

Creo Tool Design – A Practical Approach

Creo 3.0



www.caeuniversity.com

Creo Tool Design – A Practical Approach

Creo 3.0

Copyright © 2018. All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the www.caeuniversity.com

CAE UNIVERSITY makes every effort to ensure the accuracy of the training material and assumes no responsibility or liability for errors or inaccuracies that may appear in this publication.

Contents

1. INTRODUCTION.....	3
EXERCISE 1.....	9
EXERCISE 2.....	59
PARTING SURFACES.....	75
1. <i>Manual Parting Surfaces</i>	77
EXERCISE 1.....	78
EXERCISE 2.....	89
EXERCISE 3.....	99
EXERCISE 4.....	113
EXERCISE 5.....	139
EXERCISE 6.....	151
EXERCISE 7.....	174
EXERCISE 8.....	191
EXERCISE 9.....	208
2. <i>Automatic Parting Surfaces</i>	241
EXERCISE 1.....	242
EXERCISE 2.....	253
EXERCISE 3.....	270
EXERCISE 4.....	275
EXERCISE 5.....	284
EXERCISE 6.....	303
EXERCISE 7.....	309
EXERCISE 8.....	330
3. SLIDERS AND LIFTERS.....	367
EXERCISE 1.....	368
EXERCISE 2.....	381
EXERCISE 3.....	393
EXERCISE 4.....	410
EXERCISE 5.....	436
EXERCISE 6.....	452
EXERCISE 7.....	467
4. MULTI-CAVITY MOLDS.....	485
EXERCISE 1.....	487
EXERCISE 2.....	498
EXERCISE 3.....	511
EXERCISE 4.....	528
EXERCISE 5.....	560
5. FEED SYSTEM.....	575
EXERCISE 1.....	576
EXERCISE 2.....	600
EXERCISE 3.....	618
EXERCISE 4.....	626
6. WATER LINE AND EJECTOR PINS.....	639
EXERCISE 1.....	640
7. MOLD ANALYSIS.....	647
<i>Water Line Clearance Check</i>	648

EXERCISE 1.....	648
<i>Mold Opening Analysis</i>	651
EXERCISE 1.....	652
<i>PARTING SURFACE CHECK</i>	656
<i>PROJECTED AREA CALCULATION</i>	657
8. MOLD LAYOUT AND EMX.....	659
<i>MOLD LAYOUT</i>	659
<i>EMX</i>	659
<i>MOLD LAYOUT VS. EMX</i>	660

Creo Tool Design

Creo is not only a design tool but can handle every aspect of product development from conceptual design to finished product. It includes complete family of manufacturing applications which are completely integrated with Creo. So from the conception of the product the user is working in the same environment and there is no need to translate data between part design, mold design and NC toolpath creation, saving significant cycle time. All the downstream data is associative to the design model and hence all the changes made to the design model are automatically updated.

Creo Tool Design is an optional module for Creo that provides the tools to simulate the mold design process.

This self-study course starts from the very basic concepts and teaches advanced techniques step by step. After completing these tutorials an Engineer or Designer will be able to create the advanced parting geometry, split the mold into core (male) and cavity (female) halves, create Sliders and Lifters, define multi-cavity molds, create runners, gates, sprue, cold slugs, create cooling channel and check their clearance from neighboring surfaces.

The training material is divided into sections. Each section is accompanied with exercises to practice the concepts learned. The training material consists of following sections.

1. Introduction
2. Parting Surfaces
 - a. Manual Parting Surfaces
 - b. Automatic Parting Surfaces
3. Sliders and Lifters
4. Multi-Cavity Molds
5. Feed System and Molding
6. Water Line and Ejector Pins
7. Mold Analysis
8. Mold Layout And EMX

1. INTRODUCTION

Creo Tool Design provides the functionality to build parting lines and parting surfaces and extract core, cavities and slides. All the components extracted are associative to the design model and hence automatically update to the changes.

The capabilities of this module can be summarized as

- Analyze if a design part is moldable, using Draft Check and Thickness Check capabilities
- Fix problem areas by creating draft, rounds, and other features as needed
- Apply a shrinkage that corresponds to design part material, geometry, and molding conditions
- Automatically create the workpiece stock from which core, cavity, and inserts will be split
- Create parting geometry, including sliders, inserts, automatic parting lines, and automatic parting surfaces
- Split the workpiece to create core, cavity, and inserts as solid models
- Add gates, runners, and waterlines as mold features
- Check for interference of parts during mold opening

A Creo Tool Design session begins with the creation of a manufacturing model. All manufacturing information is stored in the .asm file

The manufacturing model requires a Creo part model called a **Reference Model** and usually represents a part that should be molded. Reference Model uses the geometry of Design Model as the source.

Design Model usually represents the product designer's vision of his final product. Typically, the design model geometry contains all necessary design elements that are required for the functioning product, but does not contain the elements that molding or casting technology requires e.g. shrinkage, draft or fillets. So usually the Reference part and Design part are not identical.

There are three ways to create a Reference Model.

1. Inherited

The reference part inherits all geometry and feature information from the design part. You can modify the dimensions of all the features that are inherited from the Design model. All the changes made in the design part are updated in the Reference model but the features added or modified in the reference model have no effect on Design model. So Inherited model has one-directional associativity to the Design model.

2. Merge by Reference

Creo copies design part geometry into the reference part. In this case, only the geometry and layers are copied from the design part and there is no feature information in the Reference part. All the changes made in the design part are updated in the Reference model but the features added in the reference model have no effect on Design model. So in this case Reference Model has one-directional associativity to the Design model.

3. Same Model

Creo uses the selected design part as the mold reference part.

The next step is the addition of **shrinkage** to the reference model. When the molten material injected into the mold cavity solidifies, it shrinks by a certain amount that is a characteristic of each material. The shrinkage feature enlarges the reference model so that after solidification and cooling, the final dimensions of the part are as desired. In Creo the shrinkage is specified either By Dimension or By Scaling

By Dimension—Allows to set up one shrink coefficient for all model dimensions, and specify shrink coefficients for individual dimensions.

By Scaling—Allows to shrink the part geometry by scaling it with respect to a coordinate system. If it is applied in Mold mode, it applies only to the reference model and does not affect the design model.

In either case, shrinkage is computed by one of the two formulas, $1+S$ or $1/(1-S)$ where S is the shrinkage ratio. Before specifying shrinkage method the desired formula should be selected. The default formula is $1+S$

The next step is the creation of **workpiece model**. The workpiece represents the material from which the mold inserts are created. It is the overall volume of the mold components that directly participate in shaping the molten material. The workpiece can have standard overall dimensions to fit in the standard base, or it can be custom-made to accommodate the geometry of the design model. If the workpiece is a pre-existing part, it can be assembled to the manufacturing assembly. Creo Tool Design also provides the tools to create the workpiece automatically or manually within the assembly mode.

If the workpiece is created in the mold assembly, it automatically uses the same accuracy as the reference model. But for this automatic operation to work, the configuration file option `enable_absolute_accuracy` must be set to Yes.

The next step is usually the creation of **parting surfaces** that are used to split the workpiece into volumes. Parting surfaces can be created manually using the basic Creo surfacing techniques or can be created automatically with special tools available in manufacturing mode.

After the parting surfaces have been defined, workpiece can be **split** into volumes. The mold volumes are closed quilts which can be later used to extract components. When a split operation is performed to the workpiece, system creates a closed quilt by copying all the surfaces of the workpiece, subtracts the reference part out of this volume then calculates the mold volumes to either side of the parting surface.

Solid parts can be **extracted** out of mold volumes. Extracted parts are produced by filling previously defined mold volumes with solid material. This process, performed automatically, is called extracting. These parts can be used for generating the MCD data in Pro/NC for driving the CNC machine tools.

Importance of Accuracy in Creo Tool Design

Accuracy determines the smallest distance between two entities. These entities may be points, edges or surfaces. So lesser the value of accuracy more detailed items can be represented in Creo. We can also say that accuracy is the smallest size (measured in the current units) that Creo can recognize.

But as we decrease the value of accuracy (i.e. increase the accuracy) regeneration time of the model increases. It also results in increased memory usage and file size.

There are two methods of specifying accuracy in Creo

1. Relative accuracy
2. Absolute accuracy

Relative accuracy relates the level of detail in a model to its overall size. It is based on the assumption that the smaller models require more detailed work and vice versa. When the accuracy is set to relative, system determines the accuracy value fore each step of a part's creation. As the part's size is increased it will calculate a new value for accuracy and uses it from that point on for all geometry related calculation. So the features created at start and at end of the model usually have different accuracy. For example we create feature No. 30 as a surface copy of feature No.5. These two surfaces will not be geometrically equal because both features were created at different accuracy.

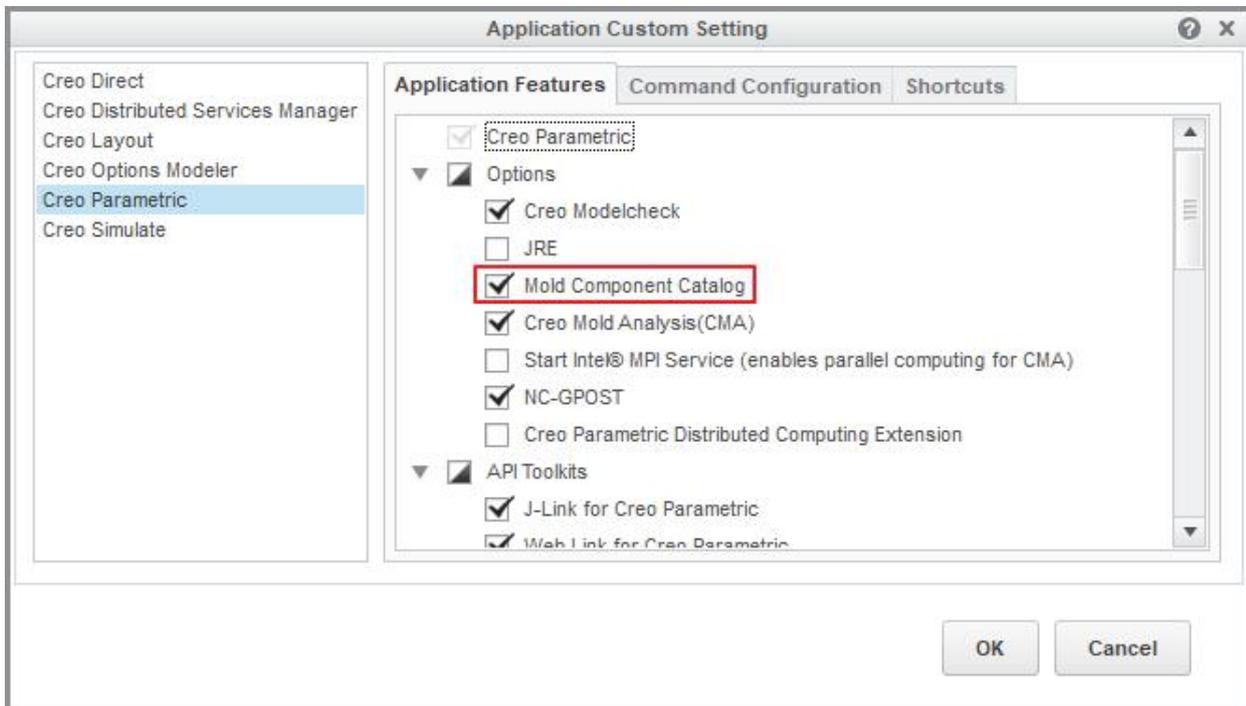
When a part's accuracy is set to **absolute accuracy** then its limits or size has no effect on the parts accuracy and therefore all new features are created at the same accuracy level. The value specified for the absolute accuracy is used as the actual accuracy for all the features of the model.

In Creo Tool Design accuracy becomes more important because we work in assembly environment with different models having different accuracies. The reference model is usually set to higher accuracy (i.e. lower accuracy value) to represent detail work. But the manufacturing model is usually set to lower relative accuracy. All the manufacturing geometry is created at the assembly level so it is the assembly feature and hence manufacturing model accuracy is used to create that geometry. Now suppose that we trim a volume to the reference model. The accuracy of the manufacturing assembly may not be sufficient to represent that geometry which has been created in reference model at a much higher accuracy. So to avoid problems, we should set the accuracy of manufacturing assembly to absolute accuracy and equal to that of reference model.

It is always recommended to use absolute accuracy in manufacturing mode. Creo Tool Design automatically sets the accuracy of manufacturing assembly equal to that of reference model if the configuration file option `enable_absolute_accuracy` is set yes. In Tool Design, setting this option to yes helps to maintain uniform accuracy for the reference model, the workpiece, and the mold assembly. At the time we add the reference model to the mold assembly, the system will inform if a discrepancy exists between the assembly model accuracy and the reference model accuracy. We can then accept or reject setting the assembly model accuracy to equal the reference model accuracy. If we create workpiece in Mold mode, its accuracy is automatically the same as the accuracy of the assembly model.

Before Starting the Creo Tool Design

1. During the installation of Creo you must make sure that Mold Component Catalog is installed.

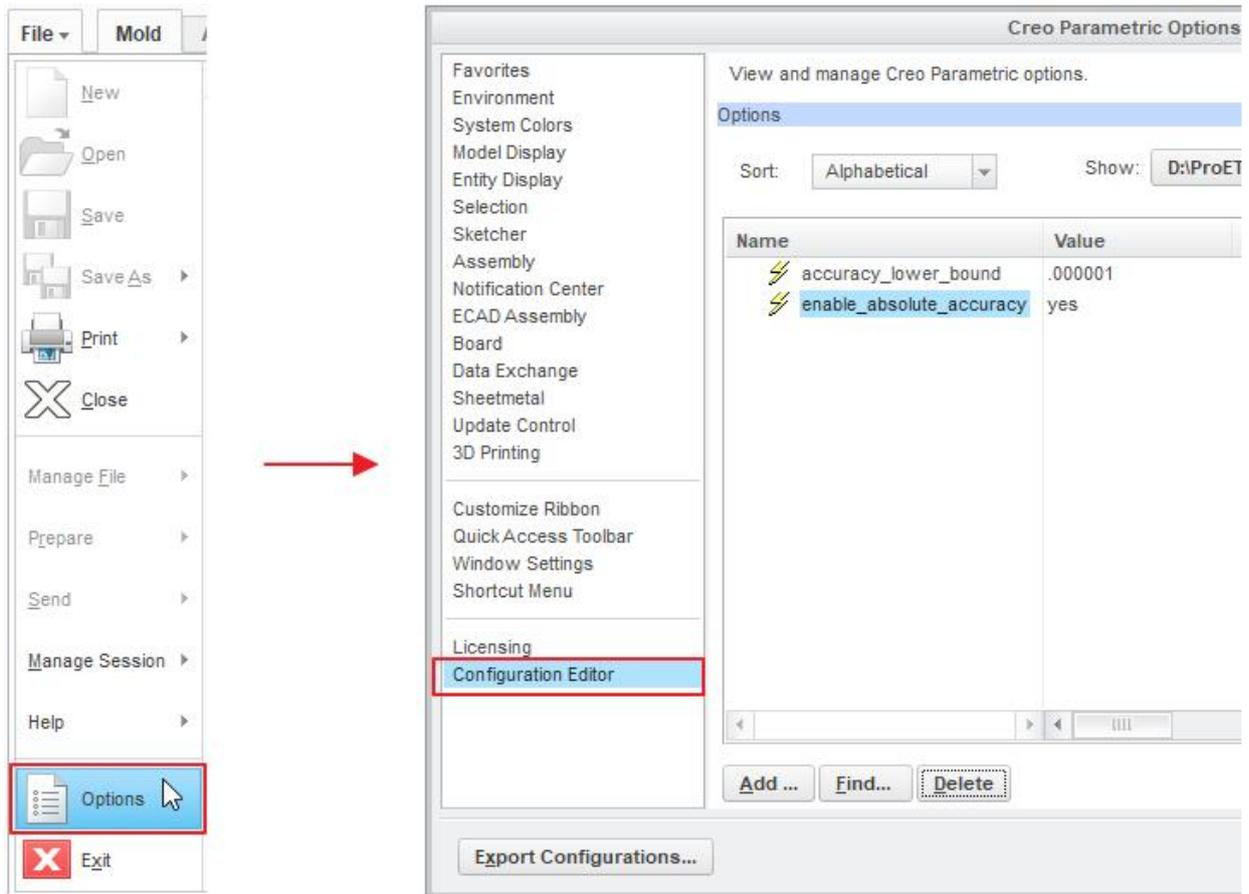


If it is not installed then you must run the setup and add the Mold Component Catalog to the list of installed components.

2. Before starting any session of Creo Tool Design, it is recommended to set the following Configuration file options

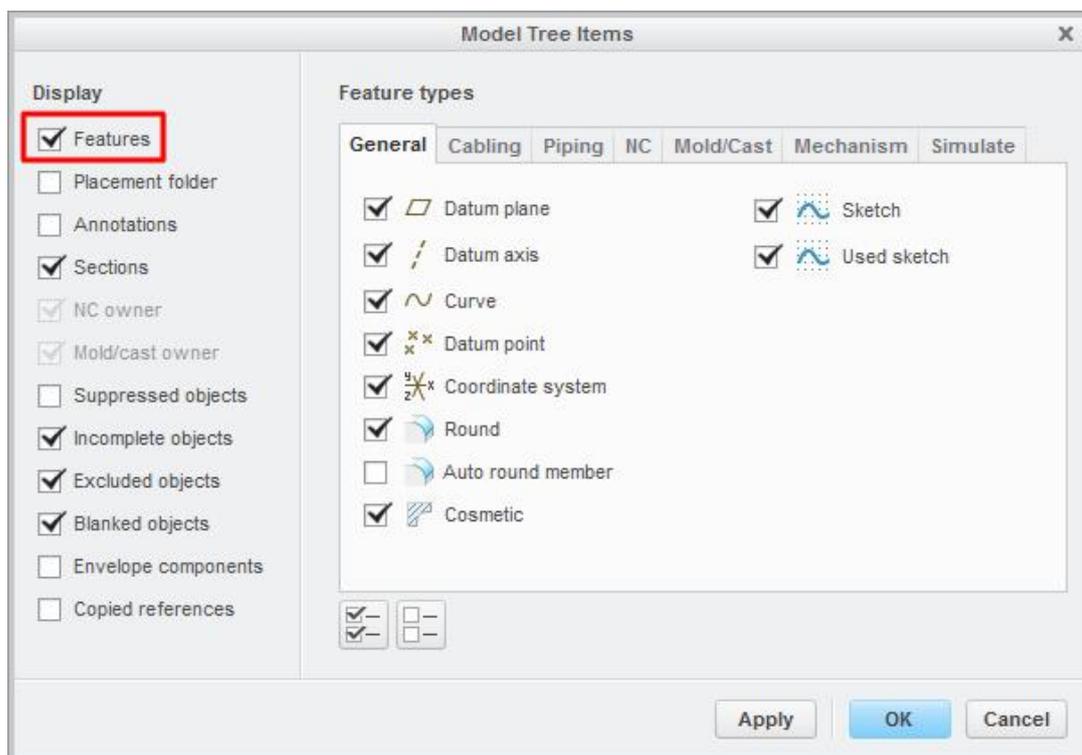
enable_absolute_accuracy	Yes
accuracy_lower_bound	0.000001

These option can be set by picking **File > Options > Configuration Editor** as shown below.

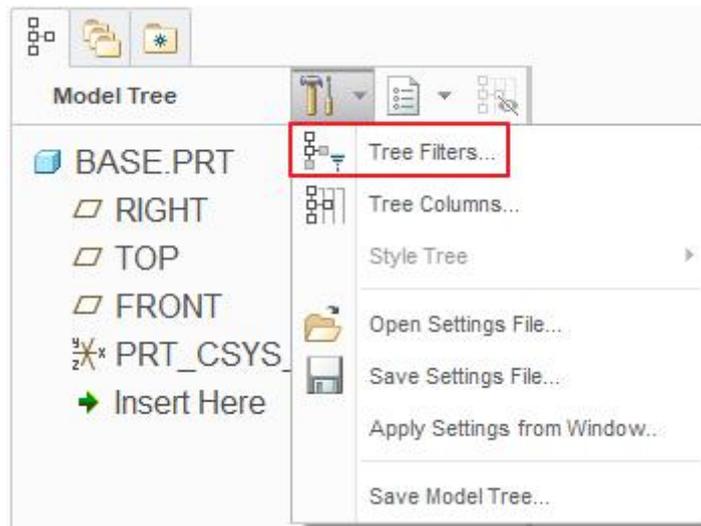


accuracy_lower_bound value is used to override the default lower accuracy limit of 0.0001

- To display the features in the Model Tree, make sure that Features option is checked in Model Tree Items dialog box.



This dialog box can be accessed at



Displaying the features in the Model Tree is necessary as most of the work will consist of creating features.

Note: In some exercises, the background and geometry are shown in different colors compared to the default system colors in Creo 3.0 for clarity.

EXERCISE 1

In this exercise we will go through the basic steps of setting up a manufacturing assembly for the topcover.prt model.

Topics covered in this exercise are as follows

- ❖ Creating a new manufacturing model
- ❖ Placing the reference model
- ❖ Analyzing the part for sufficient draft
- ❖ Analyzing the part for uniform thickness
- ❖ Adding shrinkage to reference model
- ❖ Creating the Automatic Workpiece
- ❖ Defining the style state for Workpiece
- ❖ Creating the Parting Surface.
- ❖ Splitting the workpiece into two volumes
- ❖ Extracting components out of split volumes
- ❖ Uncluttering the view
- ❖ Splitting off the insert volume
- ❖ Splitting the insert volume into two

Set the working directory to the TOPCOVER folder and open the model topcover.prt. It will appear as shown below.



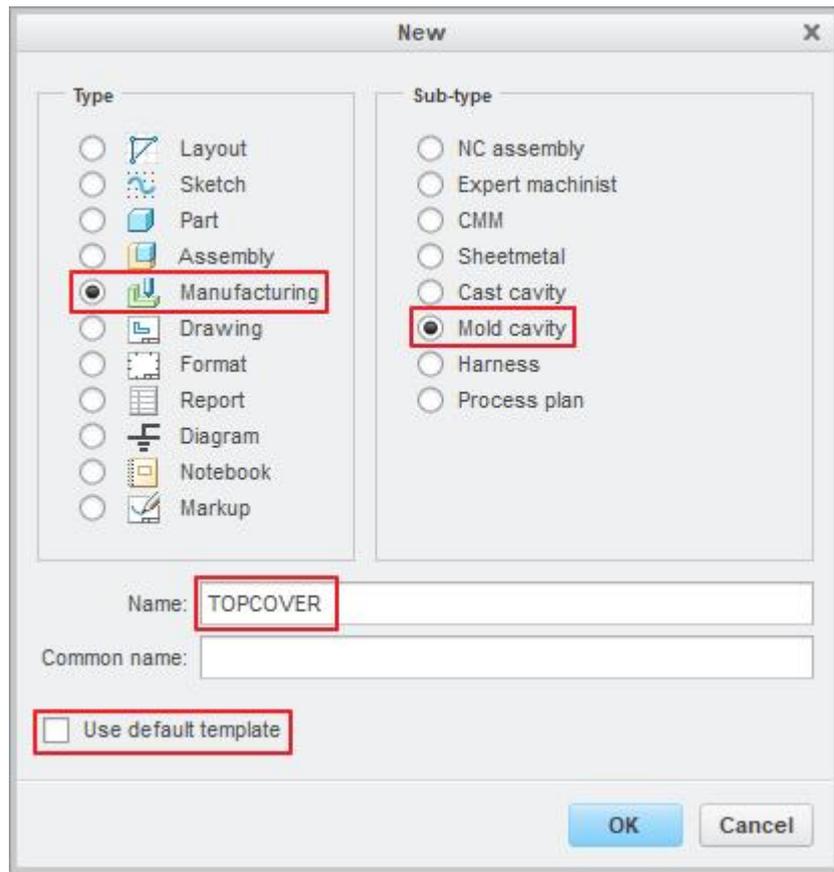
This part will be used as the design part for the manufacturing model. Notice that it has predefined coordinate system which will be used to assemble to the manufacturing assembly.

⇒ Creating a new Manufacturing Model

To define a new manufacturing model select **File > New** and New dialog box will appear.

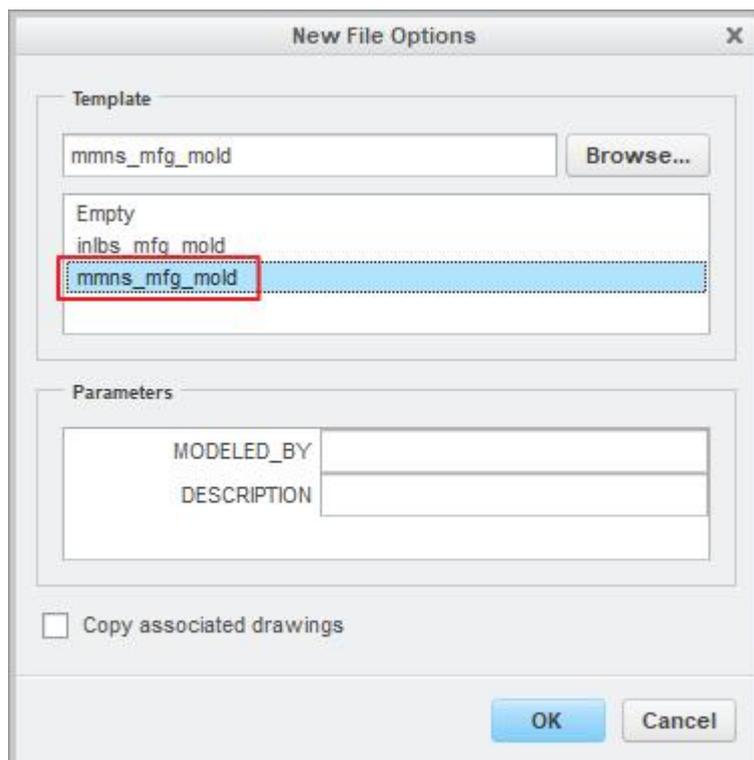
In the New dialog box select **Manufacturing** type and **Mold Cavity** in the Sub-type menu.

Enter the name **TOPCOVER** and uncheck the **Use default template** option.



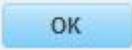
Pick **OK** to proceed.

