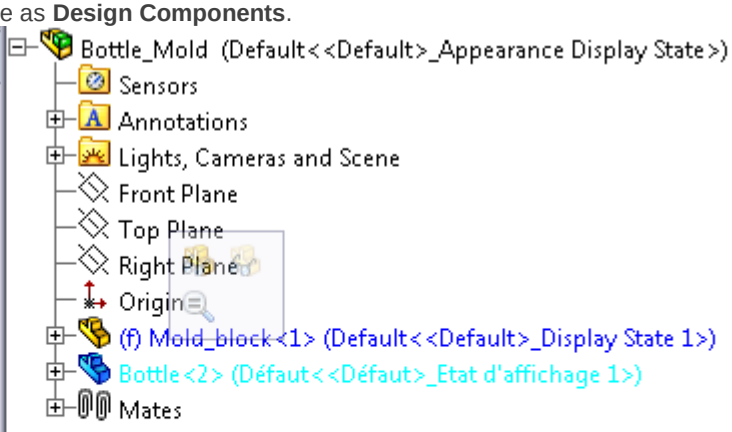


I will show you how the **Cavity feature** works.

Go to: **Insert > Features > Cavity** or click at the Cavity icon 

Select the Bottle in the Feature Tree as **Design Components**.





Click OK 

Click at the **Edit Part button** in the upper right corner to close the Part mode. 

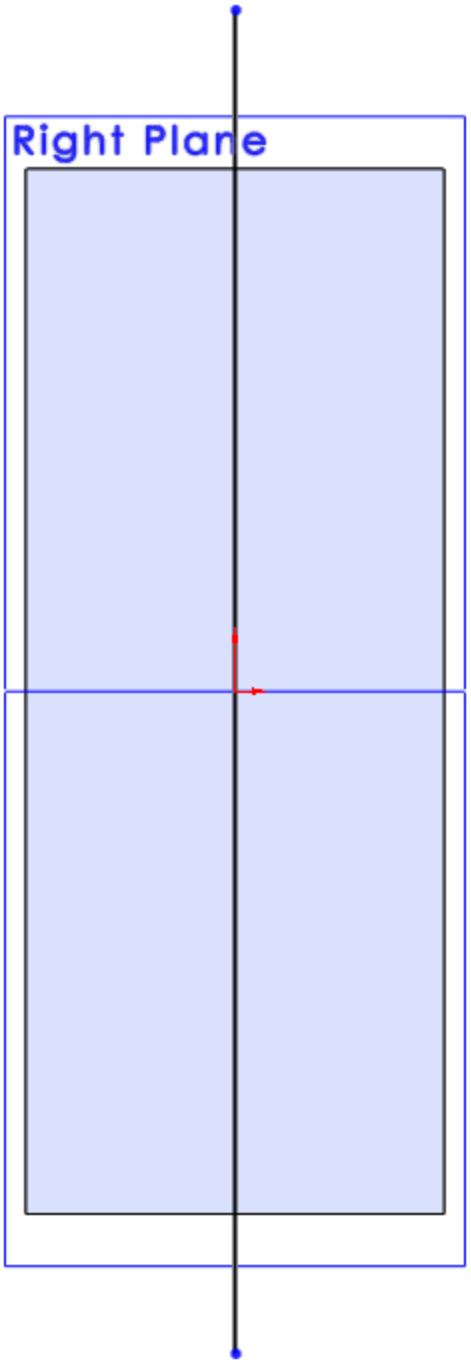
Now I will show you how to split the Mold into two parts

Draw a Split line

Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw a vertical line as shown in the picture 

Click at the Sketch button in the upper right corner to close the 2D Sketch 

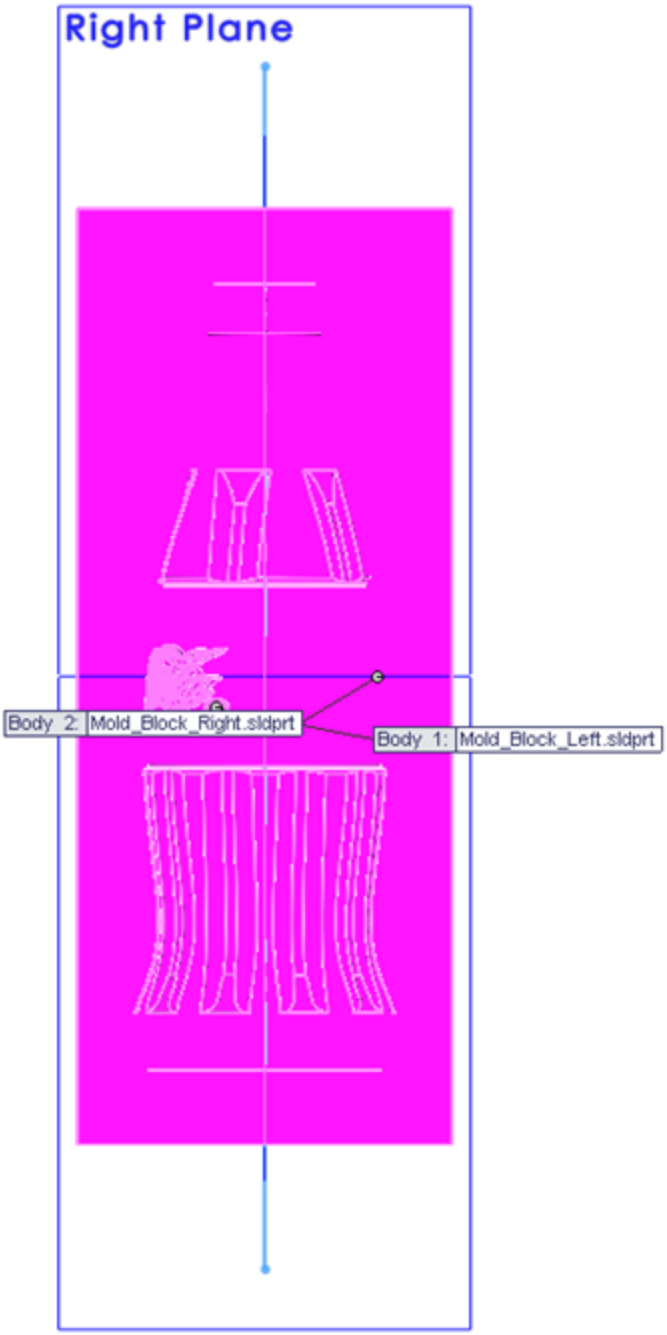


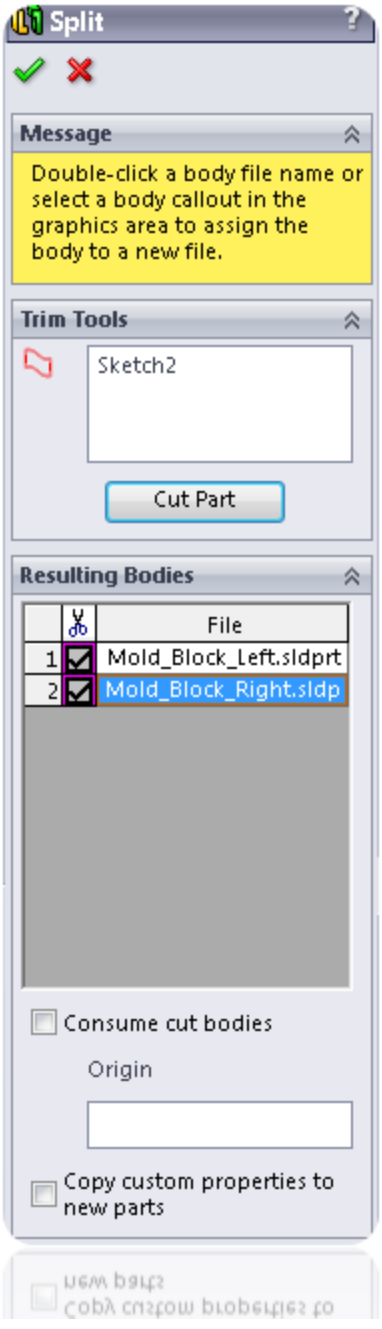
Go to **Insert > Features > Split** or click at the Split icon 

Select Sketch2 as **Trim Tool** and click at **Cut Part** 

Select **Part 1** and **Part 2** in the **Resulting Bodies** tab and change the names of the parts

Click OK 





You're done!

Now you know how to model a Mold in SolidWorks.

Hopefully you've learned something from it.

Leave a comment below and don't forget to Like or to Tweet this post to share it with other SolidWorkers. Google +1's are also very appreciated!