In the New File Options dialog box select **mmns_mfg_mold** as the template

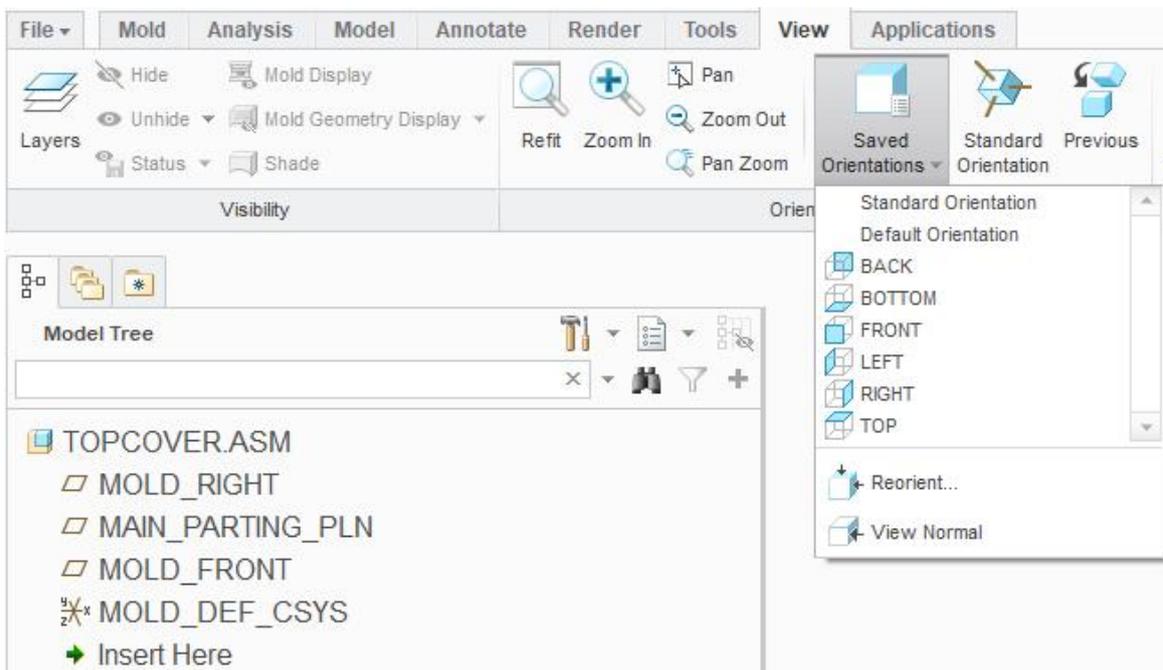


mmns_mfg_mold is the system supplied metric template for Mold Cavities.

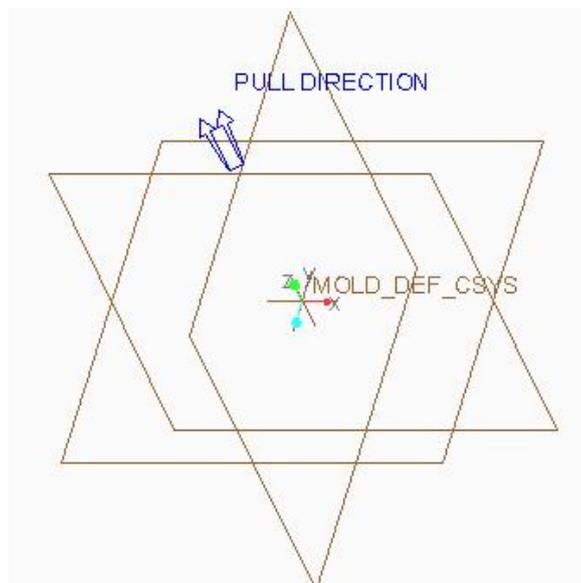
A template file contains all the necessary setting which a user performs routinely while defining a manufacturing model. Since many routine tasks are predefined, manufacturing models are created faster and productivity is greatly enhanced.

Pick  to create the manufacturing model. System will create topcover.asm that will contain all the mold information. Typically, the mold assembly consists of the reference model, workpiece, parting surfaces, mold volumes, extracted components and Mold specific and general Creo features.

Notice that system automatically creates the default datums and views as shown below.



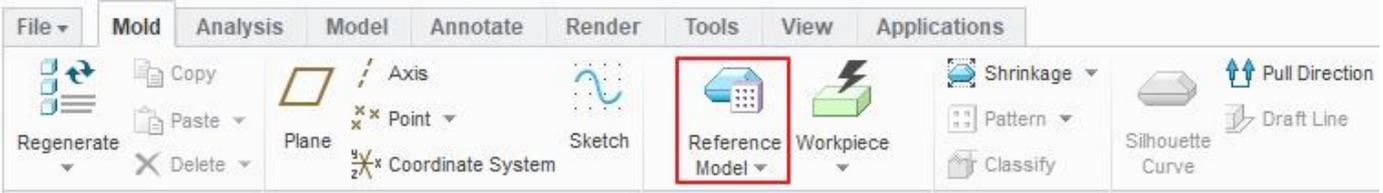
If you look in the graphics window you will notice that there is a PULL DIRECTION arrow pointing upward. This indicates a default direction for the mold opening; this default direction is used by other features like Silhouette Curve (parting line).



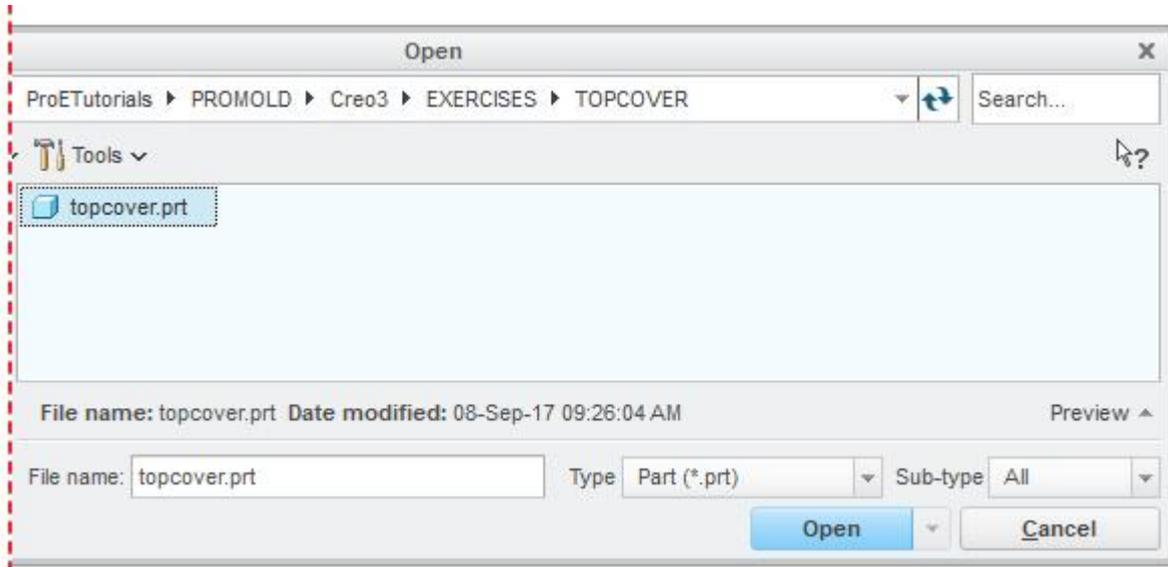
➔ **Placing the Reference Model**

Now we will place the reference part in the mold assembly.

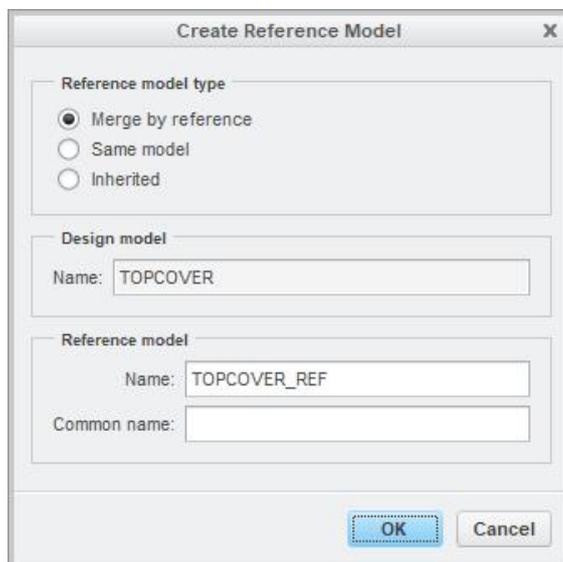
To place the topcover.prt, pick  icon located on Mold tab.



Select the **topcover.prt** in the Open dialog box then pick  .

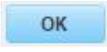


Now Create Reference Model dialog box will appear as shown below.

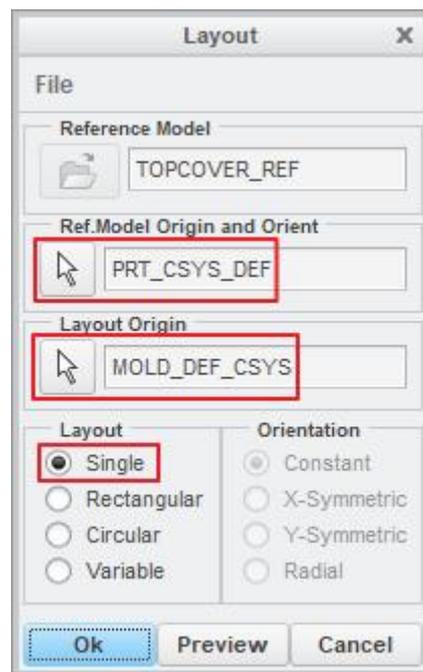


Notice that “Merge By Reference” is selected by default and system has given the TOPCOVER_REF name to the new Reference Model.

Merge By Reference option copies design part geometry into the reference part. All the changes made in the design part are updated in the Reference model but the features added in the reference model have no effect on Design model. So in this case Reference Model has one-directional associativity to the Design model.

For this exercise we will use the default options so pick  to proceed.

Now the Layout dialog box will appear as shown below. This dialog box is used to place and populate a reference model within a mold assembly.

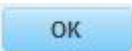


Notice that system has automatically selected the coordinate systems of the Reference model and Mold model. These coordinate systems are used to place the reference part within the mold assembly.

The reference part coordinate system specifies the orientation of the reference part within the mold assembly. By default, system selects the first coordinate system in the reference part.

You can place a reference part either once or position it multiple times within a mold assembly by accessing the Layout rules present in the Layout dialog box.

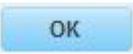
By default **Single** option is selected for Layout which means that only single instance of Reference model will be placed by mating its coordinate system with that of mold assembly.

Pick  as we want to place the reference part once.

Now the system will warn to change the absolute accuracy of the manufacturing model by displaying the following message.



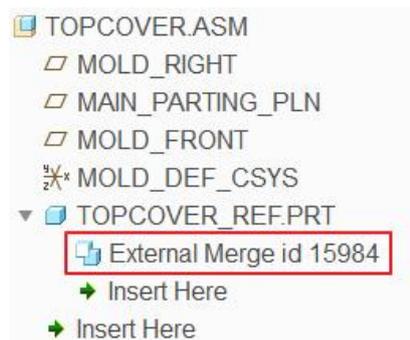
This is to change the absolute accuracy of the manufacturing model equal to that of reference model.

Pick  to accept change of absolute accuracy.

The placed model will appear as shown below.



If you look in the model tree, you will notice that Reference part consists of only a single feature as shown below.



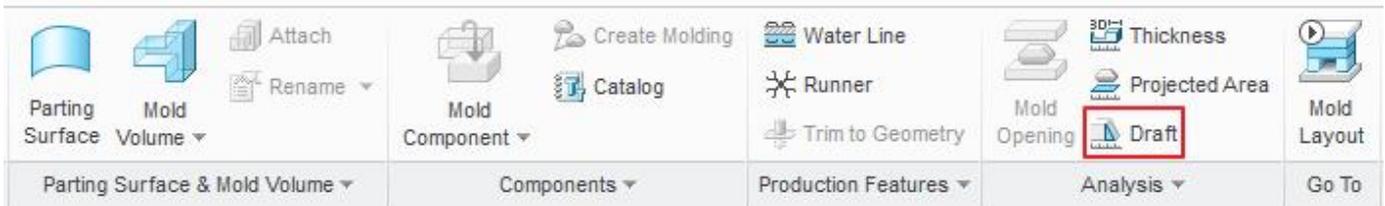
This is the Merge feature that copies the geometry of the design model into the reference model.

In this example, TOPCOVER.PRT is the design model and TOPCOVER_REF.PRT is the reference model.

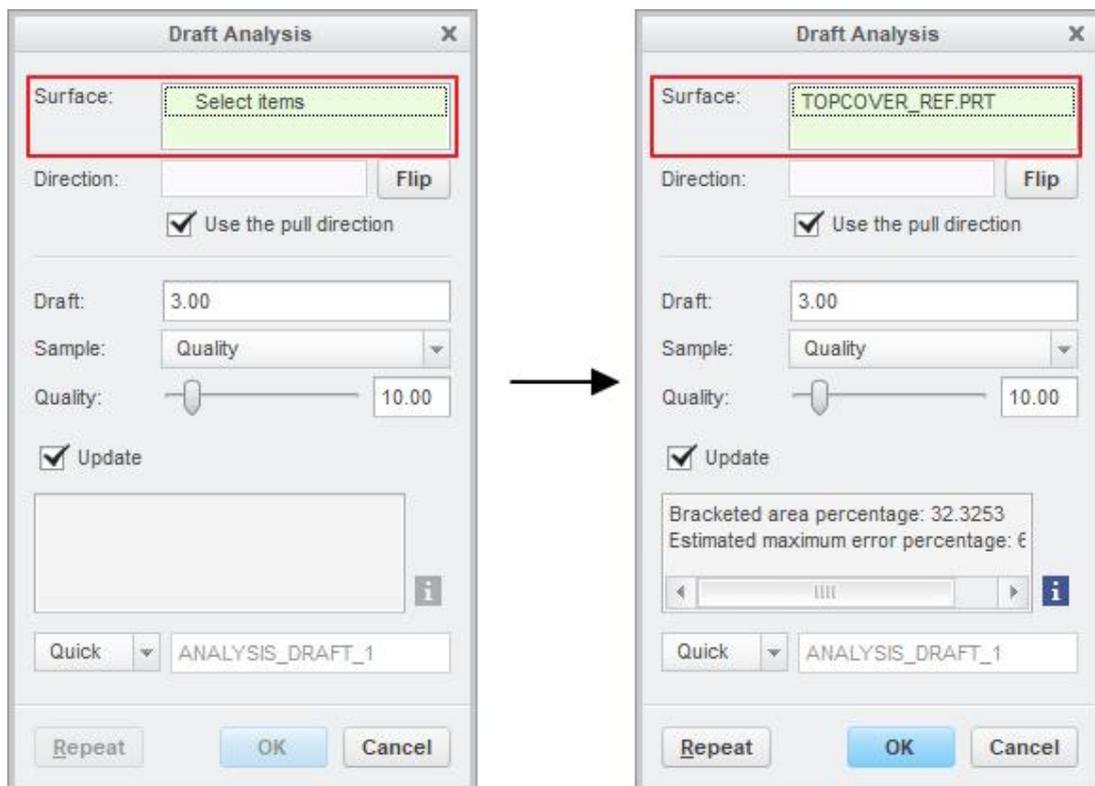
 **Analyzing the part for sufficient Draft**

Now we will analyze the reference part for sufficient draft. The draft is very important for molded products as it facilitates the ejection process.

Pick  Draft icon located on Mold tab.



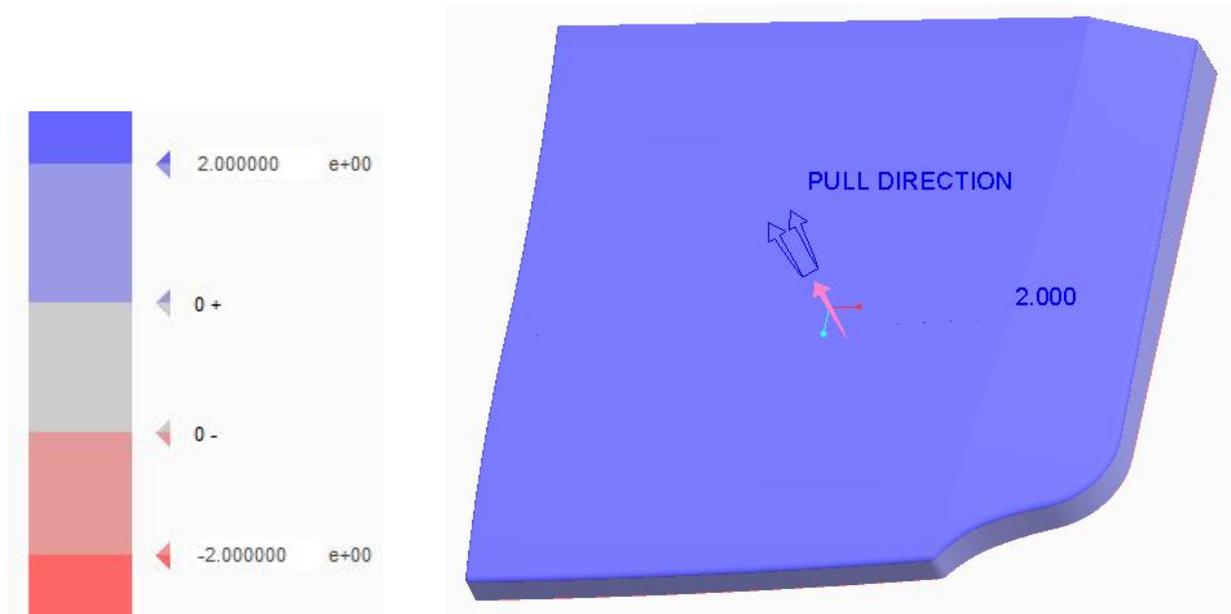
Draft Analysis dialog box will appear. As the Surface collector is active so select pick the reference part in the graphics window.



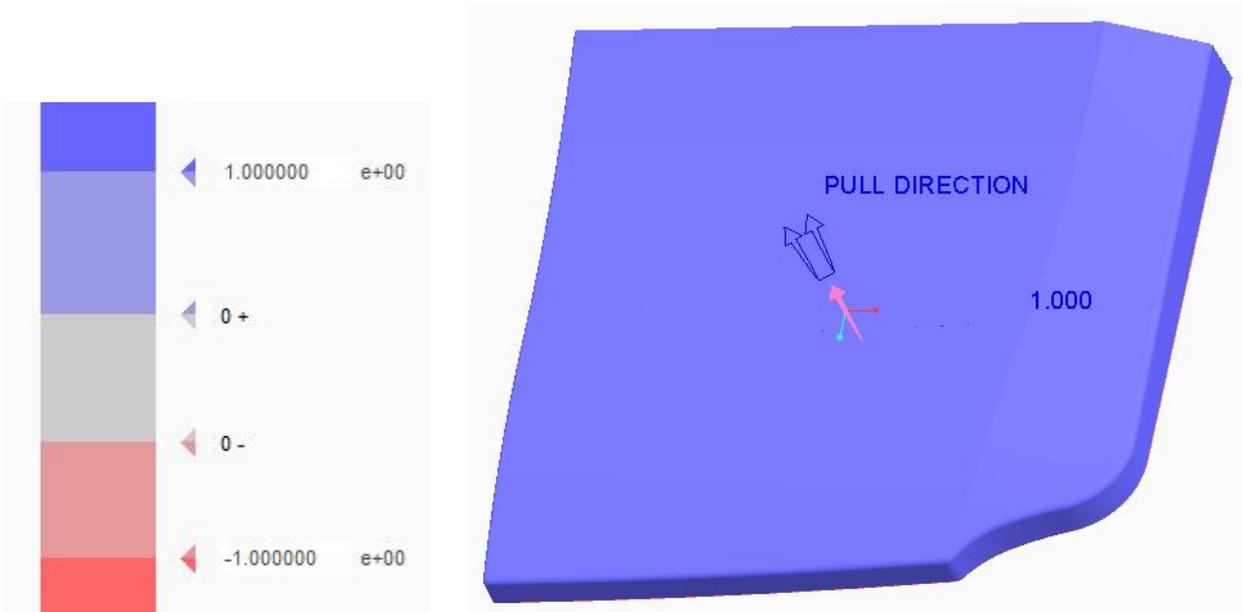
You can specify a surface, quilt or complete part to perform the draft check.

Enter **2** in the Draft angle field and hit Enter key.

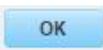
The system will display the results as shown below.



The above figure shows that the walls having less than 2° draft. Now change the Draft angle to 1 and and hit Enter key. The part will appear as shown below.



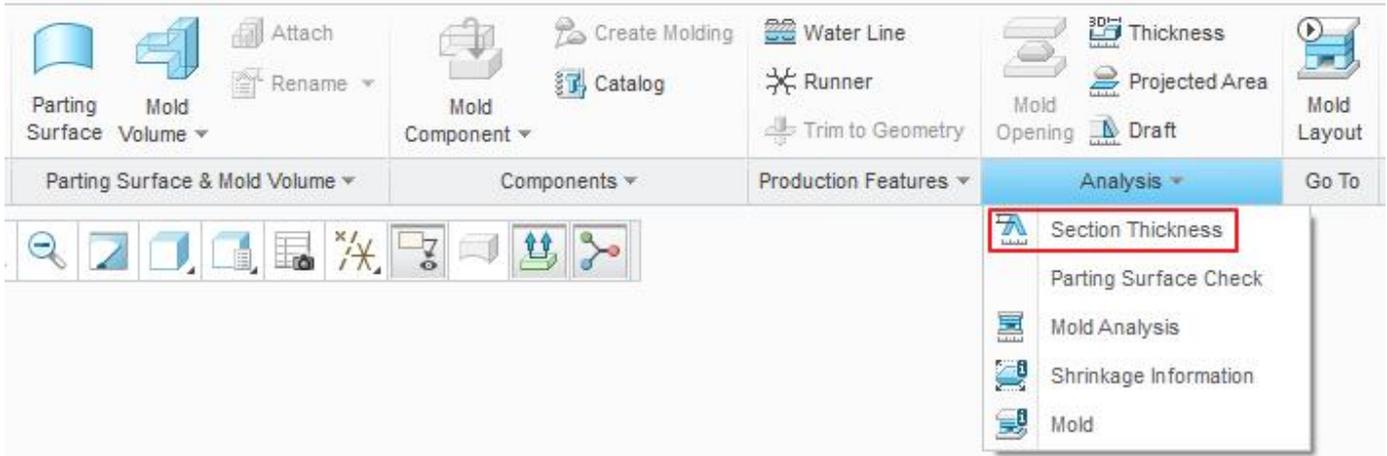
Notice that all the surfaces have more than 1° draft according to legend. The designer should decide, based upon the material to be molded and surface finish of the product, whether this draft is sufficient or not.

Pick  to close the dialog box.

 **Analyzing the part for uniform Thickness**

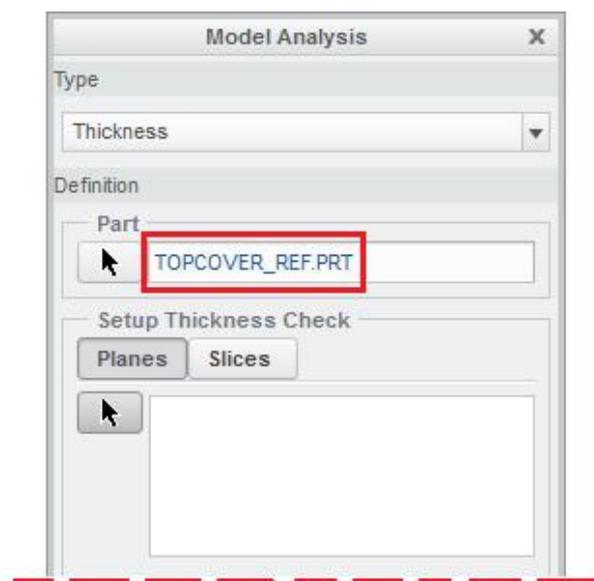
Now we will analyze the reference part for uniform thickness using Section Thickness Check tool.

Pick  icon located on Mold tab.



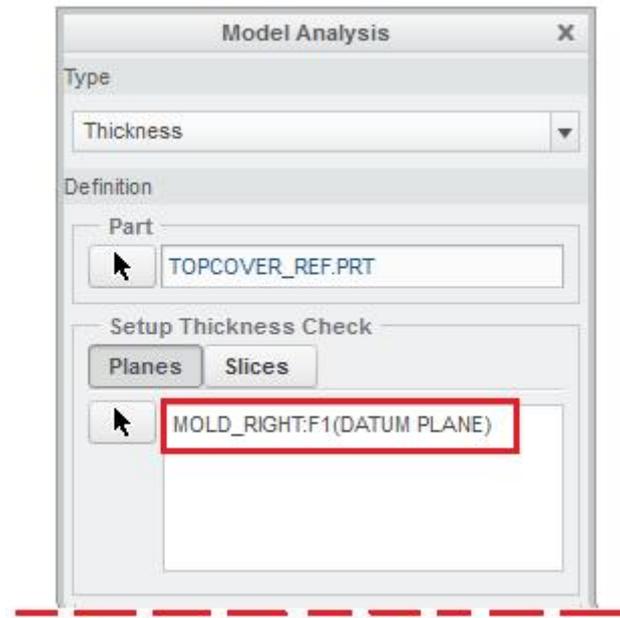
Model Analysis dialog box will appear and system will ask you to select a part to analyse.

So select the **TOPCOVER_REF** as the reference.



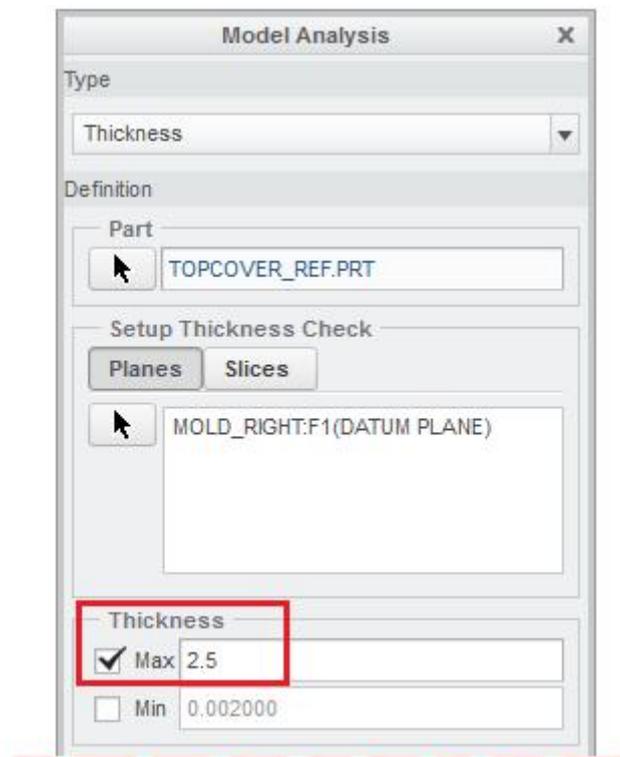
Now system will ask you to select the plane along which to check the thickness.

Select **MOLD_RIGHT** datum plane and middle-click twice.

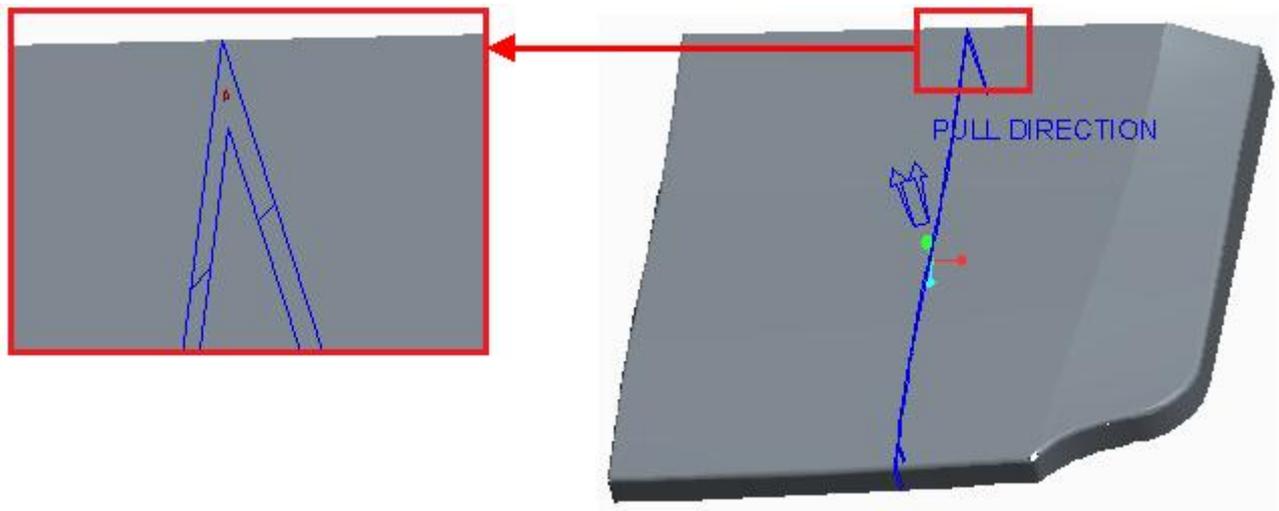


Notice that Setup Thickness Check section has two options. By default Planes (Planes) is active. This is used to analyze a part along a selected plane. More than one plane can be specified by holding the Ctrl key and picking the desired planes.

Enter **2.5** in the Max field for maximum thickness.



Pick the Compute icon and the result will appear as shown below on screen.



The red triangle shown in the above figure represents the area that is thicker than the maximum thickness entered in the Max field.

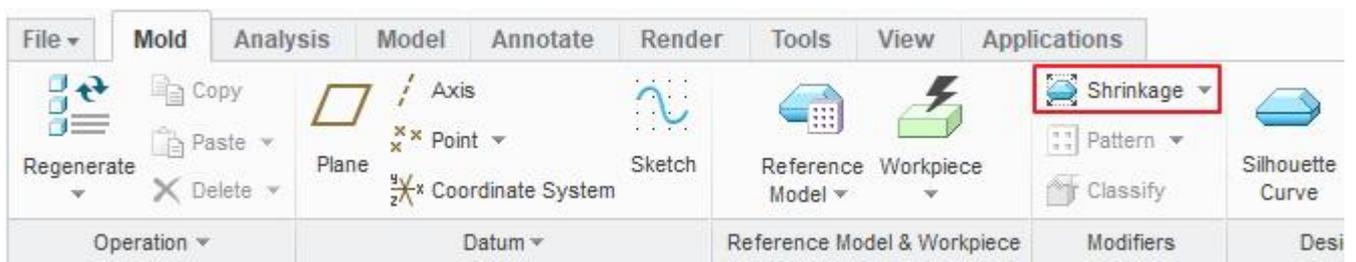
In plastic parts, as the material solidifies, it freezes from the outside (near the mold surface) toward the inside. In thick sections this results in inward pulling that can cause sink marks in the outer surfaces of the part. In addition, because thinner sections will freeze faster than thicker sections, this can result in part warpage. So in the design of parts to be injection molded, it is a good idea to maintain consistent wall thickness.

Pick to close the dialog box.

➡ Adding the Shrinkage

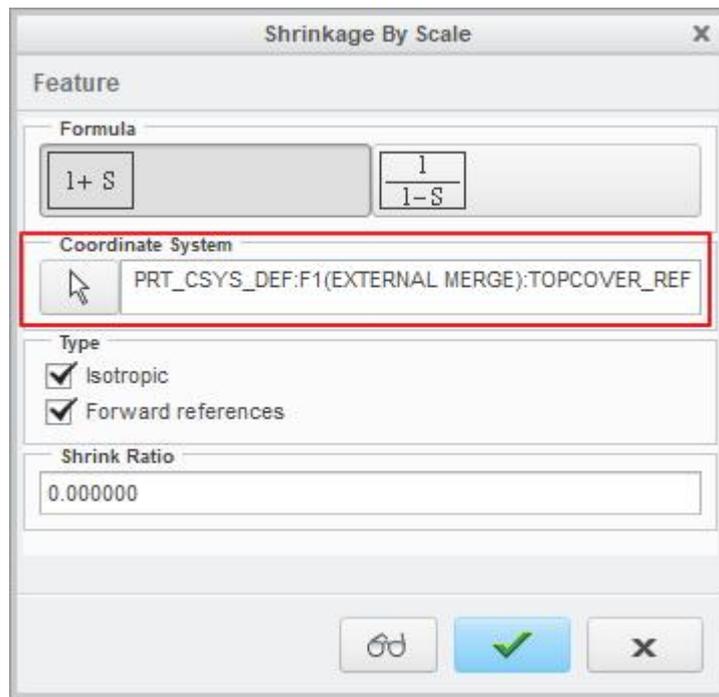
Now we will add the shrinkage to the reference model by the “By Scale” rule.

To add the shrinkage to the reference model, pick the “Shrinkage by Scale” icon () located on Mold tab.



Shrinkage by Scale dialog box will appear and system will ask you to select a Coordinate System.

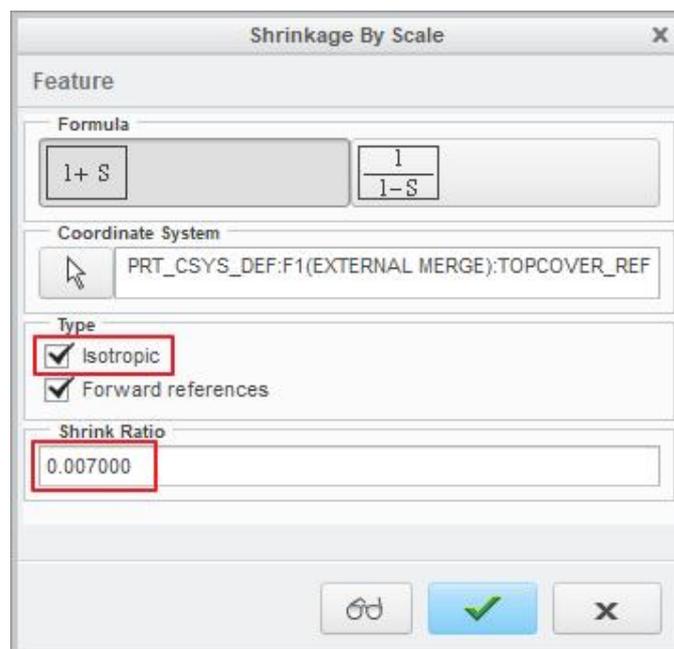
So pick the **PRT_CSYS_DEF** coordinate system of the reference part.



The coordinate system is used in calculating the shrinkage. The system scales the part with reference to this coordinate system and the point at the origin of this coordinate system remains stationary. It is good practice to select the coordinate system in the reference part as it minimizes the references.

We assume that the reference part will be produced in ABS plastic material with 0.7% shrinkage that is same for the X, Y and Z directions.

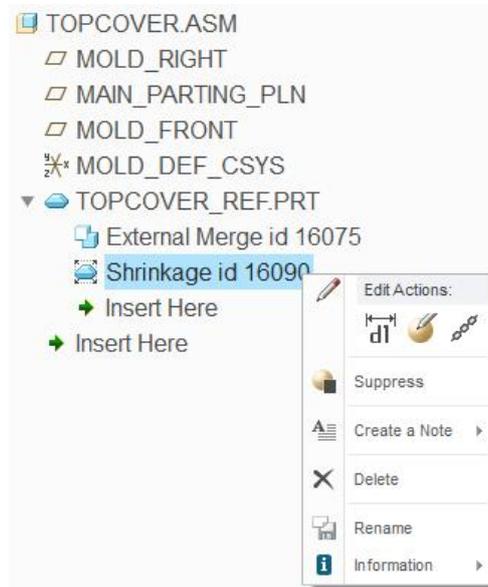
So enter **0.007** in the Shrink Ratio field



Notice that Isotropic option is checked which means that system will use the Shrink Ratio value for all the directions (i.e. Shrink Ratio will be same for X, Y and Z directions.)

Pick  to apply the shrinkage to the reference part.

Look in the model tree and notice that shrinkage is added to reference part as a separate feature. You can right-click over the shrinkage feature to access a shortcut menu as shown below.



This menu can be used to delete or modify the shrinkage by picking appropriate options.

Creating the Workpiece

Now we will create an Automatic workpiece. But before that make sure that “Mold Component Catalog” is installed. If the catalog is not installed the following message will appear.

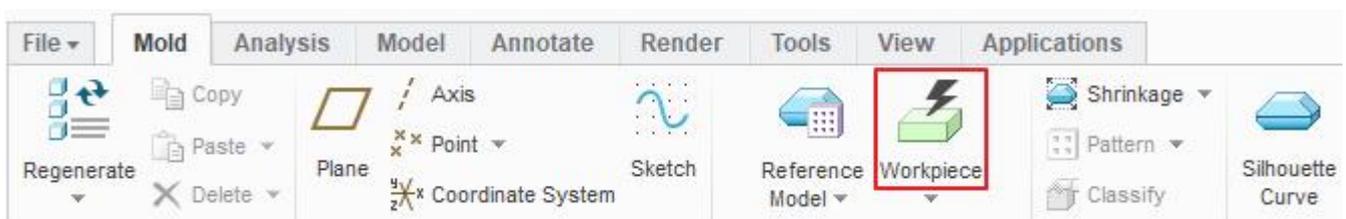
“No files or directories found.”

If this message appears then you must run the Creo setup and install the Mold Component Catalog.

We will use the Automatic Workpiece Creation functionality of Creo MOLD.

The Automatic Workpiece creation functionality gives you the ability to create a workpiece based on the reference model's size and position.

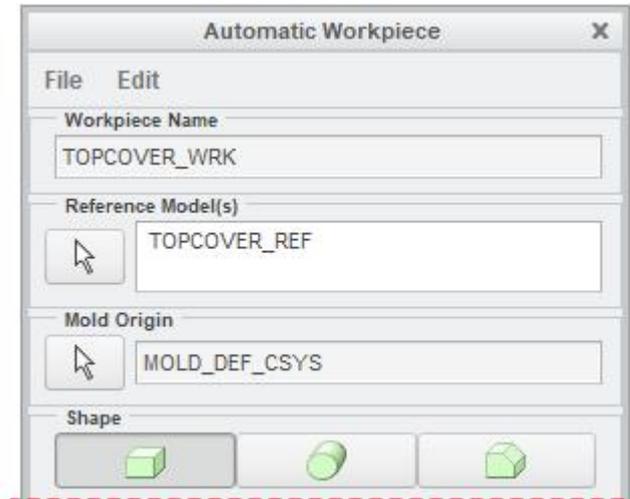
To create the automatic workpiece, pick the “Create a Workpiece” icon () located on Mold tab.



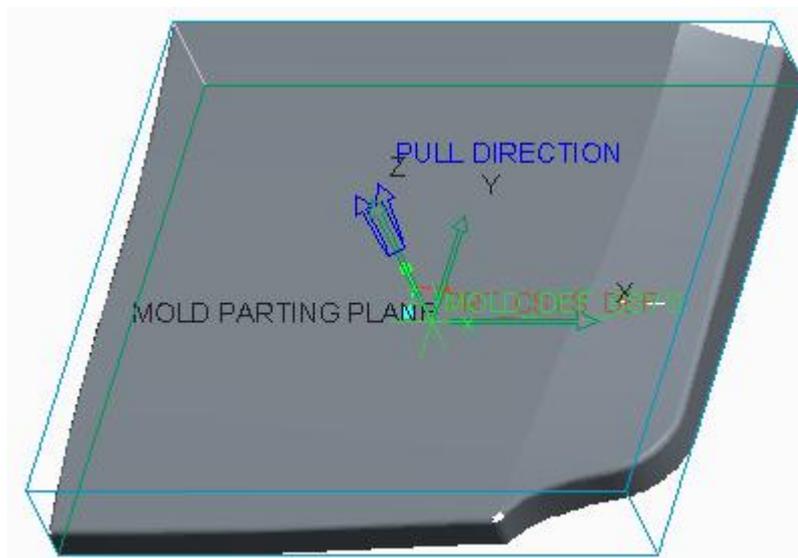
Automatic Workpiece dialog box will pop up and system will ask you to select a Coordinate System to orient the workpiece.

So select **MOLD_DEF_CSYS**.

The Z axis of this coordinate system should always be parallel to the pull direction (Mold Opening direction).



Look in the graphics window and notice that a rectangular bounding box appears around the reference part.



The bounding box is the smallest box which completely encapsulates the reference model.

The dialog box gives us different option to control the size of workpiece. Currently we will use the Offsets option.

Enter the offset values in the dialog box as shown in the figure below.

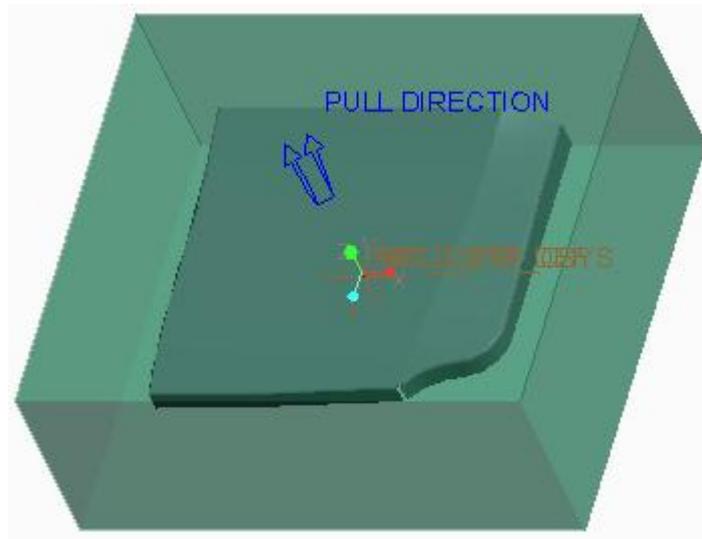
	-	+
X direction	70.000000	70.000000
Y direction	60.000000	60.000000
Z direction	75.000000	40.000000

Overall Dimensions	
X	416.925000
Y	376.784622
+ Z Cavity	85.125124
- Z Core	75.000000
Z Total	160.125124

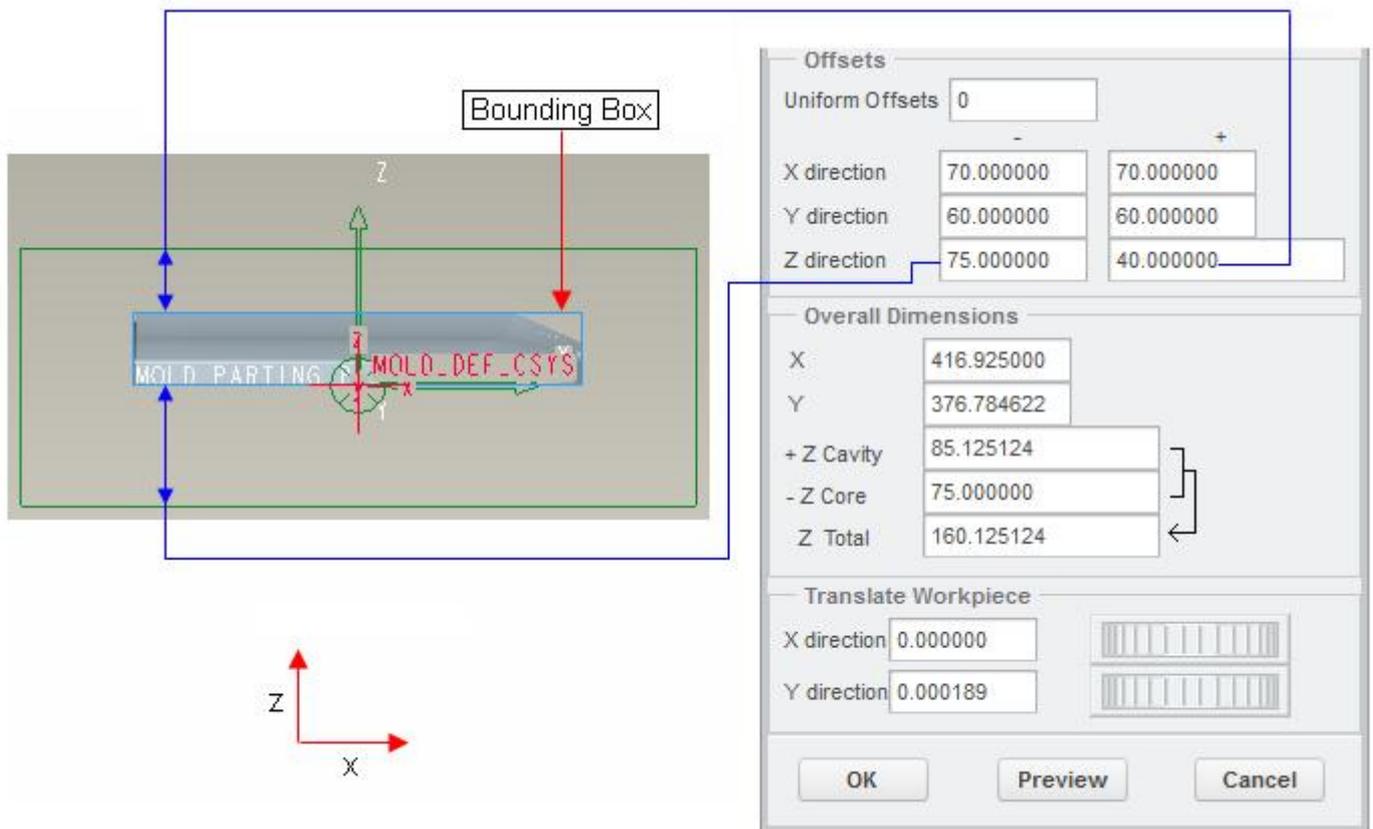
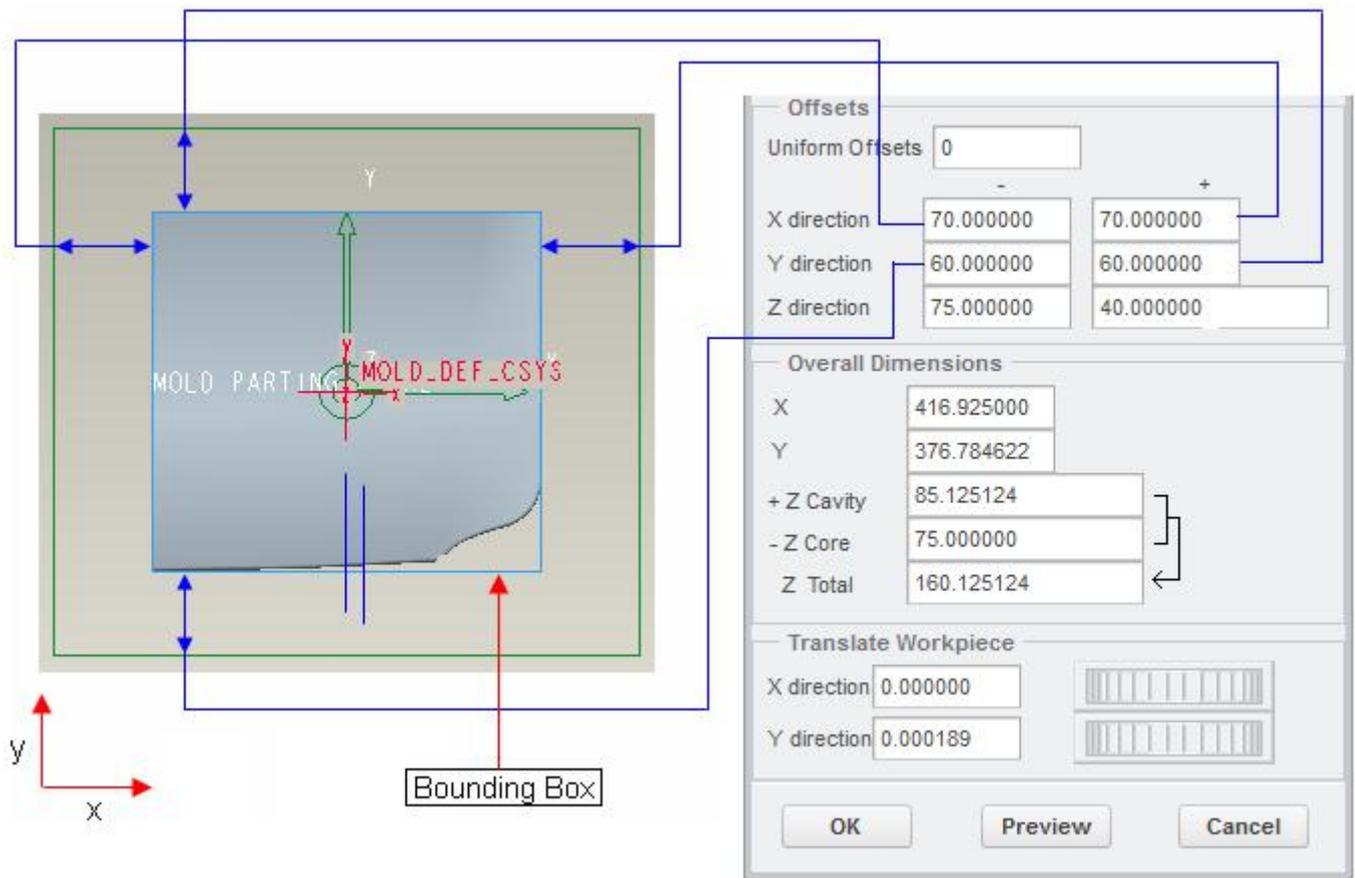
Translate Workpiece	
X direction	0.000000
Y direction	0.000189

The Offsets add the material around the bounding box by the input value for the specified direction.

Pick to complete the definition of workpiece. It will appear as shown below.



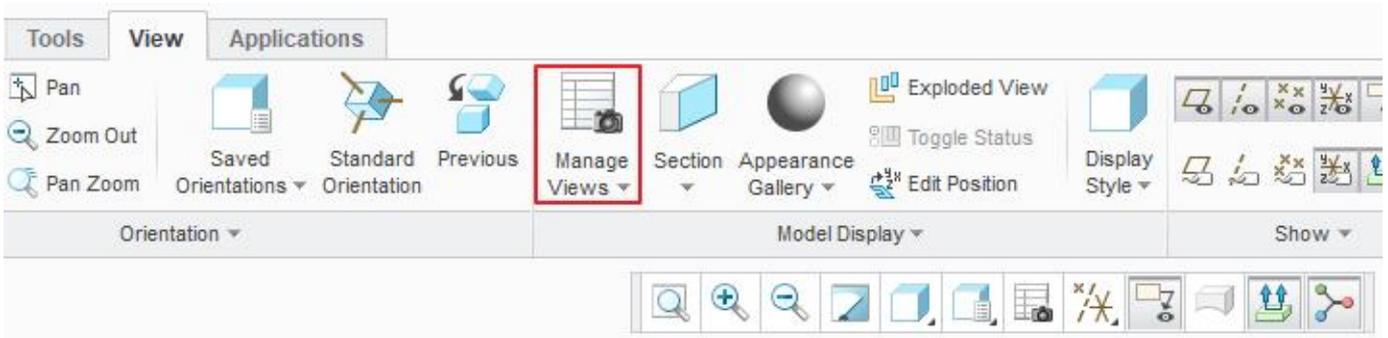
The offsets of the workpiece can be explained by the following figures.



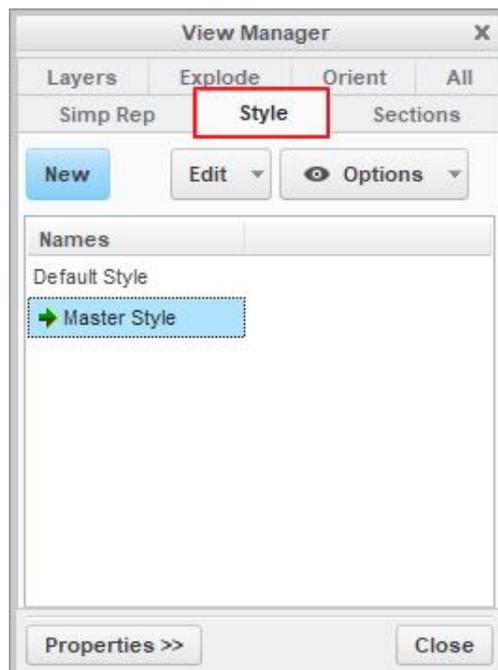
➔ Defining a Style state

Now we will define a style state that shows the workpiece in wireframe mode.

So select **View Manager** icon () on the View tab (or on the Graphics toolbar) to launch the View Manager Dialog box.

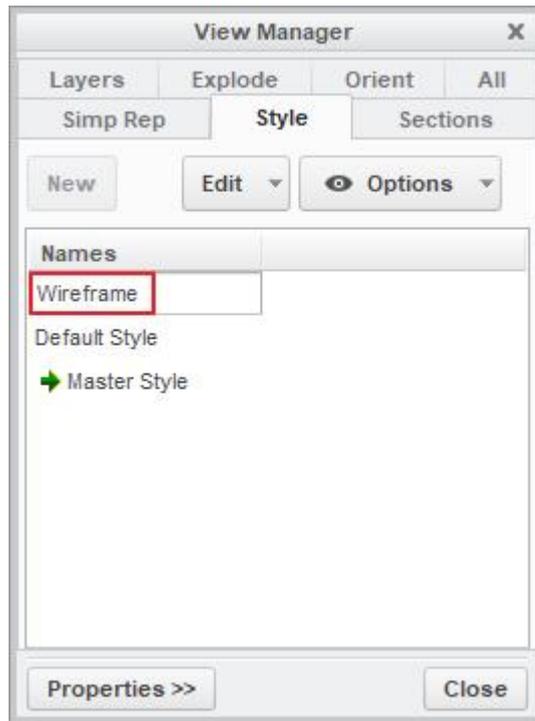


Make the Style page active, if not by default, as shown below.

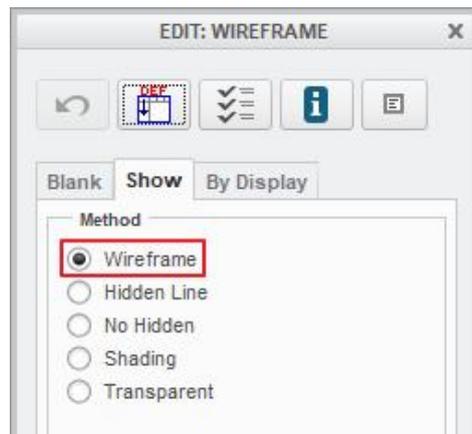


To define a new style state, select  .

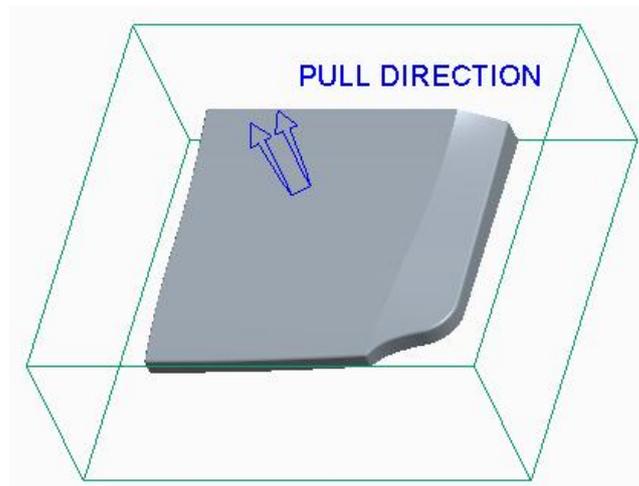
Enter **WIREFRAME** and hit Enter key on keyboard.



The EDIT dialog box will pop up. Select **Show** tab and notice that Wireframe is selected by default.



Select the workpiece with left mouse click and select **OK** tab to exit. Notice that workpiece display changes from shading to wireframe as shown below.



Select **Close** to exit the View Manager Dialog box.

➔ Creating the Parting Surface

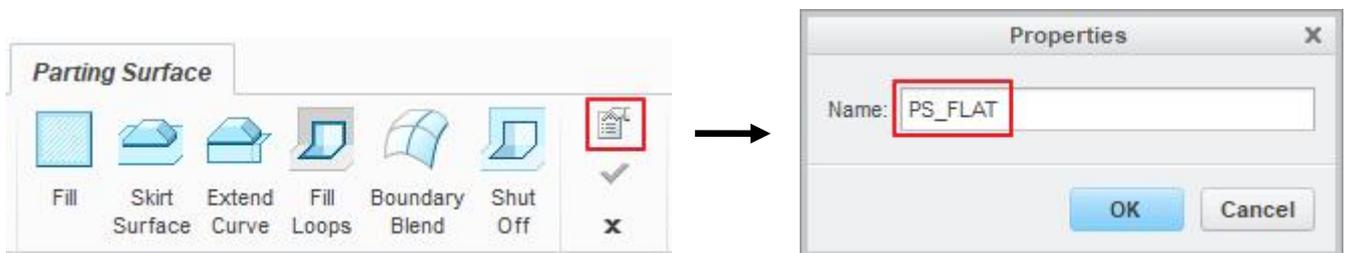
Now we will create a flat parting surface for this model.

So pick the parting surface icon () that is located on Mold tab.



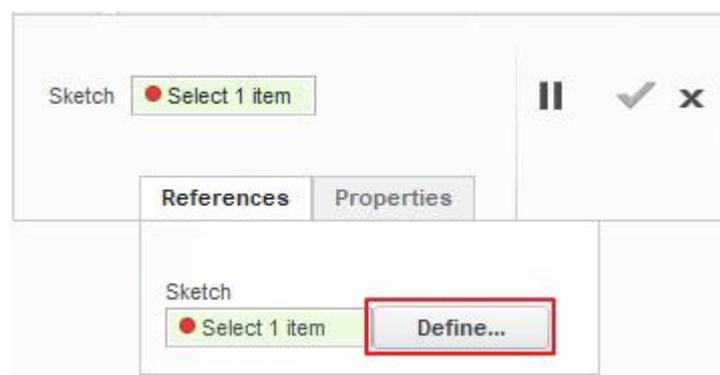
Pick  icon to give a name to this parting surface. (Note: if this icon is not active then pick anywhere in the graphics area, and it will become active.)

Enter **PS_FLAT** as the name of the parting surface and select **OK**.

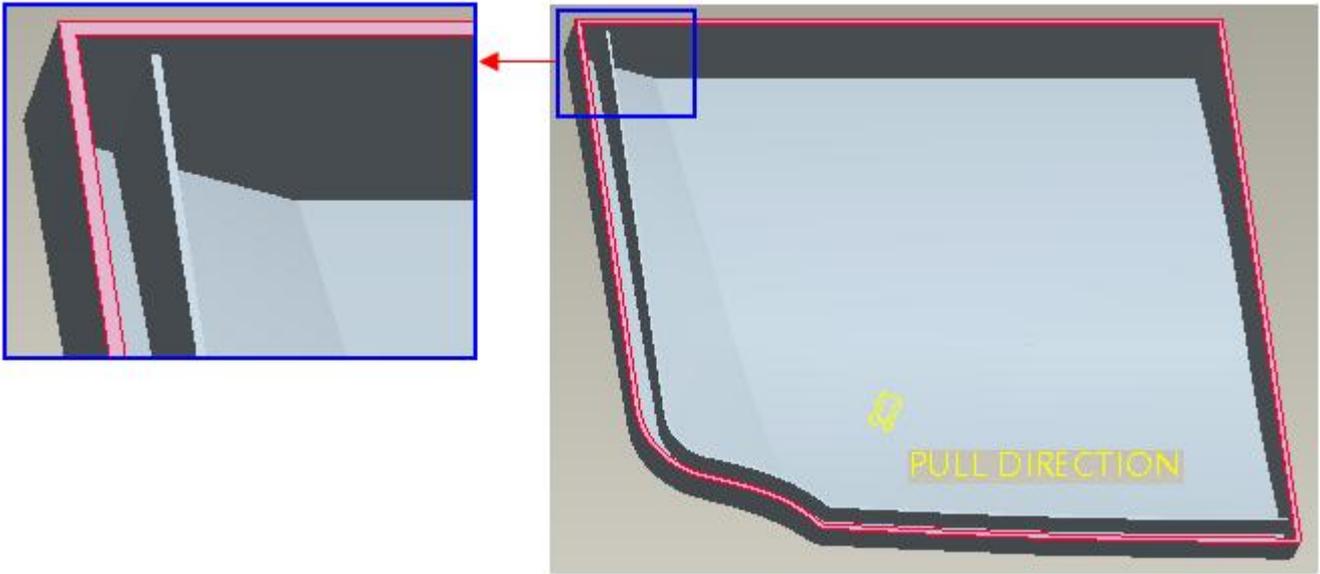


To create a flat surface, pick Fill icon ().

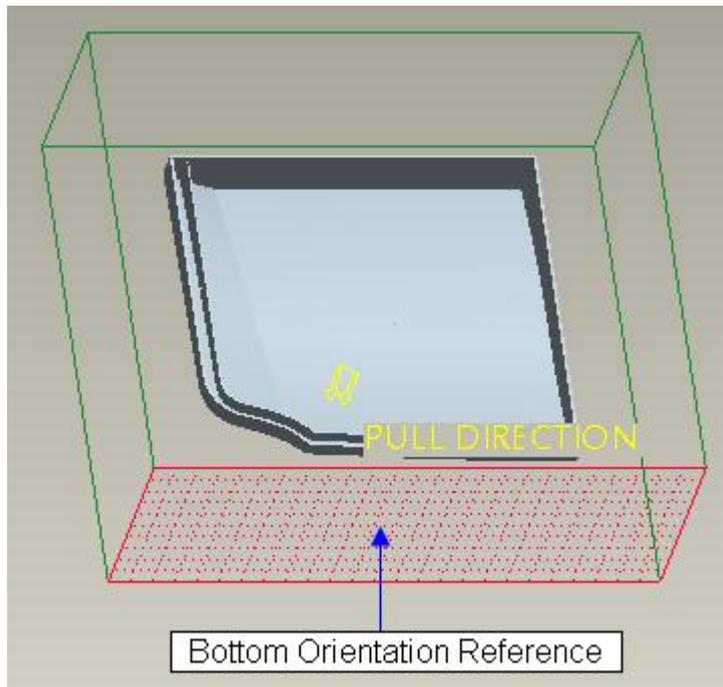
Pick **Define...** tab in the References panel.



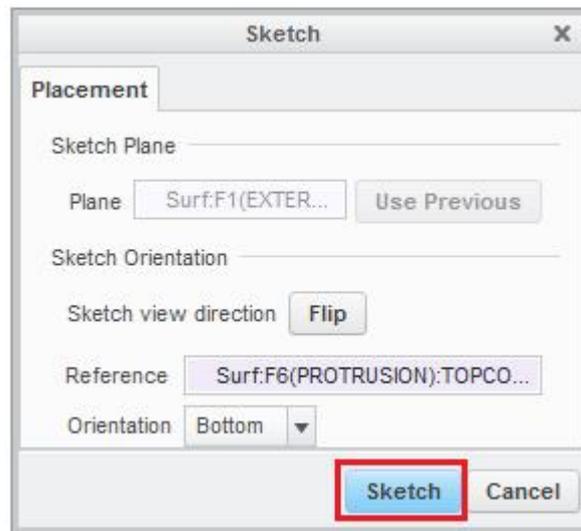
Now rotate the model and select the following highlighted surface as the sketching plane.



Pick the following surface as Bottom Orientation Plane.

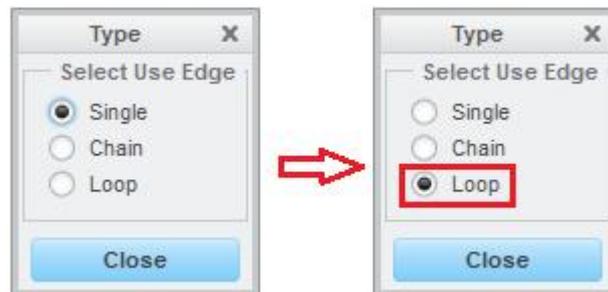


Pick **Sketch** and system will place you in sketcher.

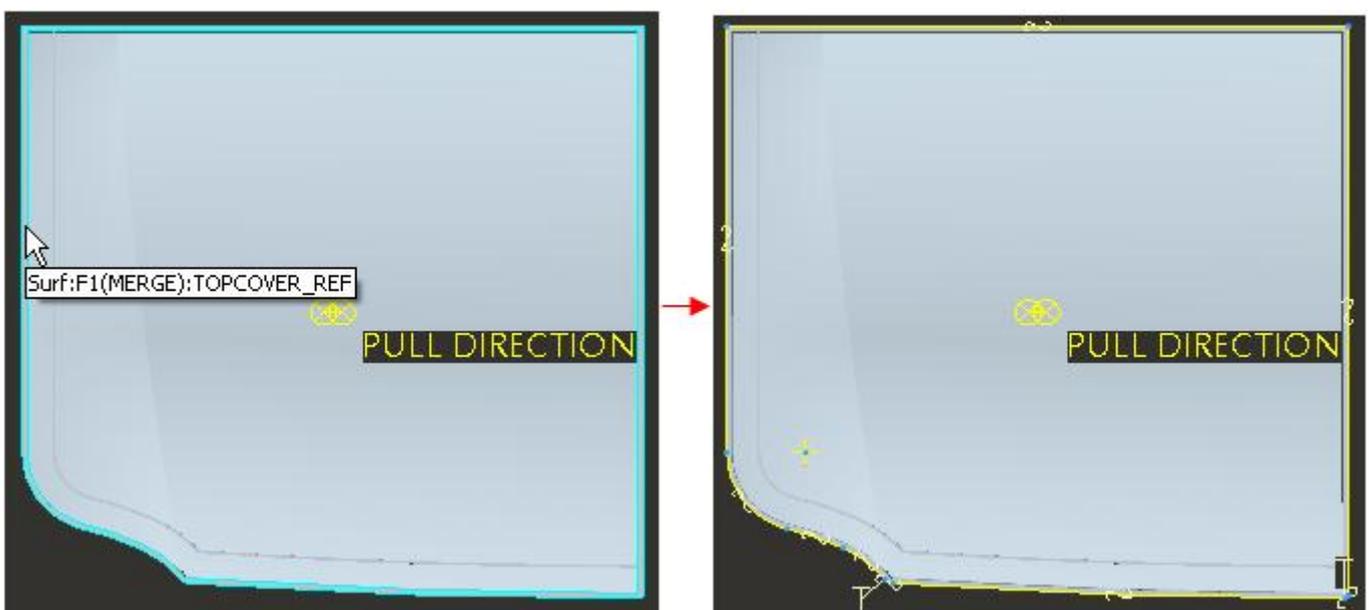


Pick project icon ( Project).

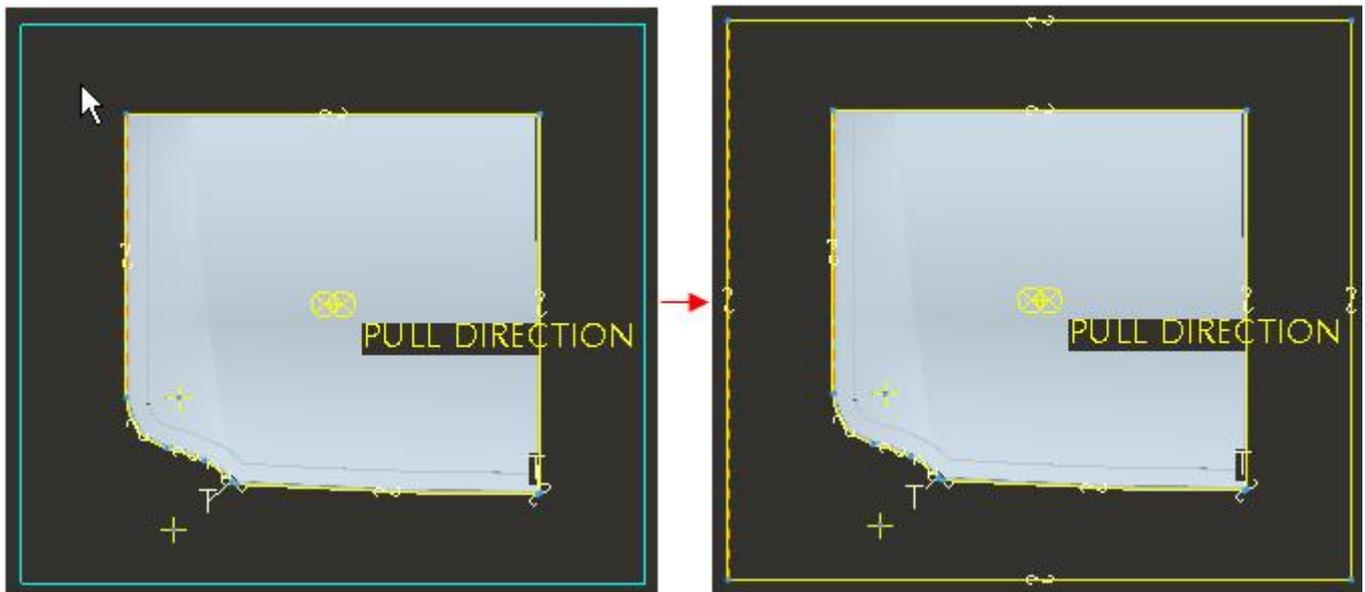
In the Type menu, select **Loop** option.



Select the outer loop of bottom surface of reference part as shown below. (Note: Sketch entities are shown in such colors so that they are easy to distinguish. You will see different colors depending upon the configuration of your system.)



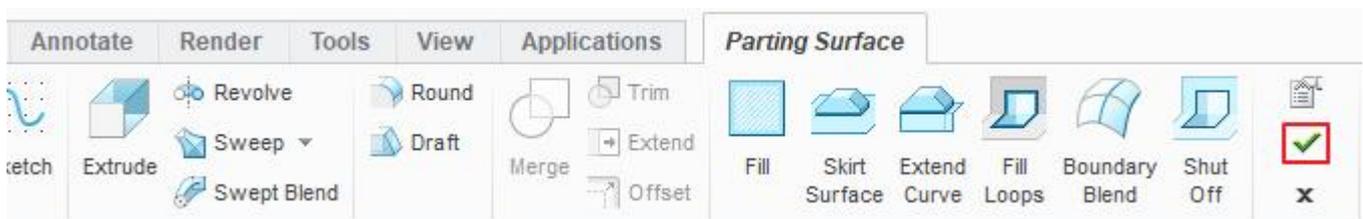
Now select face of the workpiece so that sketch appears as shown below (some detail not shown for visual clarity).



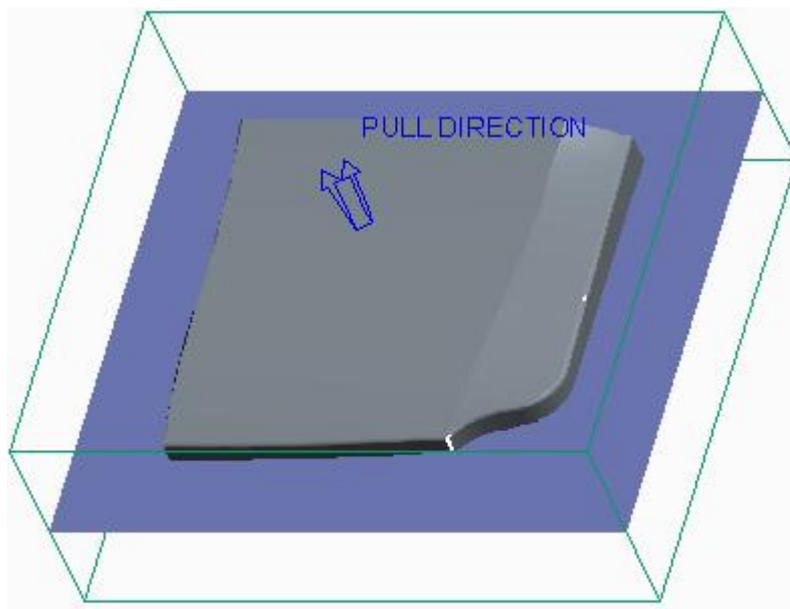
Pick  to save the sketch and exit.

Pick  to complete the flat surface feature.

Pick  and it completes the definition of the parting surface.



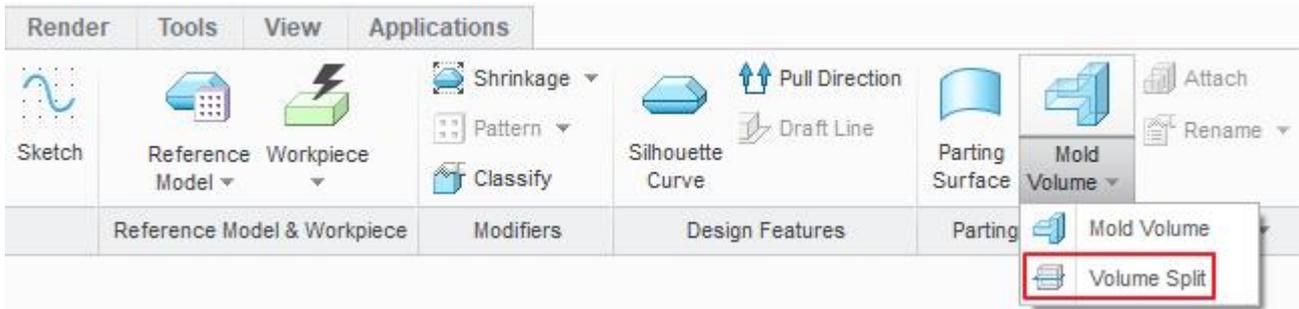
The parting surface will appear as shown below .



➔ Splitting the Workpiece

Now we will split the workpiece into core and cavity volumes

Pick  Volume Split icon located on Mold tab.

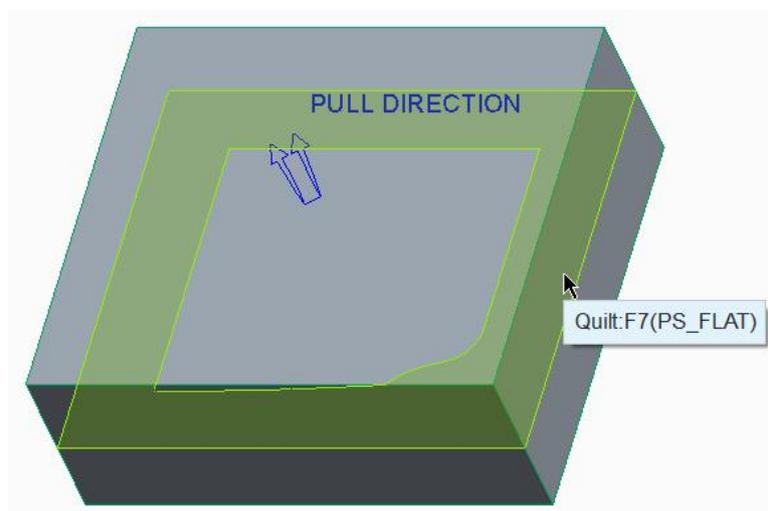


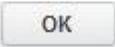
Leave the default options of Two Volumes and All Wrkpcs intact in the SPLIT VOLUME menu as we want to split the workpiece in two volumes.

Select **Done** or middle-click to proceed.

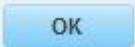


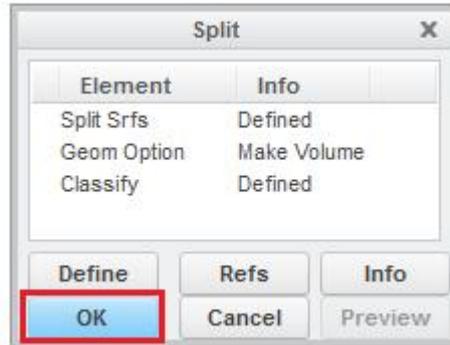
System will ask to select the parting surface for splitting the workpiece, so select the previously defined surface **PS_FLAT** as shown below.



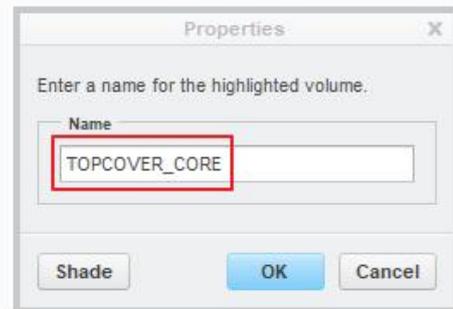
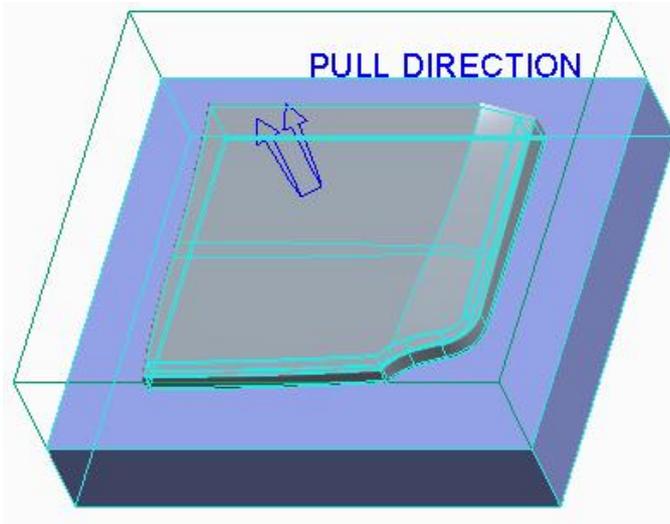
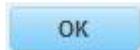
Pick  or middle-click to complete the selection of surfaces.

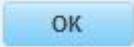


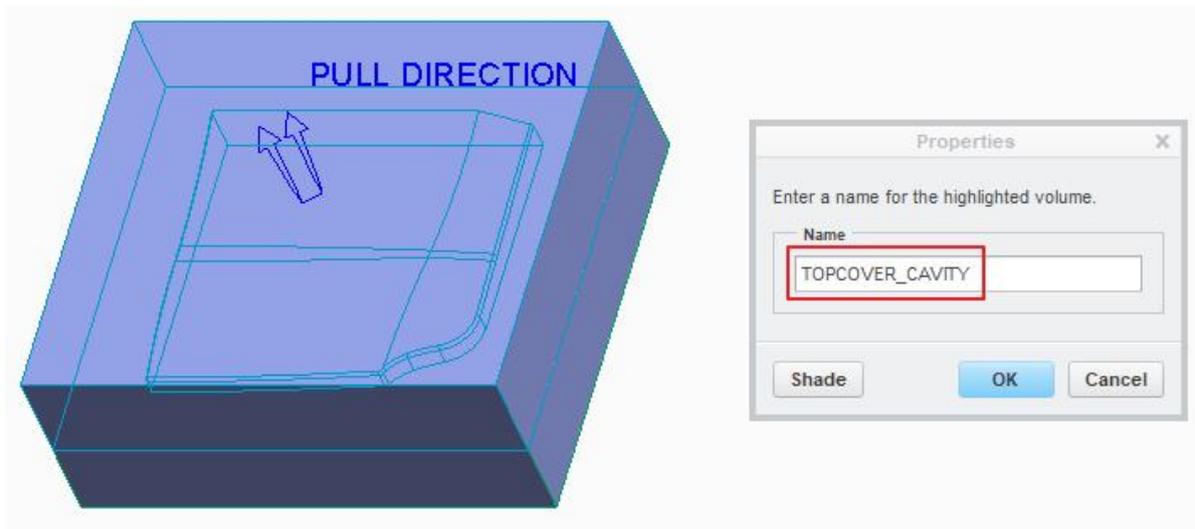
Pick  or middle-click to exit the Split dialog box.



Enter **TOPCOVER_CORE** as the name of the volume highlighted in the figure below and pick



The system will highlight the second split volume. Enter **TOPCOVER_CAVITY** as the name of volume highlighted in the figure below and pick .



Remember that splitting does not alter the workpiece geometry. When a split operation is performed to the workpiece, the system

1. Creates a volume (closed quilt) by copying all the surfaces of the workpiece
2. Subtracts the reference part out of this volume
3. Divides this volume into regions (volumes), splitting it at parting surface
4. Gives unique names to the mold volumes to either side of the parting surface.

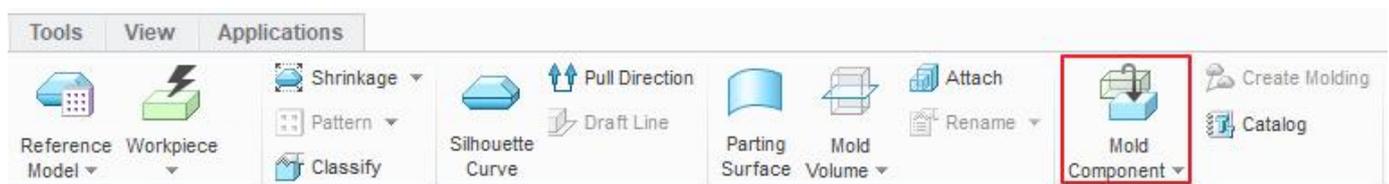
If you look in the model tree you will notice two features appear in the model tree for the split volumes as shown below.



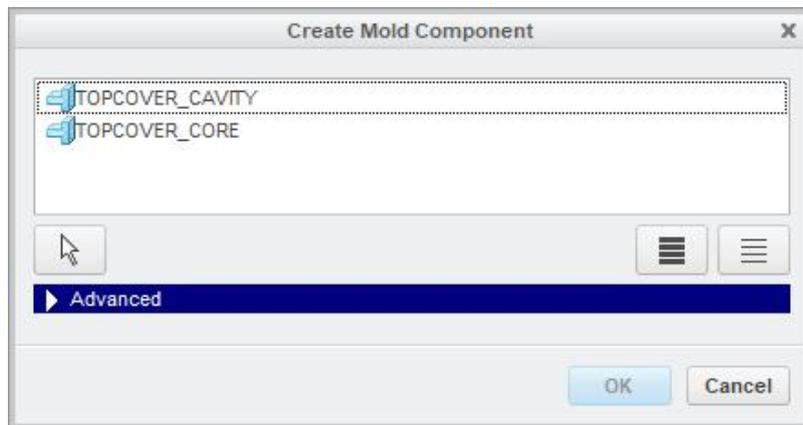
➡ Extracting the mold components

Now we will extract solid parts out of core and cavity volumes.

To extract the mold components, pick  icon on the Mold tab.

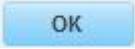


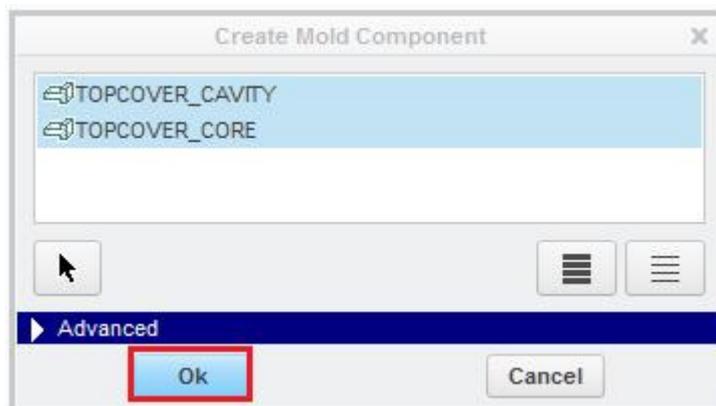
Create Mold Component dialog box will appear as shown below.



The dialog box lists all the volumes present in the mold assembly.

Pick  to select both the volume listed in the dialog box.

Pick  to proceed with current selection.



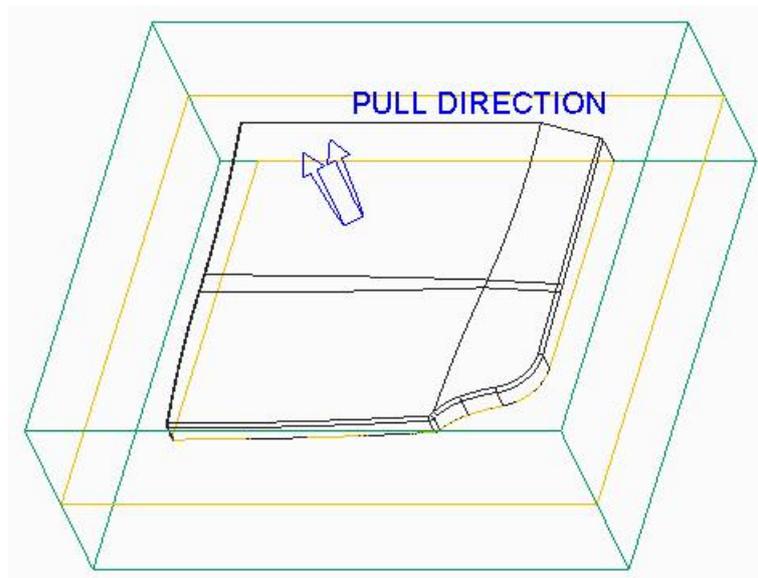
The system fills the designated volumes with solid material to create corresponding components. The name of the resulting part is the same as the name of the volume.

If you look in the model tree you will notice two new components; TOPCOVER_CORE.PRT and TOPCOVER_CAVITY.PRT



An extracted component retains associativity with the parent volume; if the volume is modified, the component is updated when the mold model is regenerated.

If you look in the graphics window you will notice that system has automatically blanked the parent volumes (which appear in magenta color by default) of the extracted components as shown below.



➡ Uncluttering the view

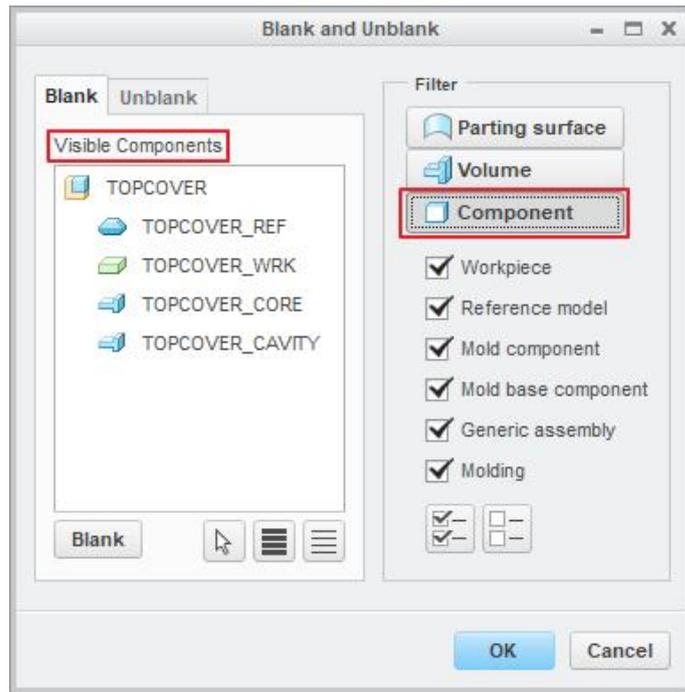
Creo Tool Design provides the “Blank and Unblank” dialog box to remove or add components, parting surfaces, or volumes to the current display of the model.

Now we will remove all the components except TOPCOVER_CORE from the current display.

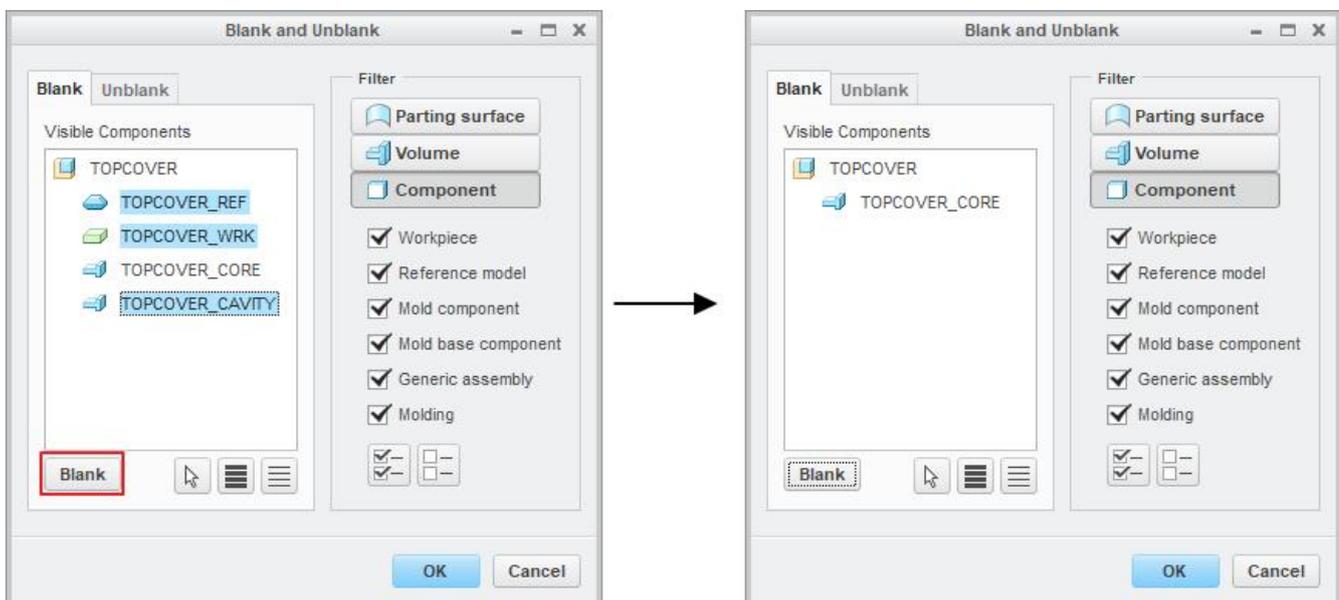
Pick  on the View tab.



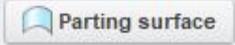
Blank and Unblank dialog box will appear as shown below. Notice that Component Filter is active and all the components present in the current mold assembly are listed as Visible Components.

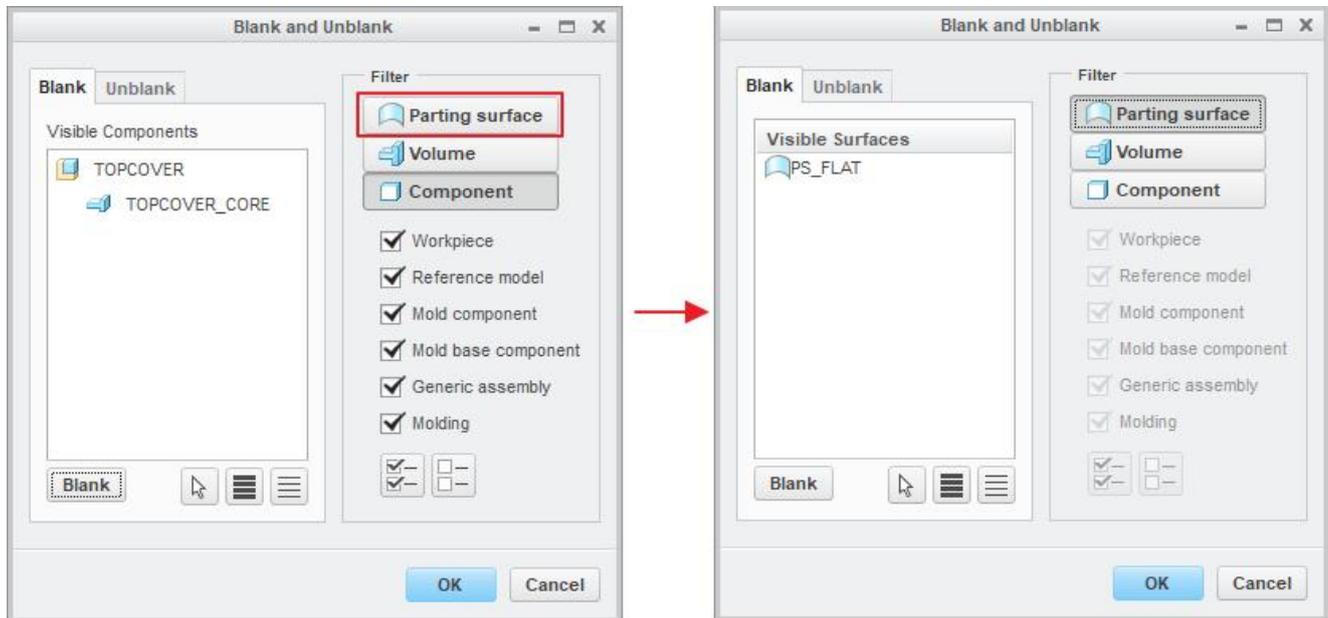


Select TOPCOVER_REF, TOPCOVER_WRK and TOPCOVER_CAVITY while holding down the Ctrl key and pick **Blank** tab.

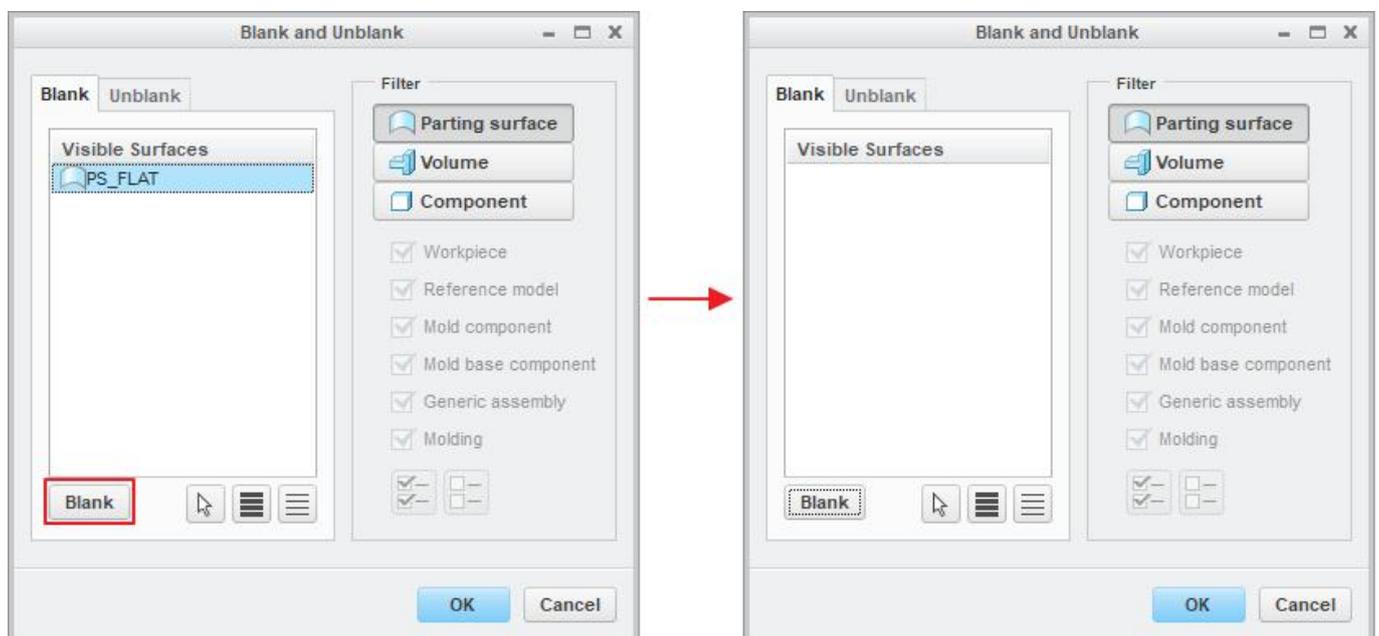


Now we will remove the parting surface from the current display.

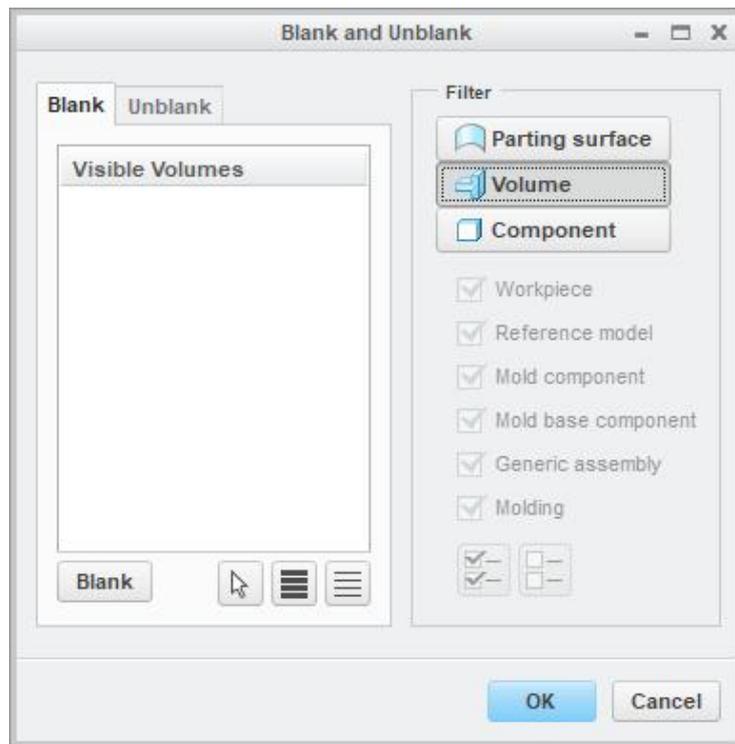
Activate the Parting Surface filter by picking  tab.



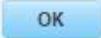
Select PS_FLAT and pick .

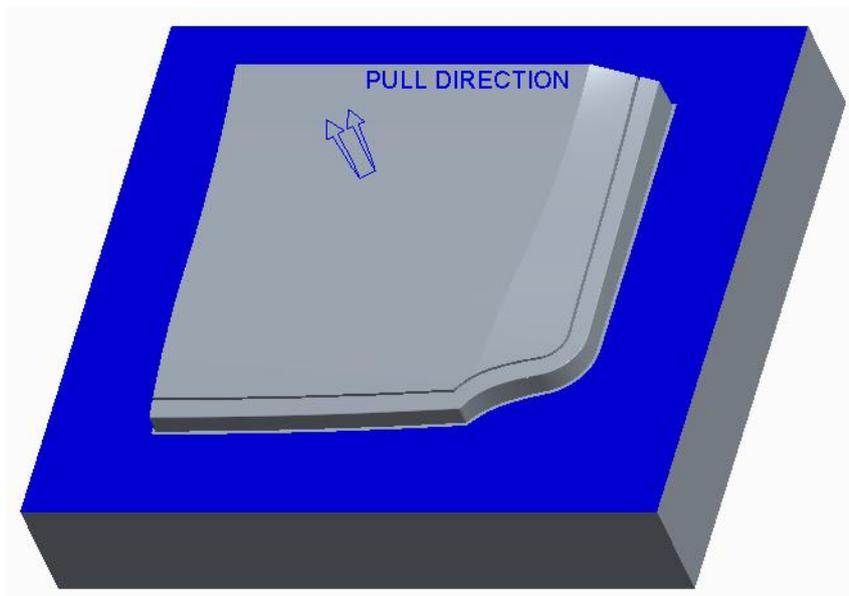


Now activate the Volume filter by picking  tab and you will notice that there is no volume listed as Visible Volumes as shown below.



It is so because when system extracts mold components out of mold volumes, it automatically blanks the parent volumes.

Pick  to exit the dialog box and notice that only the TOPCOVER_CORE is displayed in the graphics window as shown below.

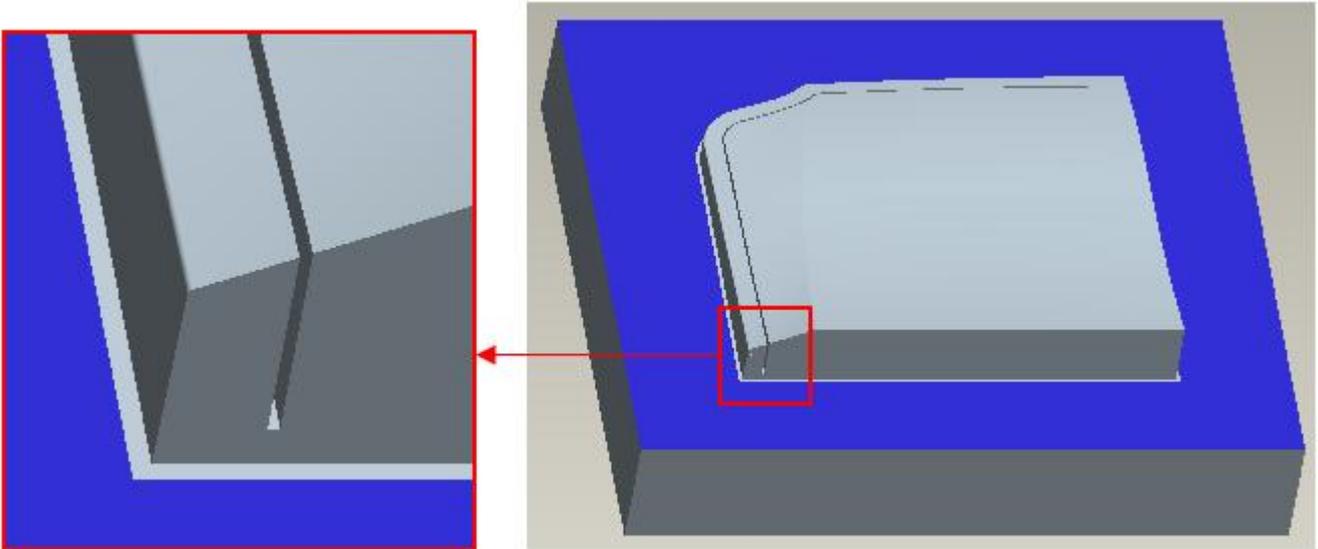


Notice that system has given a different color (blue) to split surface. This is the surface where there is a metal to metal contact between core and cavity parts.

This coloring scheme is very helpful while deciding machining strategy for the component as the NC Programmer can easily differentiate between “metal to plastic” contact surfaces from “metal to metal” contact surfaces.

⇒ Splitting off the insert volume

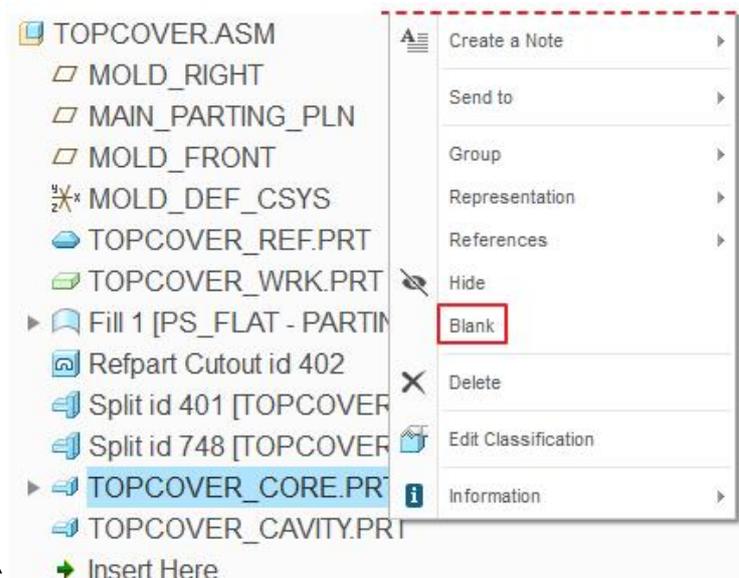
If you look at the TOPCOVER_CORE.PRT you will notice that there is a very thin slot as shown below.



From machining point of view it is very difficult to machine this thin slot into solid core. So inserts should be used to form it. Now we will create another parting surface that will be used to split the TOPCOVER_CORE volume at the position of the slot.

Before building the parting surface, blank the TOPCOVER_CORE.PRT

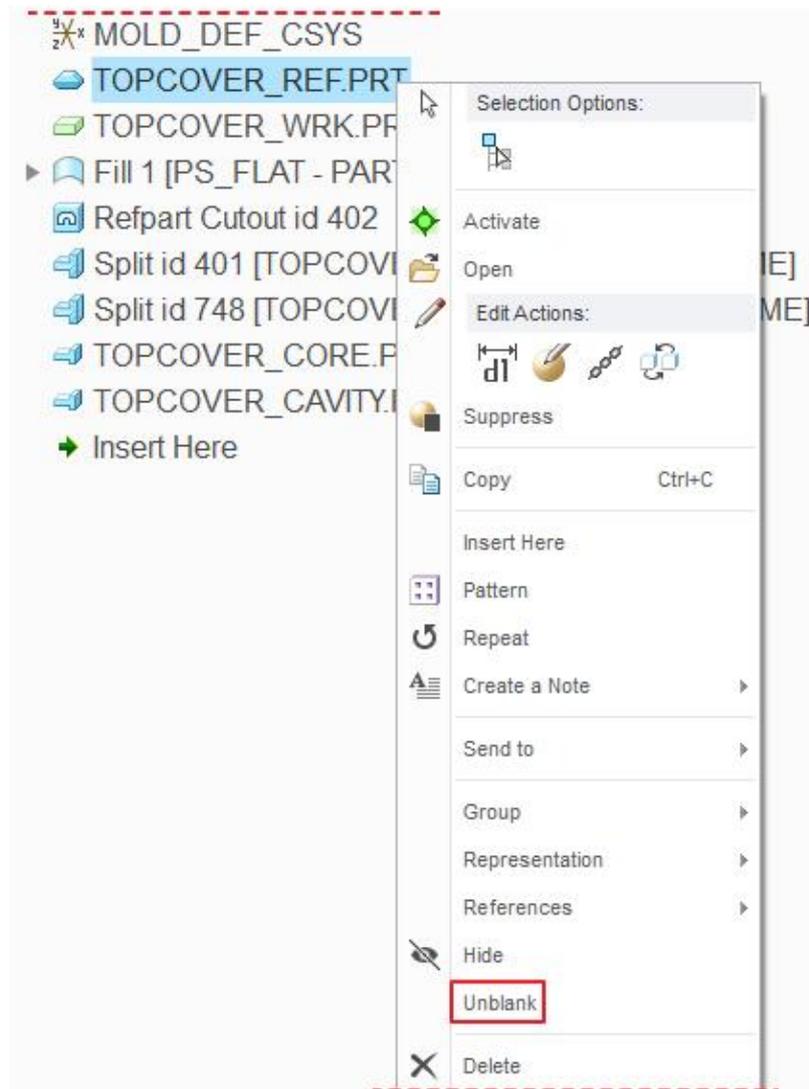
So right-click TOPCOVER_CORE.PRT in the model tree and pick **Blank** as shown below.



You can blank or unblank objects either by using Blank and Unblank dialog box or by directly in the model tree.

While defining the new parting surface we will reference the geometry of TOPCOVER_REF part. So first we will unblank it.

So right-click TOPCOVER_REF.PRT in the model tree and pick **Unblank** as shown below.



It is highly desirable not to use geometry of extracted components or mold volumes as a reference while building the parting surfaces. Because if the geometry, we use as a reference, is consumed later by a split operation, the parting surface will fail to regenerate and we will have to redefine the surface and specify new references.

Also unblank the workpiece as some operations require the workpiece displayed to work.

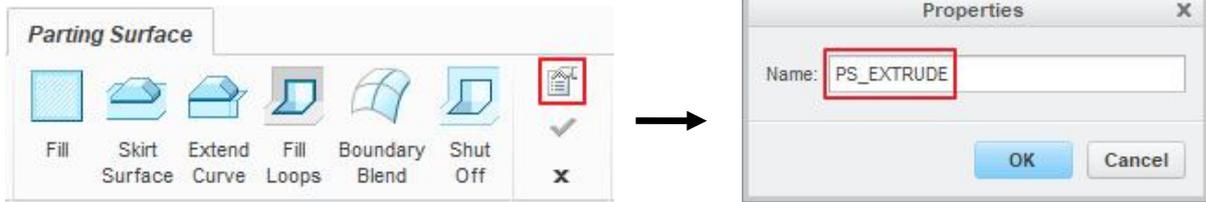
Now we will create the parting surface that will consist of an extruded surface feature. The surface will be extruded in both directions with ends capped.

To create the parting surface, pick the parting surface icon () that is located on Mold tab.

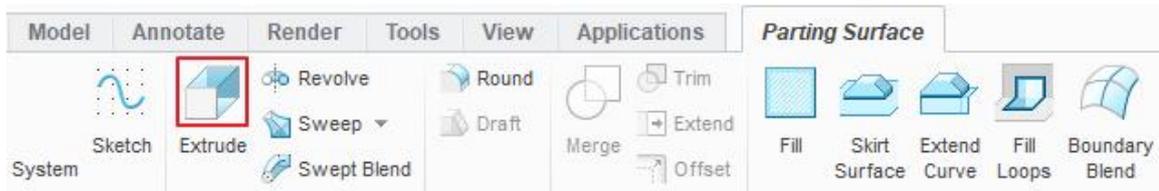
Pick  icon to give a name to this parting surface. (Note: if this icon is not active then pick anywhere in the graphics area, and it will become active.)

Enter **PS_EXTRUDE** as the name of the parting surface then select

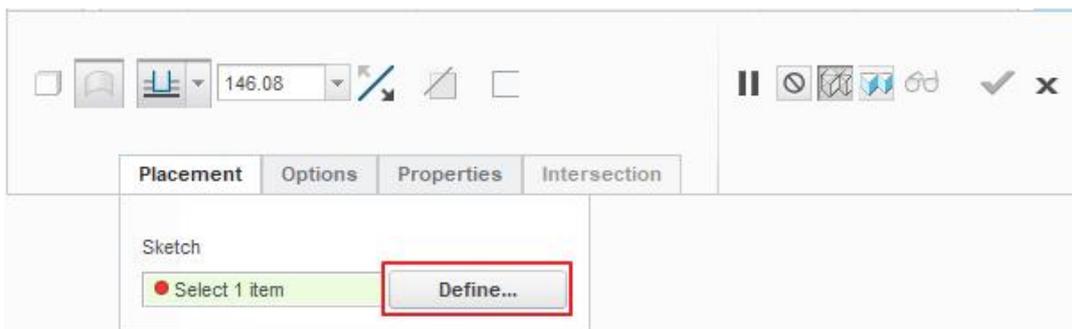
OK



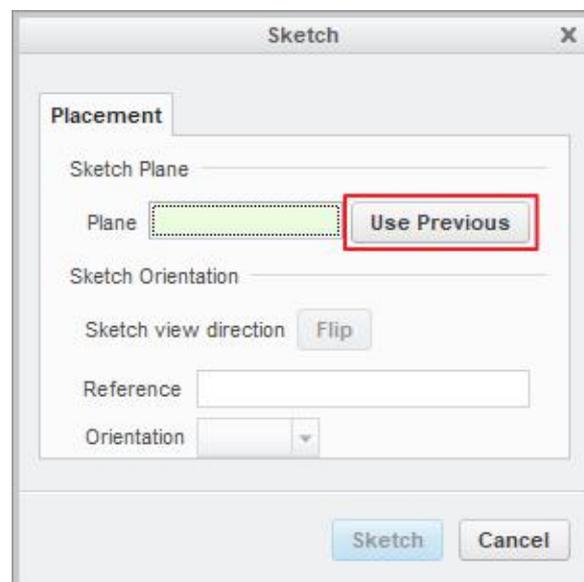
Pick  icon to invoke the Extrude Tool.



Pick  tab in the Placement panel.



For the sketching plane select .



Use Previous option is used to select the Sketching and Orientation references of the last feature.

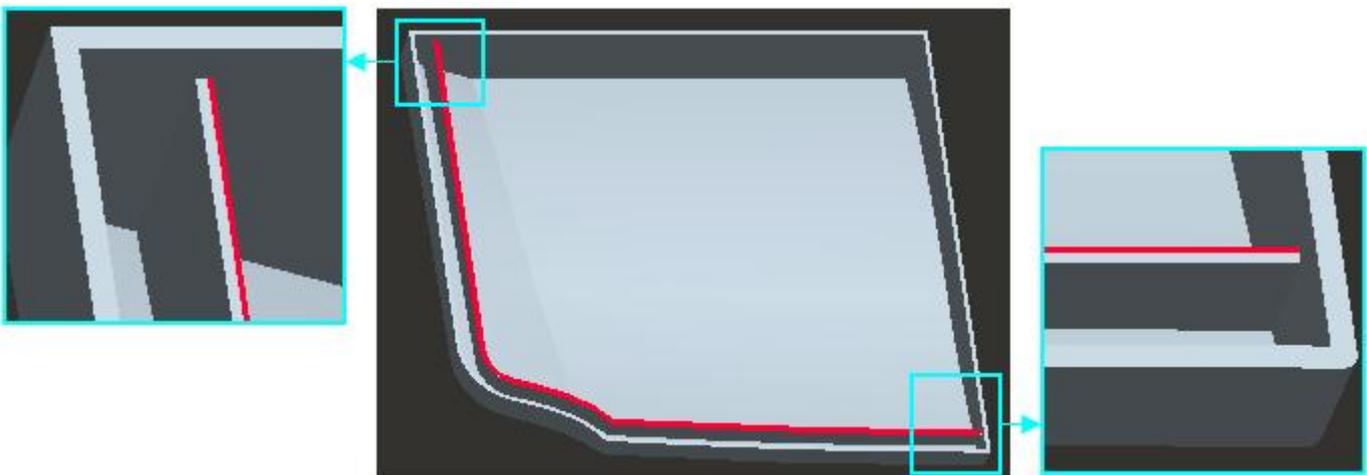
Remember that last feature, which required sketching references, was the Fill surface feature in this exercise. So system will use the same sketching and orientation plane that were used for Fill surface.

Pick Offset icon ( Offset).

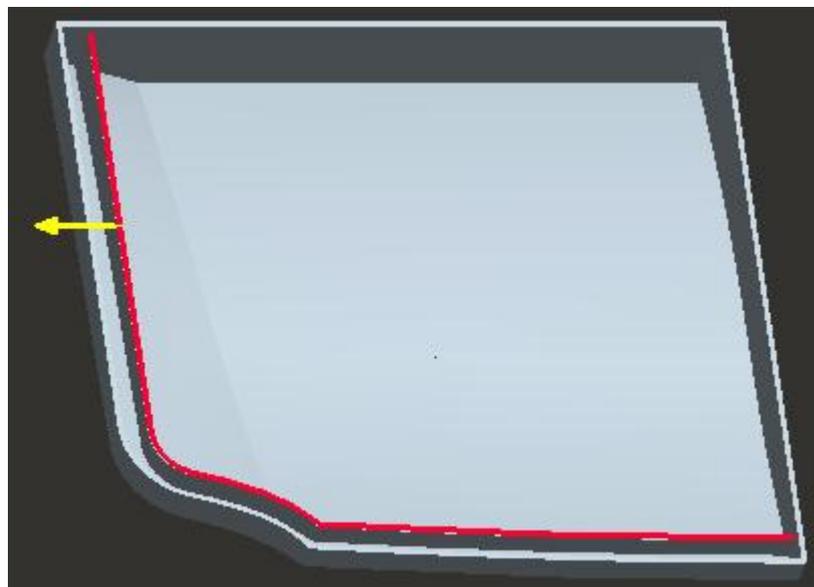
In the Type menu, select **Chain** option.



Select the inner chain of edges of lower surface of rib as shown below.

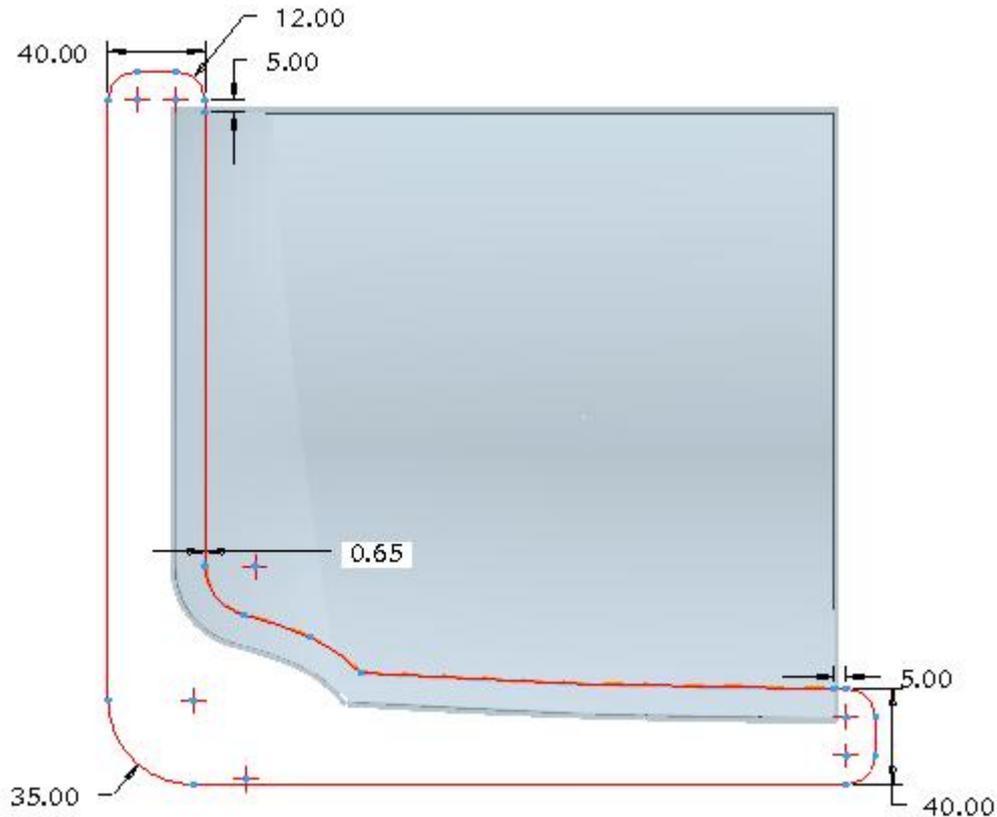


Enter the **0.65** as the offset value for the chain to place it in the mid of the rib.



We have placed the chain at the mid of the rib because most of the mold designers like it as it facilitates fitting of insert during bench work. It can be placed anywhere along the width of rib according to the choice of designer by changing the offset value.

Complete the sketch as shown below. Please note that radiuses applied at corners are equal but this constraint is not shown to unclutter the view.



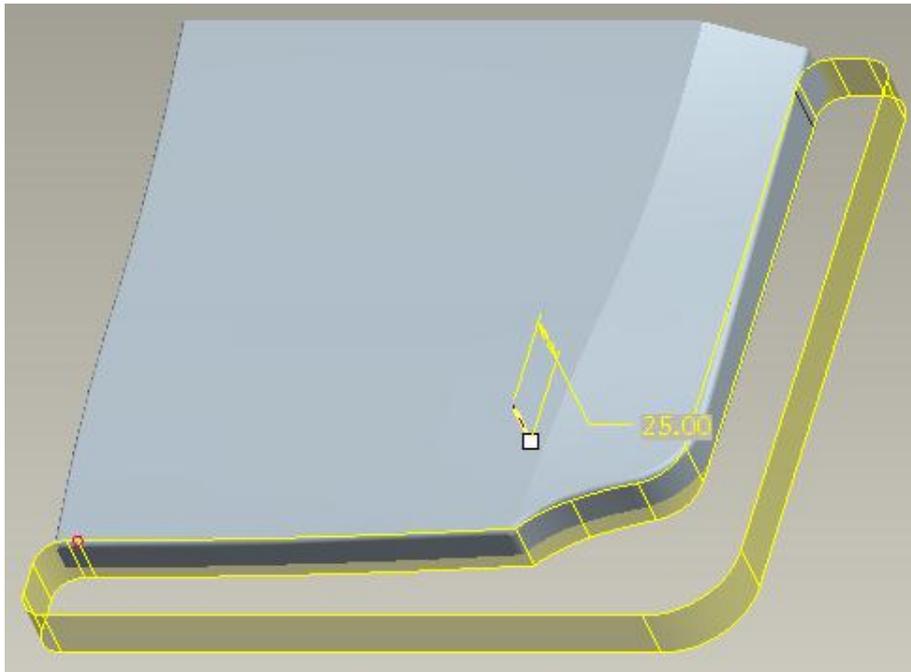
It is always good practice to apply generous radius at corners as it facilitates machining with larger cutters.

Pick  to save the sketch and exit.

Enter **25** for the extrusion depth as shown below.



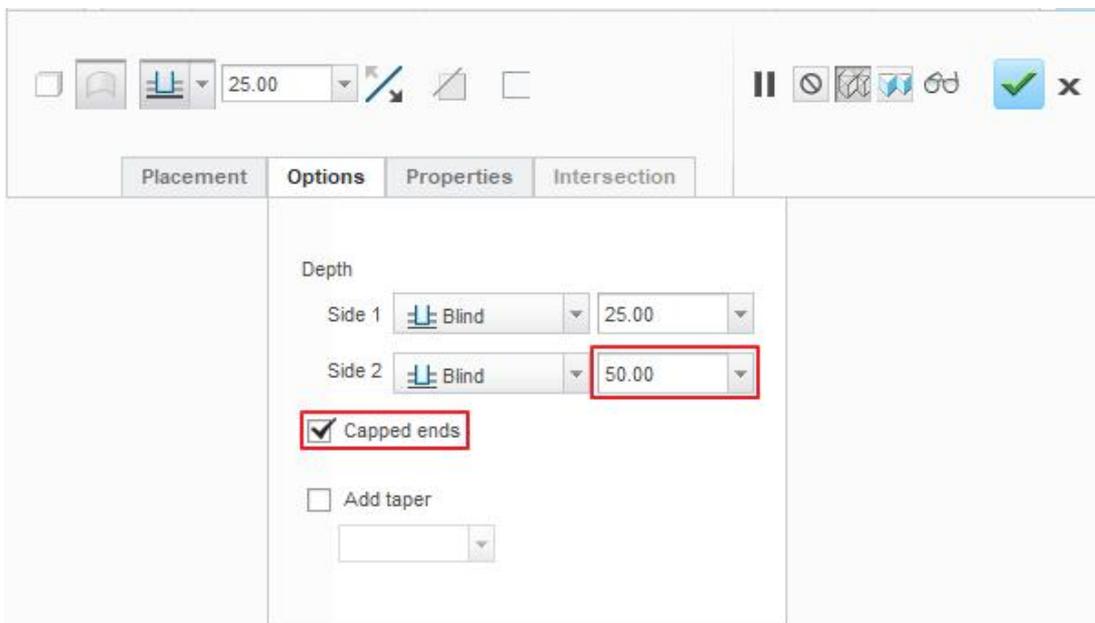
The model will appear as shown below.



This extrusion depth controls the depth of insert guiding (locating) in the TOPCOVER_CORE.

For the depth in second direction, enter an approximate number so that extruded surface goes beyond the reference part.

So pick **Options** tab and enter **50** for the extrusion depth for Side 2 as shown below.

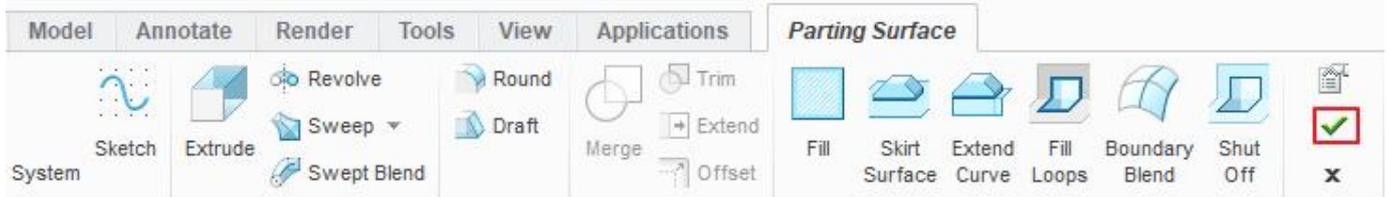


Also check the **Capped ends** option.

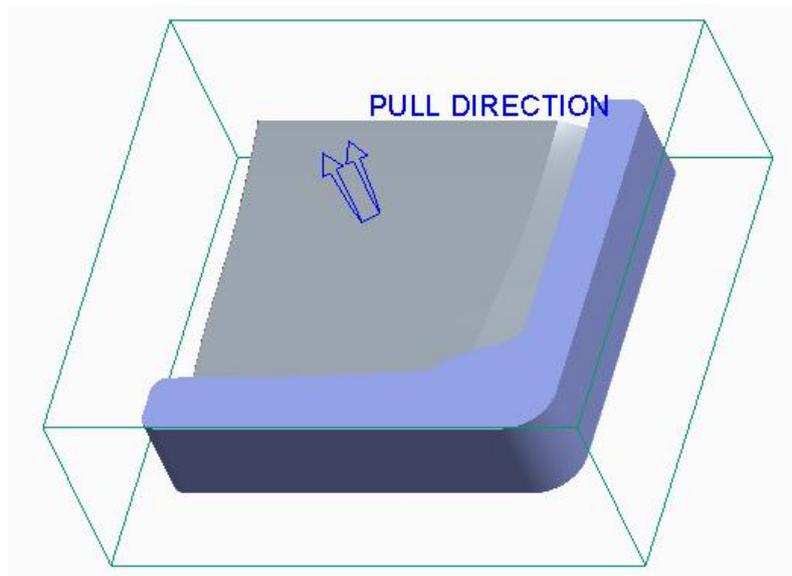
We have used the “Capped ends” option because if the parting surface does not completely intersect the volume, the split operation will fail.

Pick  or middle-click to complete the Extrude surface feature.

So pick  and it completes the definition of the parting surface.

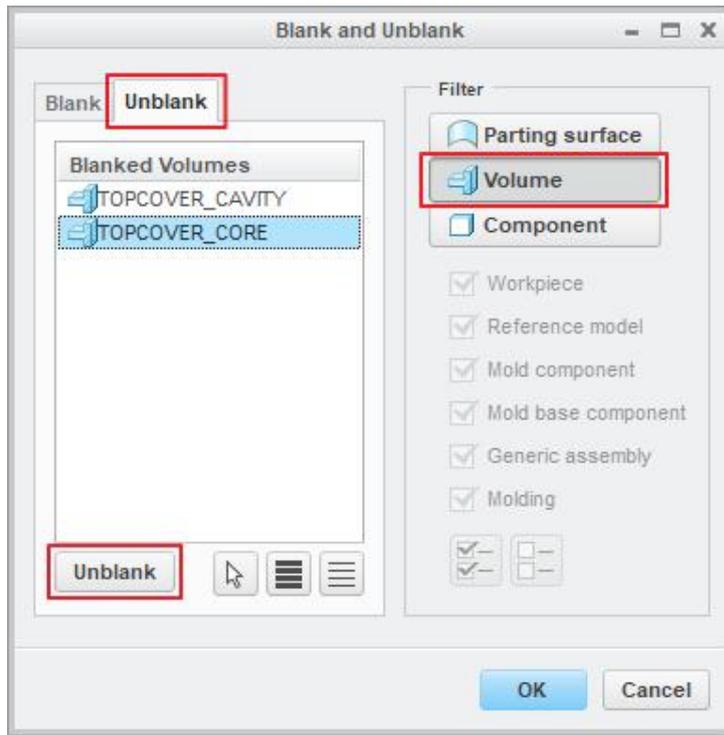


The parting surface will appear as shown below.

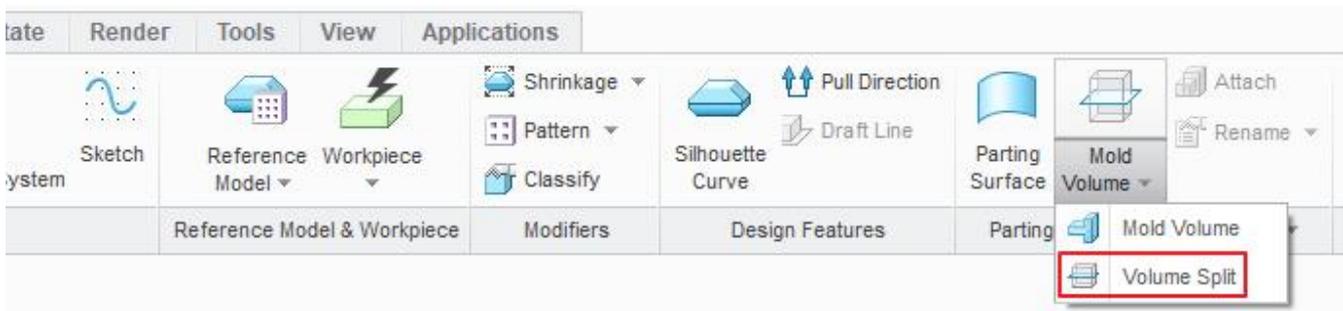


To split the core we will split the "TOPCOVER_CORE" volume and not the "TOPCOVER_CORE.PRT" part. As splitting a component does not alter its geometry but creates new that is not desired for the current situation.

First blank the reference part and workpiece and unblank the TOPCOVER_CORE volume.



Pick  Volume Split icon located on Mold tab.

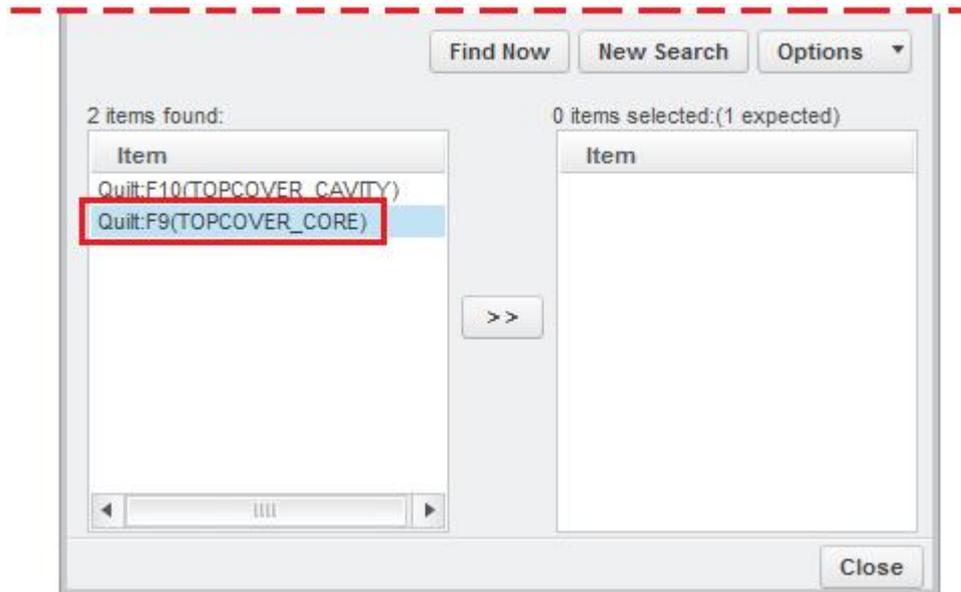


Select **One Volume > Mold Volume > Done**.

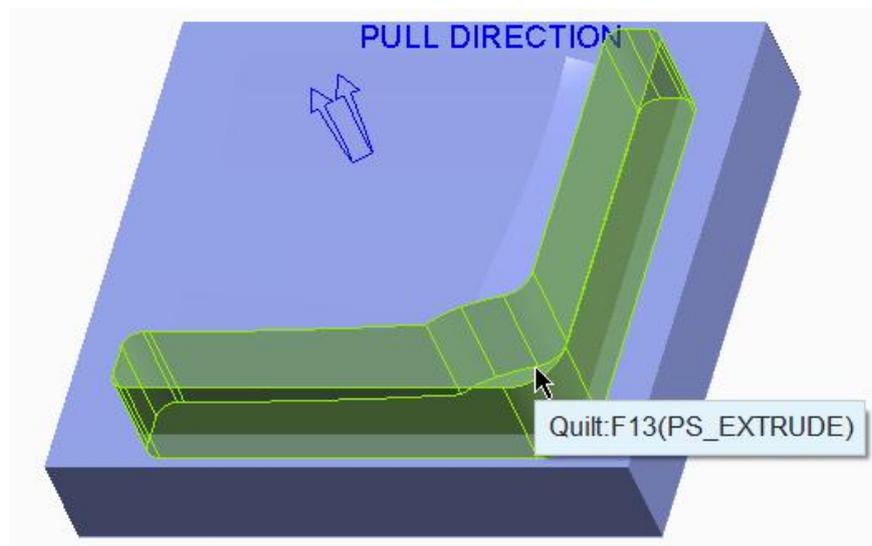


One volume split classifies the geometry to the one side of the parting surface and creates a new volume for that. The remaining geometry still belongs to the original volume that is being split.

In the Search Tool dialog box, pick the TOPCOVER_CORE quilt and then middle-click.

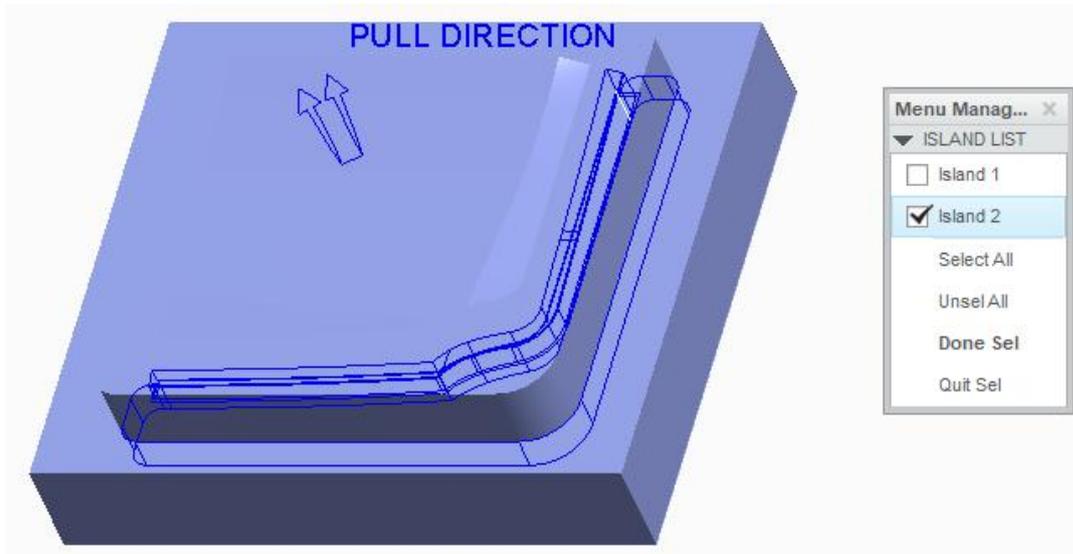


System will ask to select the parting surface for splitting the mold. So select the quilt of PS_EXTRUDE.



Pick or middle-click to complete the selection of surfaces.

In the ISLAND LIST menu check the **Island 2** box.



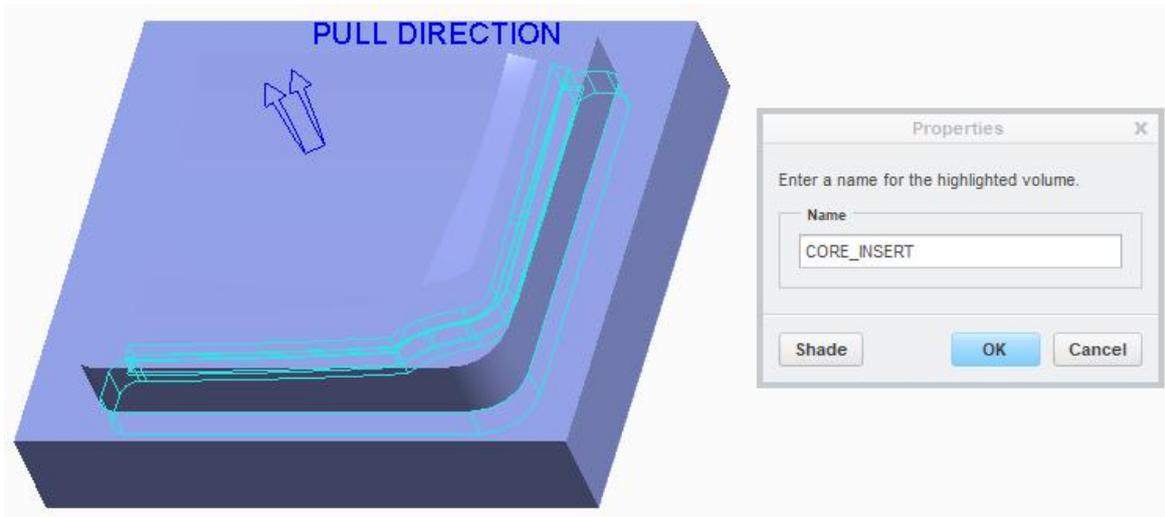
The split has recognized the geometry to the both sides of the parting surface and now is asking which side should be attached to the new volume. When mouse is placed over Island 1 or 2 in the menu the related geometry is highlighted on screen.

Pick **Done Sel** to accept current selection.

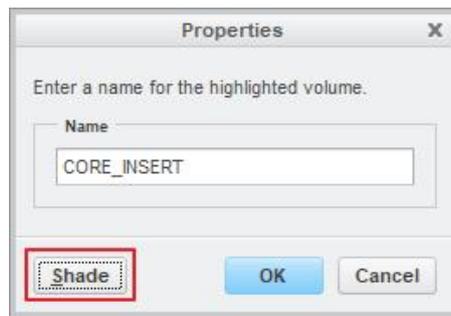
Pick or middle-click to exit the Split dialog box.

The system will highlight the calculated volume and ask to enter a unique name for it in the Volume Name dialog box.

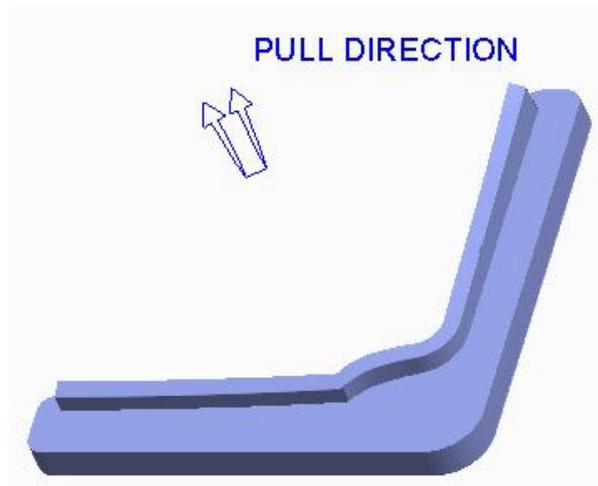
Enter **CORE_INSERT** as the name of the volume highlighted in the figure below.

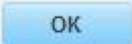


Pick to view the volume in shaded mode.

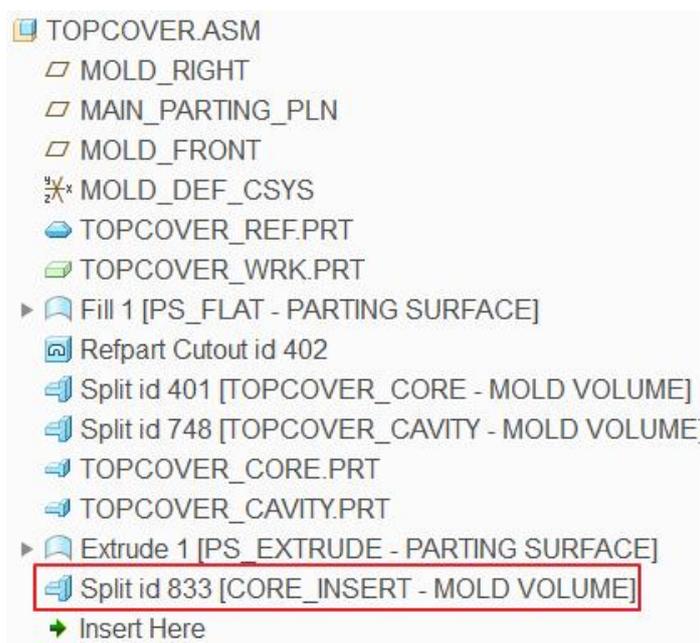


The split off volume will appear as shown below.



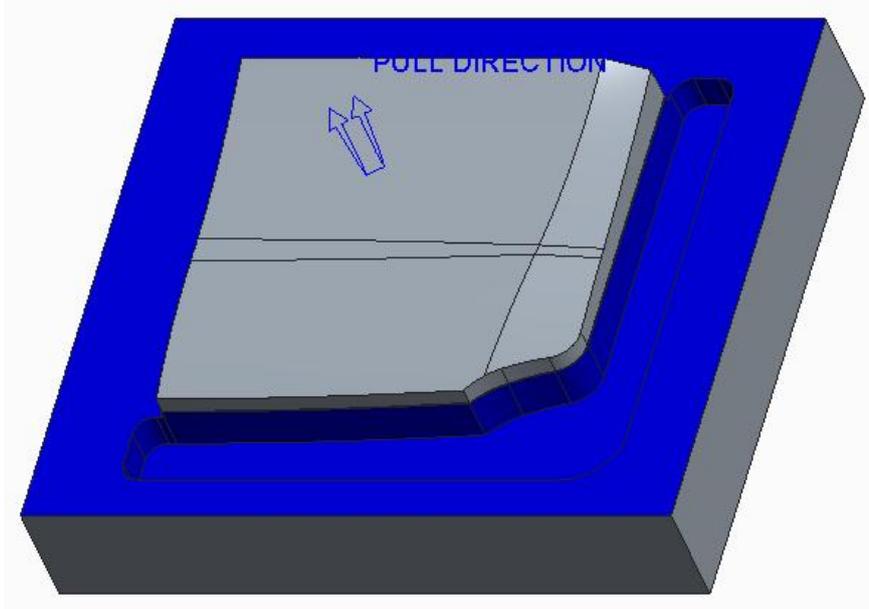
Pick  to complete the operation.

For a One Volume split operation, system has created a single feature in the model tree as shown below.



Pick  Mold Display icon on the View tab and blank all the Volumes and Parting Surfaces and unblank the TOPCOVER_CORE.PRT component.

Pick  icon on the Model tab to regenerate the assembly. The TOPCOVER_CORE will appear as shown below.



Split operation has cut the geometry from TOPCOVER_CORE volume, which was within the boundaries of the PS_EXTRUDE parting surface, to create the CORE_INSERT volume.

You should also notice that as the TOPCOVER_CORE volume changed, the TOPCOVER_CORE.PRT component updated accordingly.

An extracted component retains associativity with the parent volume; if the volume is modified, the component is updated when the mold model is regenerated.

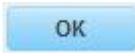
Splitting the insert volume into two

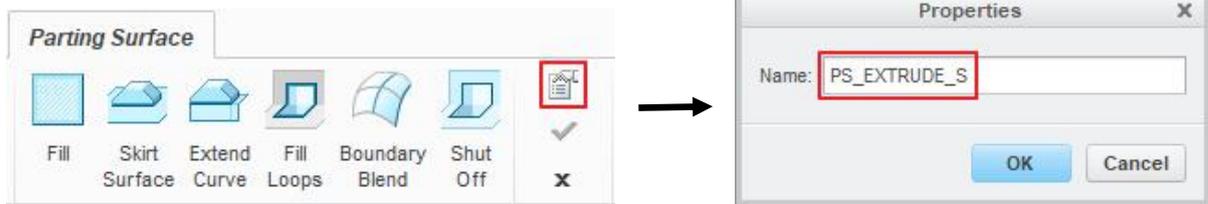
The insert CORE_INSERT is of very large size and odd shape. It will require a very large piece of metal to be machined. Therefore it will be beneficial to split it into two halves. To perform the split operation, we need another parting surface.

Before building the parting surface, blank the TOPCOVER_CORE and unblank the TOPCOVER_REF part.

To create the parting surface, pick Parting Surface icon () icon, located on Mold tab.

Pick  icon to give a name to this parting surface. (Note: if this icon is not active then pick anywhere in the graphics area, and it will become active.)

Enter **PS_EXTRUDE_S** as the name of the parting surface and select  .

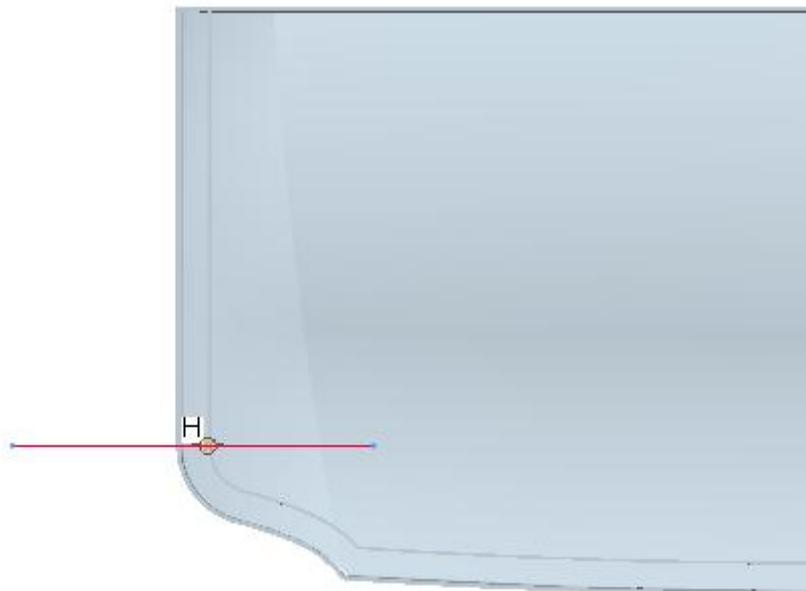


Pick  icon to invoke the Extrude Tool.

Pick  tab in the Placement panel.

For the sketching plane select .

After selecting suitable references, sketch a straight line approximately as shown below.



The size of the line is not important. The only requirement is that it should completely intersect the CORE_INSERT. For guidance we can unblank the parting surface PS_EXTRUDE or the CORE_INSERT volume. The location of this line relative to the insert depends upon the decision of the designer where to split it.

Pick  after completing the sketch.

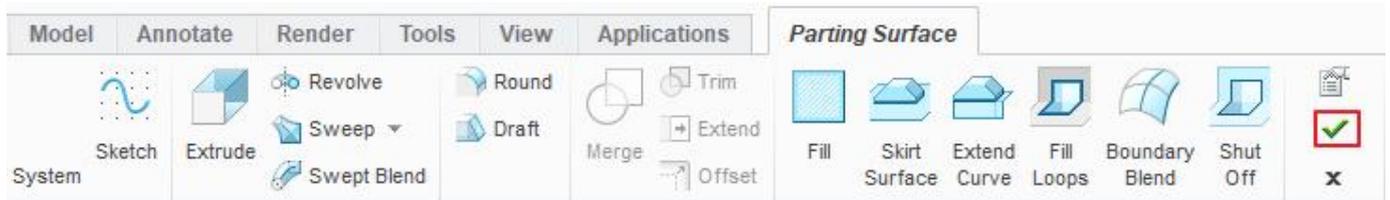
Enter **100** for the extrusion depth.

Change the depth option to Symmetric by picking the  icon in the Depth options list.



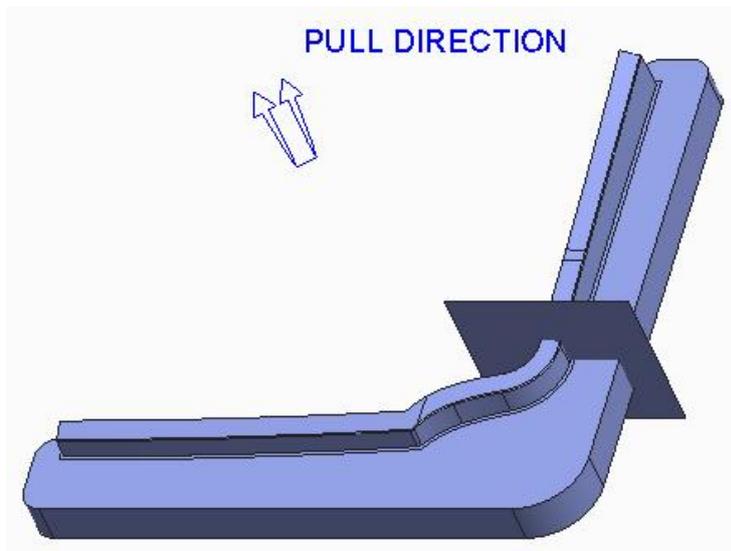
Pick  or middle-click to complete the surface feature.

So pick  and it completes the definition of the parting surface.



Now blank the reference part, workpiece and unblank the CORE_INSERT volume.

Notice that parting surface completely intersects the volume as shown below.

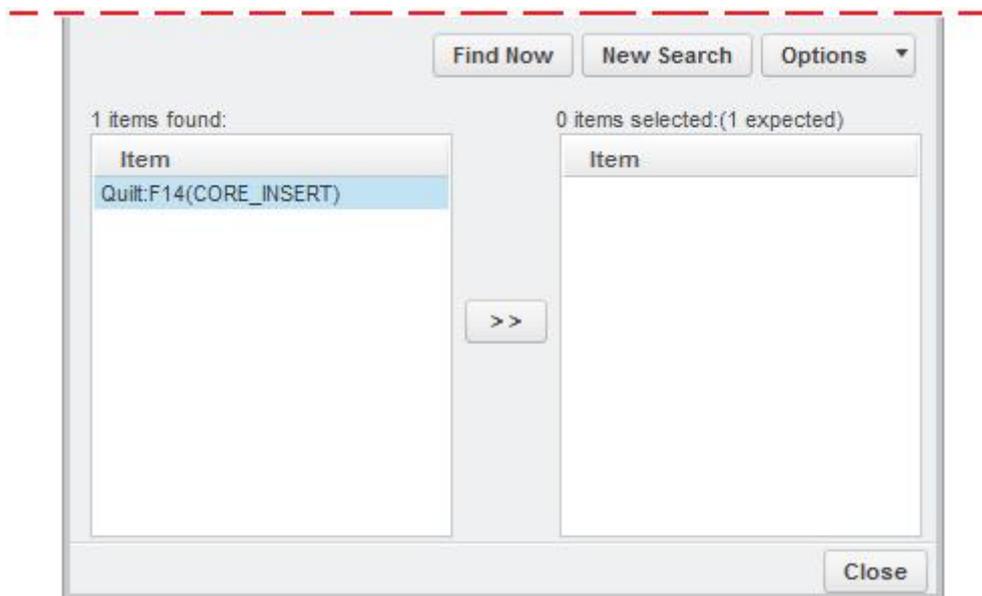


Pick  Volume Split icon located on Mold tab.

Leave the Two Volumes and Mold Volume options intact in the SPLIT VOLUME menu and pick **Done**.



In the Search Tool dialog box, pick the CORE_INSERT quilt and then middle-click.

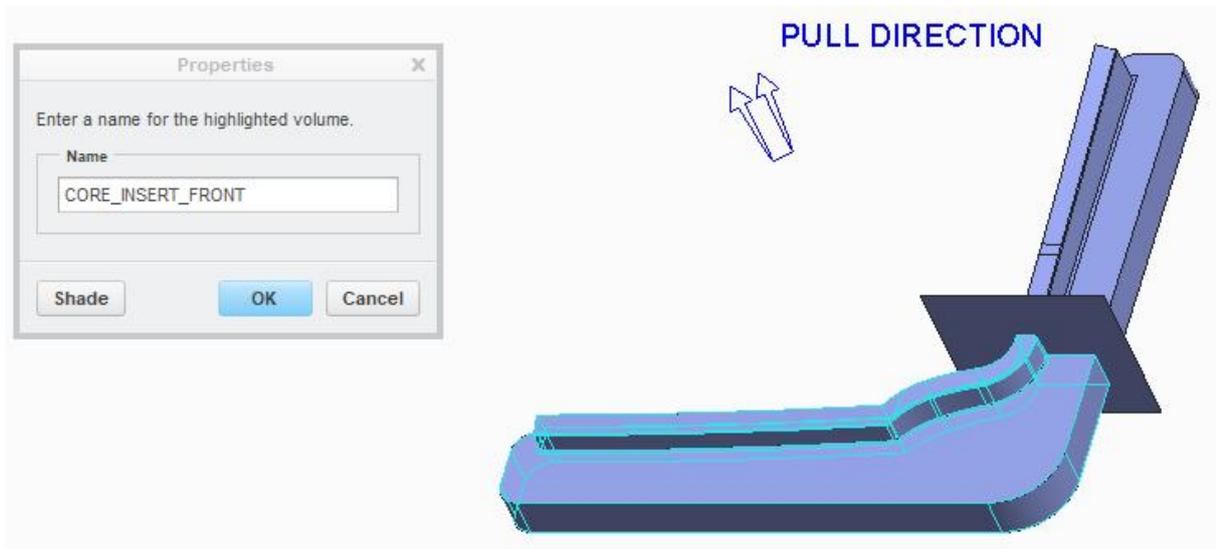


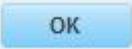
System will ask to select the parting surface for splitting the mold. So select the quilt of PS_EXTRUDE_S.

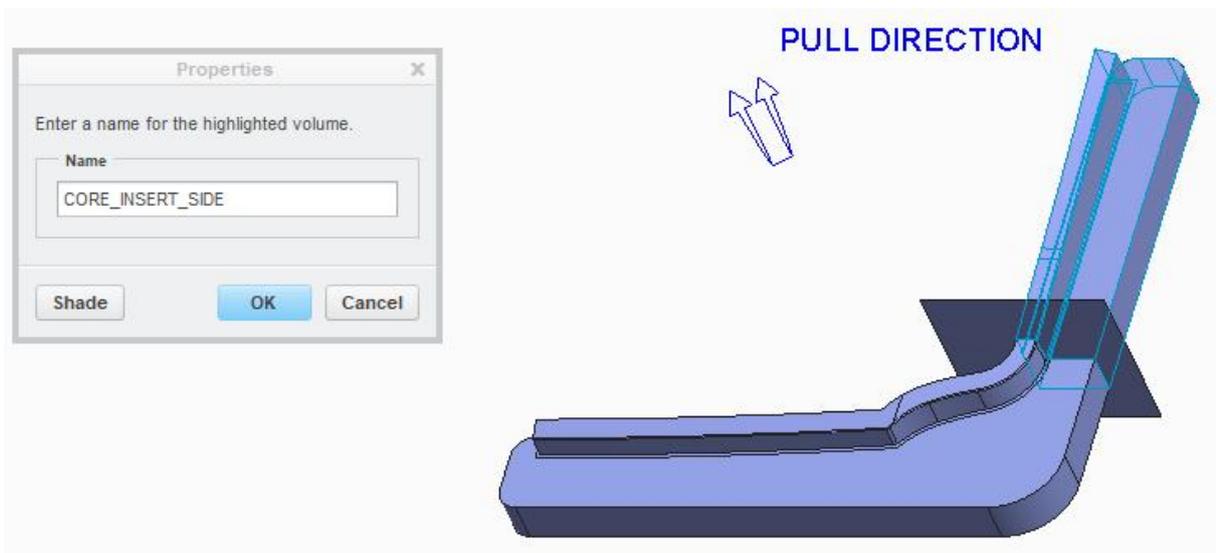
Pick or middle-click to complete the selection of surfaces.

Pick or middle-click to exit the Split dialog box.

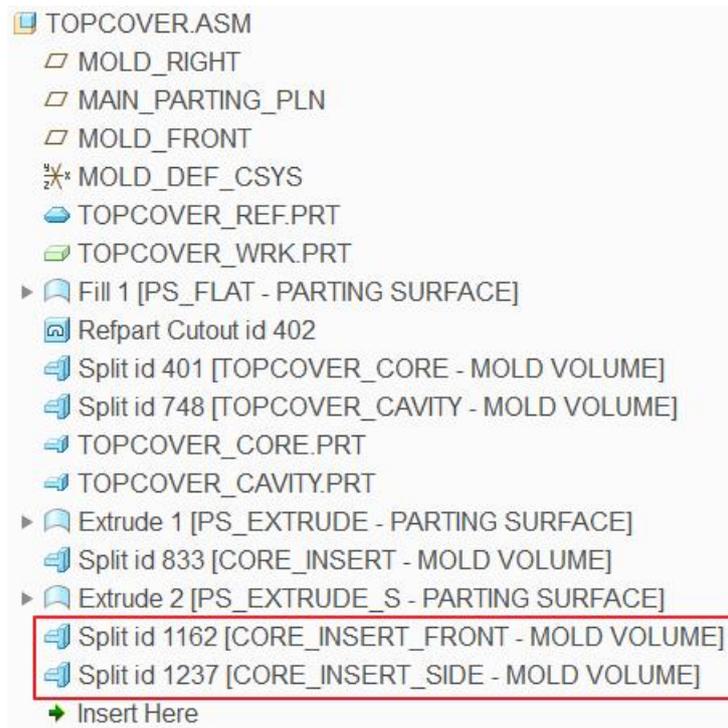
Enter **CORE_INSERT_FRONT** as the name of the volume highlighted in the figure below and pick



The system will highlight the second split volume. Enter **CORE_INSERT_SIDE** as the name of volume highlighted in the figure below and pick 

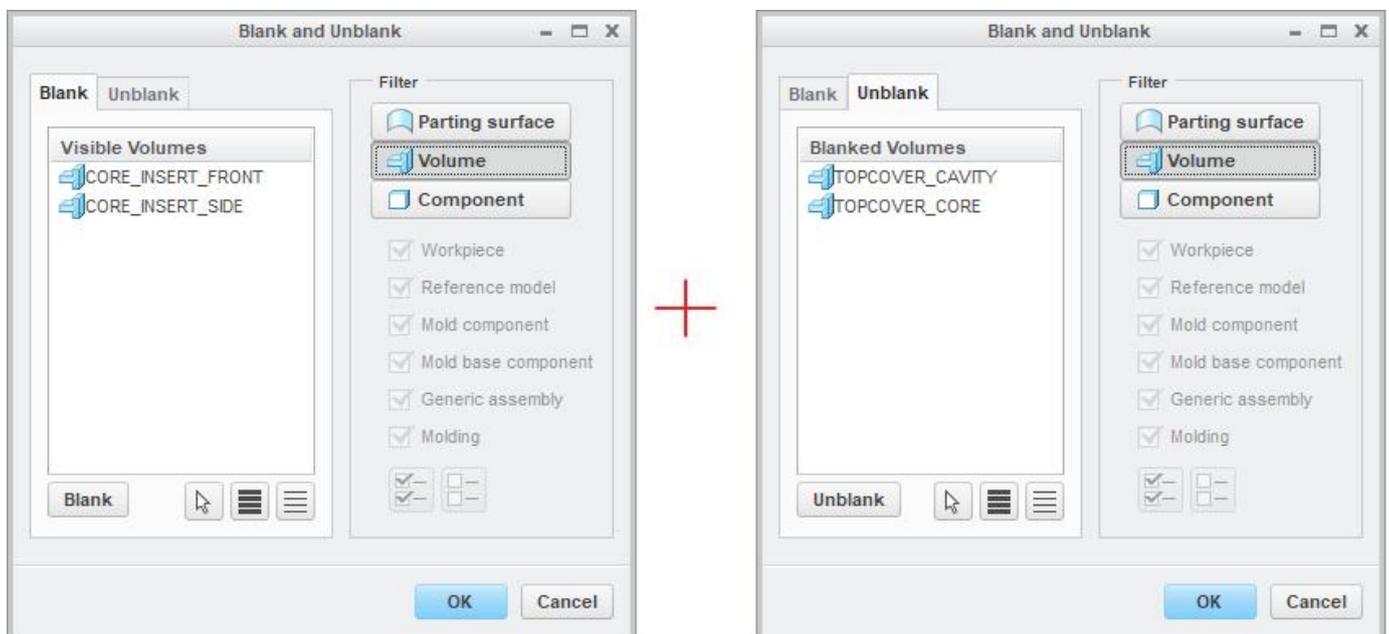


For the Two Volumes split operation, system has created two features in the model tree as shown below.

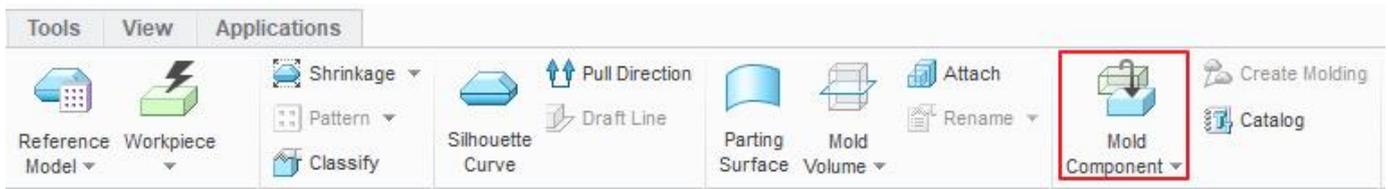


Pick  icon on the View tab and blank the PS_EXTRUDE_S Parting Surface.

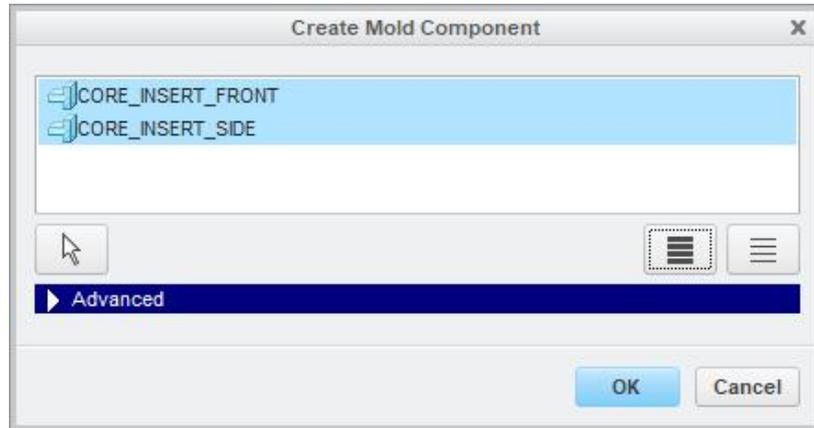
If you activate Volume filter, you will notice that now there is no CORE_INSERT volume in Blank and Unblank dialog box, as it has been consumed by the split operation for creating the two new inserts as shown below.

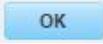


To extract the mold components, pick  icon on the Mold tab.



Pick  to select both the volume listed in the dialog box.

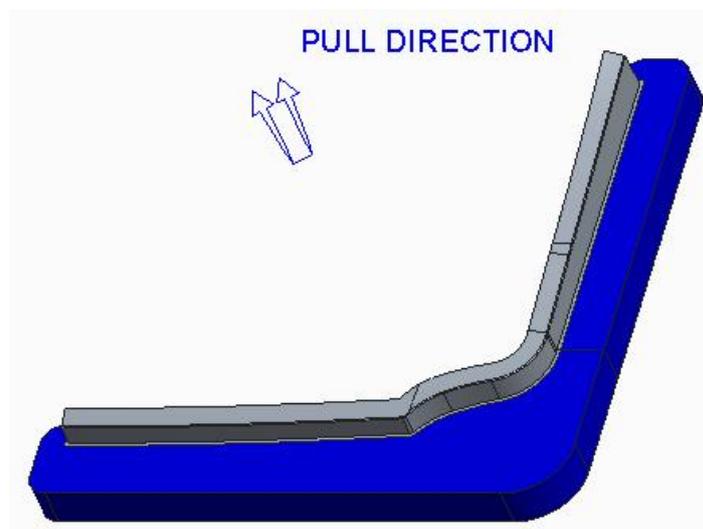


Pick  to proceed with current selection.

Two new components will appear in the model tree; CORE_INSERT_FRONT and CORE_INSERT_SIDE.

Extracted components can be used for generating the MCD data in Creo NC for driving the CNC machine tools or can be analyzed just like any other regular Creo part.

Extracted components will appear as shown below. Again notice the different colors of surfaces for “metal to plastic” contact and for “metal to metal” contact surfaces.



Select **File > Save** to save the work done so far.

EXERCISE 2

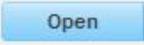
In this exercise we will learn how to retrieve and manipulate a mold model.

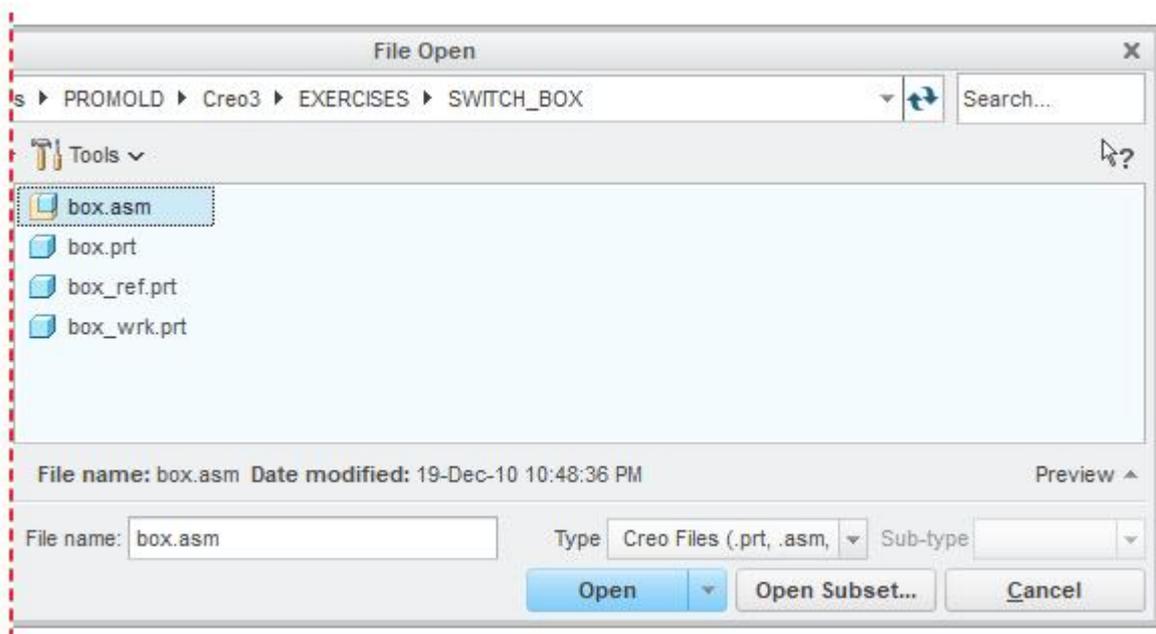
Topics covered in this exercise are as follows

- ❖ Retrieving a mold model
- ❖ Redefining the Workpiece
- ❖ Modifying a parting surface

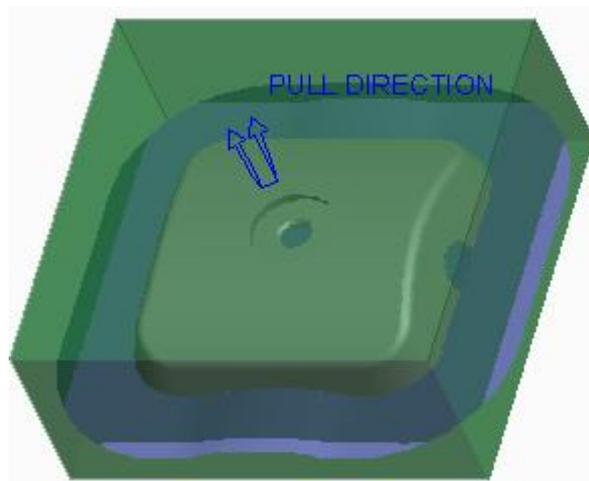
➔ Retrieving a mold model

Set the working directory to the SWITCH_BOX folder and pick **File > Open**

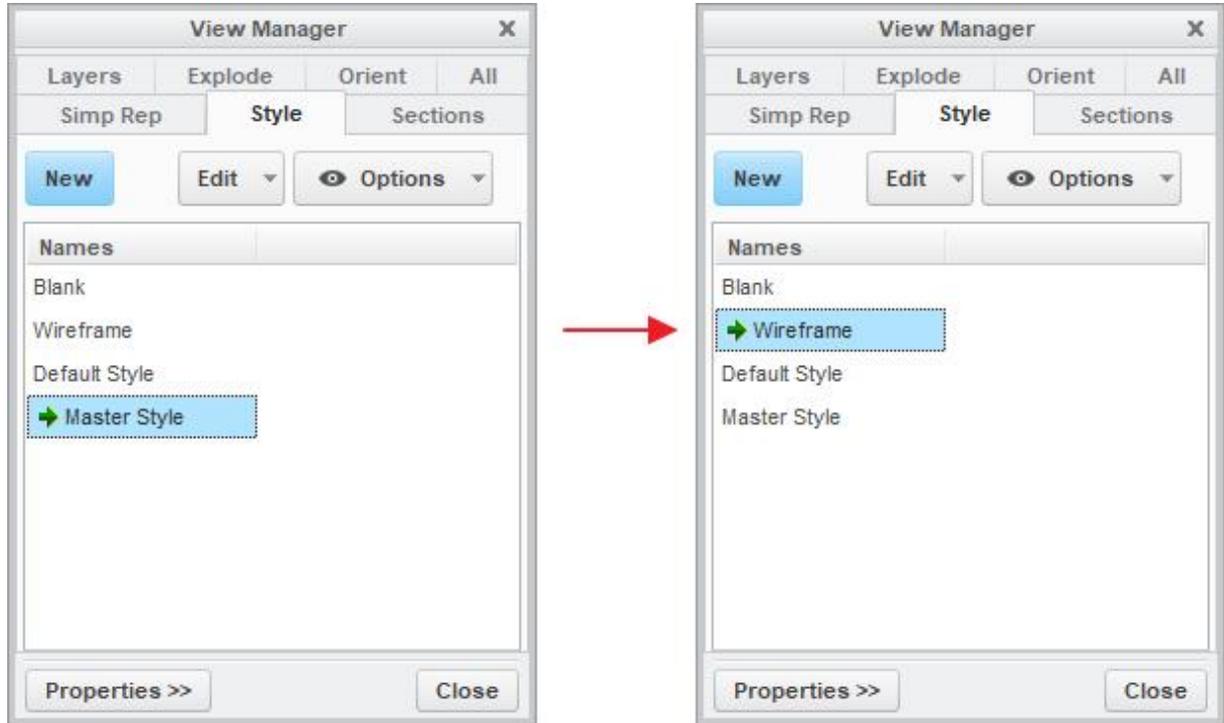
In the File Open dialog box, select the **box.asm** then pick .



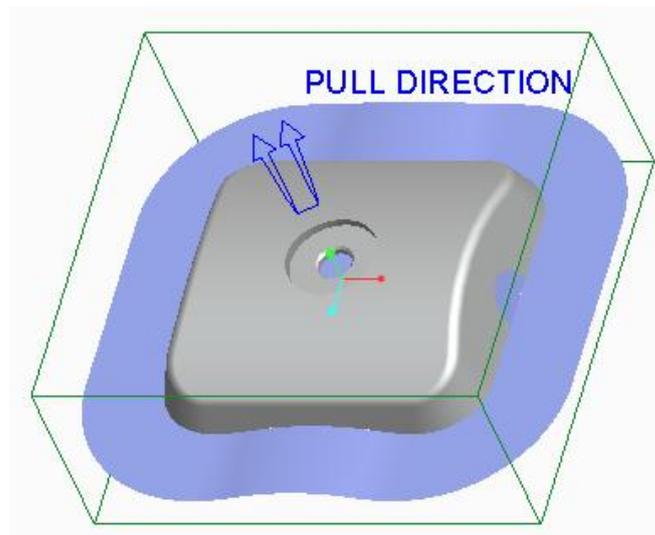
Mold model will appear as shown below.



For better visualization it is good to view the workpiece in Wireframe mode. So select **View Manager** icon () on the View tab to launch the View Manager Dialog box and double-click the predefined style state **Wireframe**.

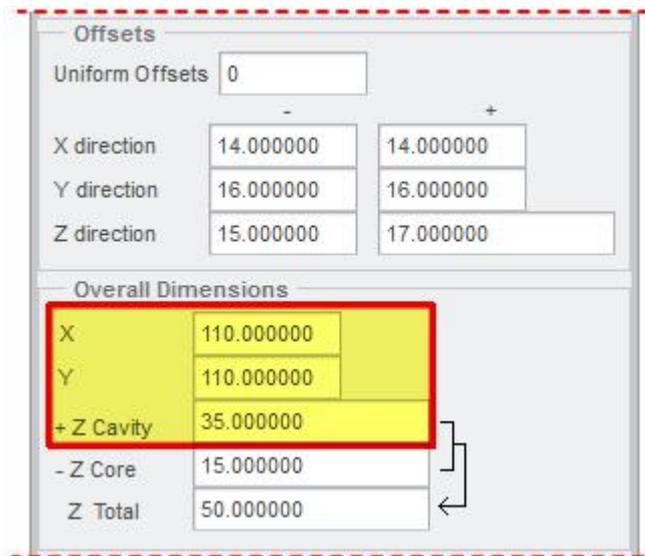


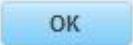
Pick  to exit the dialog box. The model will appear as shown below.



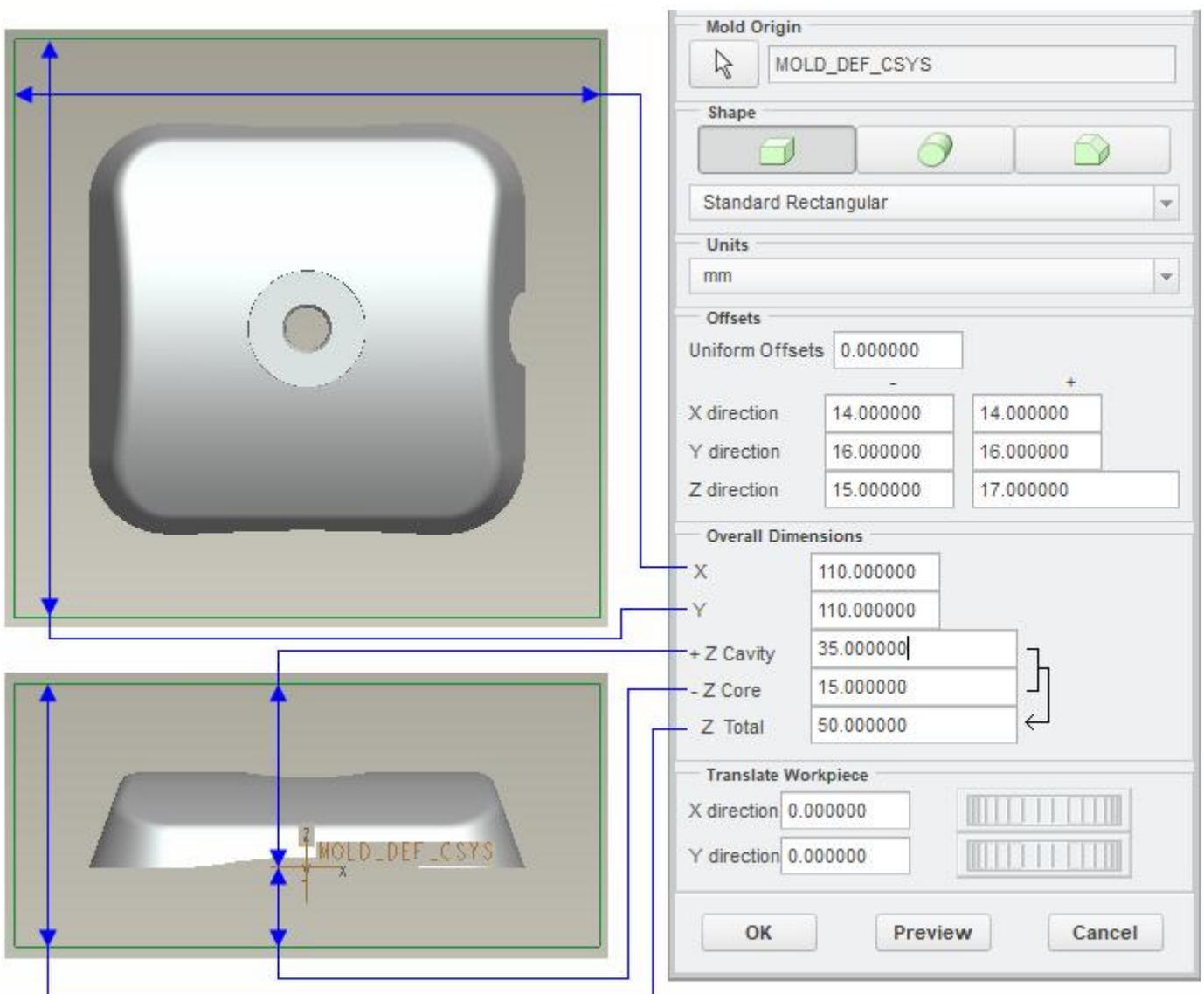
 **Redefining the Workpiece**

Now we will redefine the workpiece and change the Overall Dimensions to the nearest standard values.



Pick  after changing the Overall Dimensions.

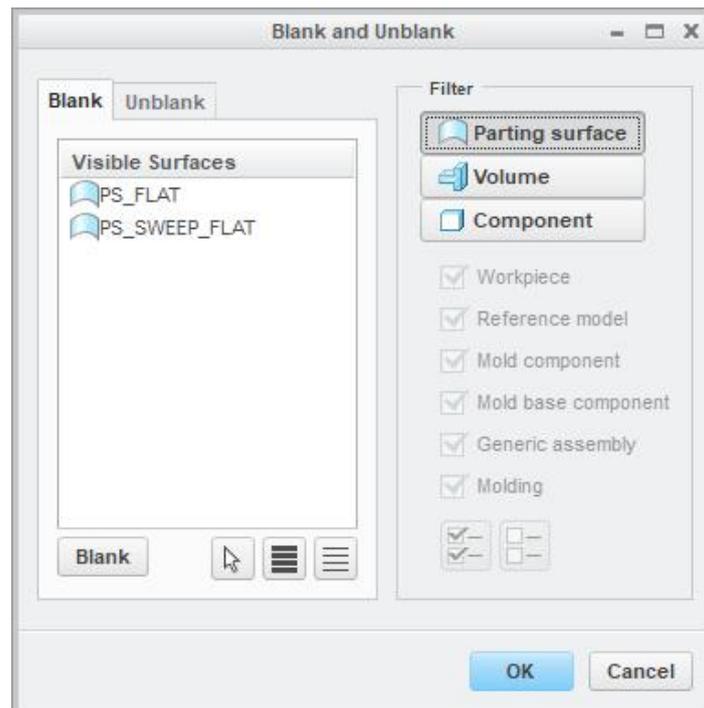
The concept of overall dimensions can be explained by the following figures.



You should notice that “+Z Cavity” and “-Z Core “ dimensions are measured from the coordinate system, selected as the Mold Origin during workpiece creation.

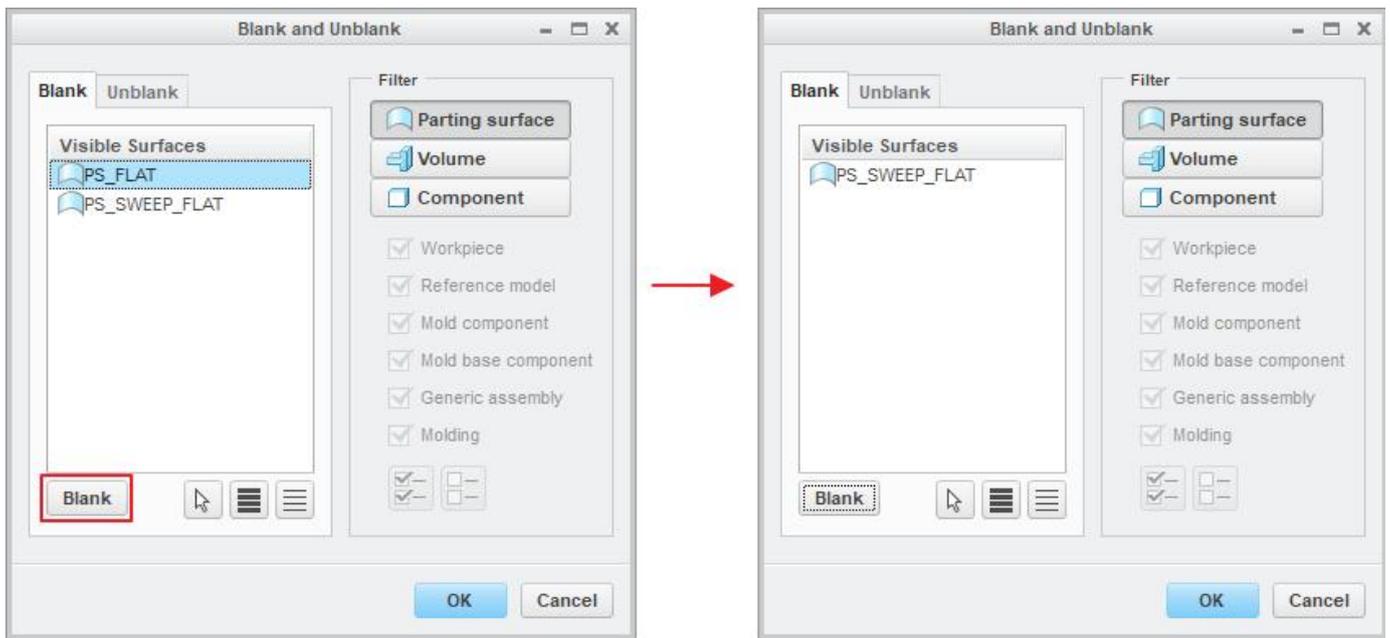
➔ Modifying the Parting Surface

Pick  icon on the View tab and Blank and Unblank dialog box will appear. Activate the Parting Surface filter by picking  Parting surface tab. Dialog box will appear as shown below.

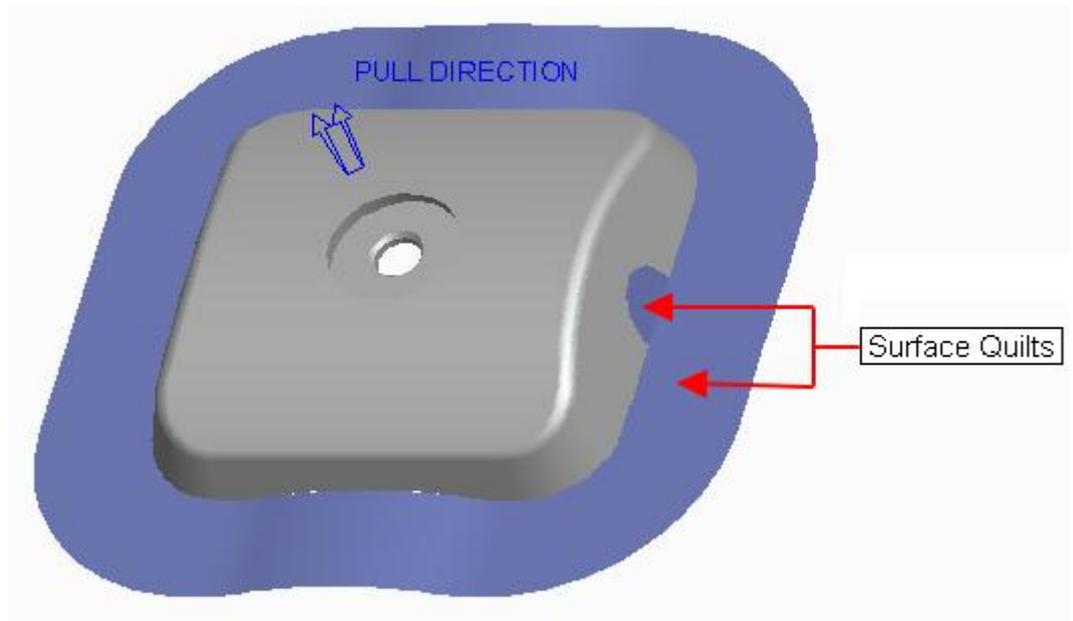


Notice that there are two parting surfaces listed in the dialog box. We will manipulate the PS_SWEEP_FLAT, so blank the PS_FLAT parting surface.

Select PS_FLAT and pick  tab.



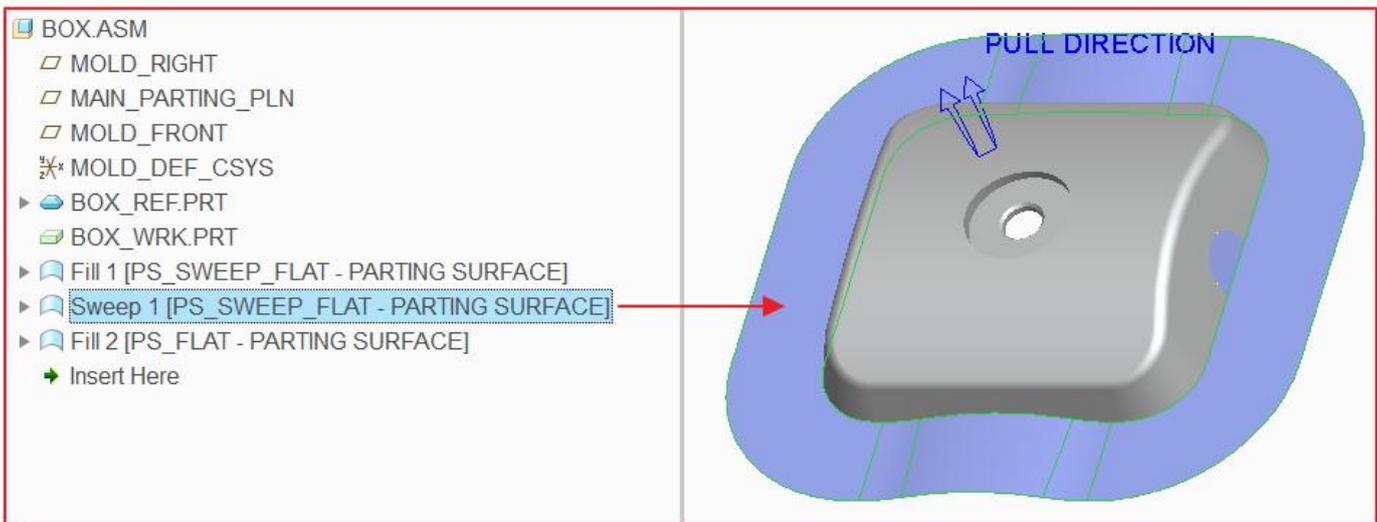
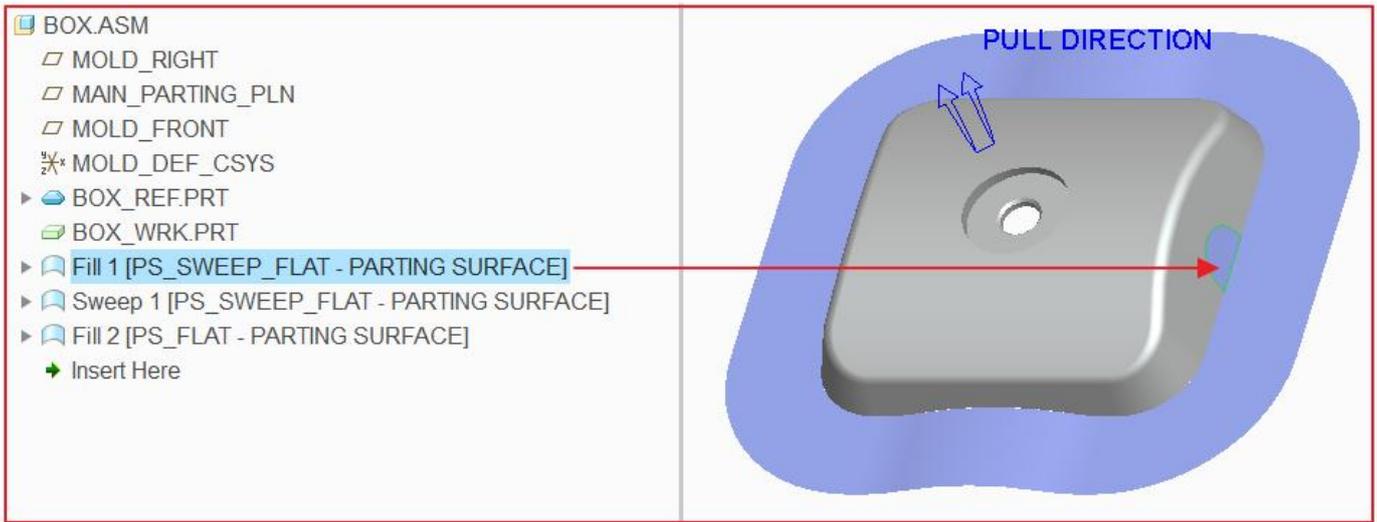
Pick **OK** to exit the dialog box and notice that there are two surface quilts displayed on the screen as shown below (workpiece is not shown).



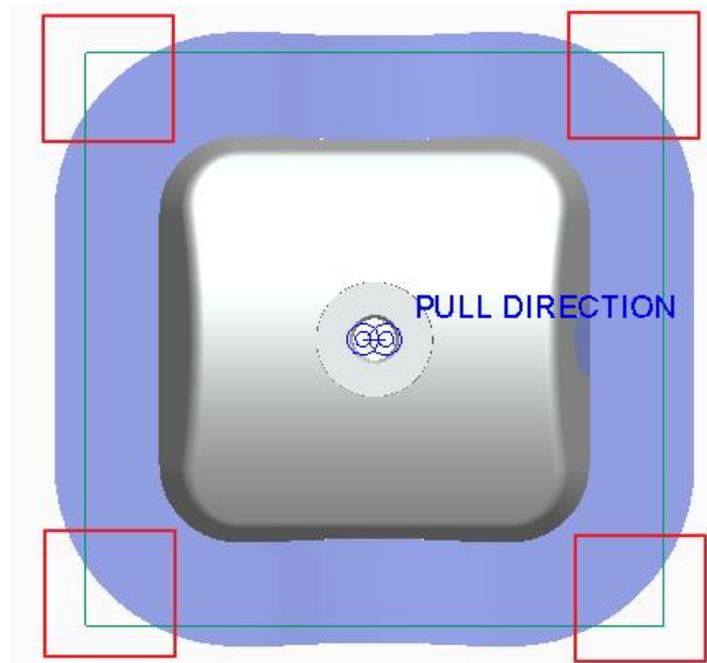
Both of these quilts are the result of different features and belong to the PS_SWEEP_FLAT parting surface.

A parting surface in a mold assembly is not necessarily a single feature; rather, it is a name of a set of features. So it may consist of either a single feature or multiple features.

If you pick any of the quilts on screen, system will highlight the corresponding feature in the model tree as shown in the figures below (workpiece is not shown).

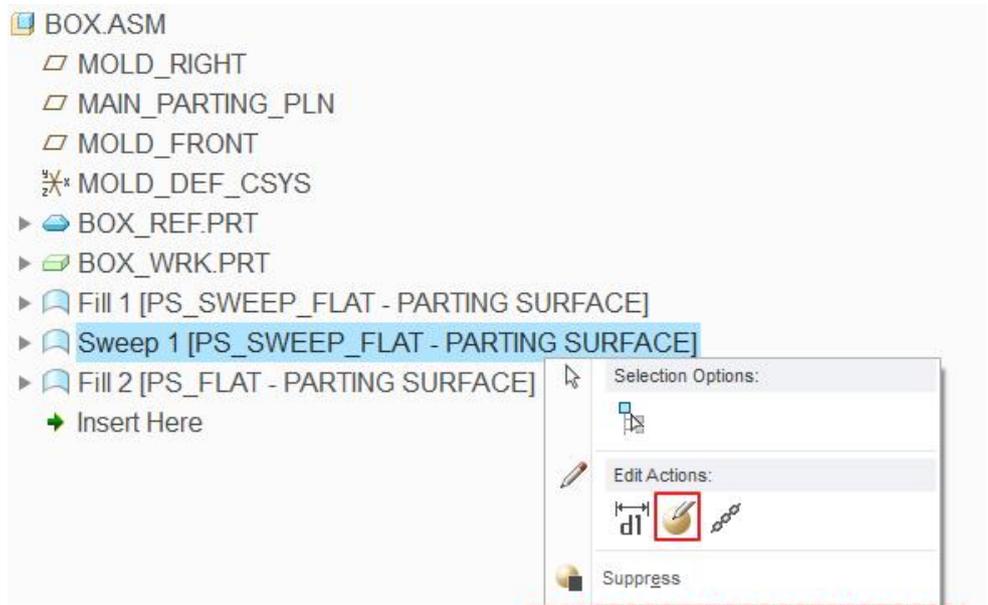


If you look from the Top view, you will notice that swept surface does not completely intersect the workpiece as elaborated below.



Now we will redefine this surface feature so that it completely intersects the workpiece.

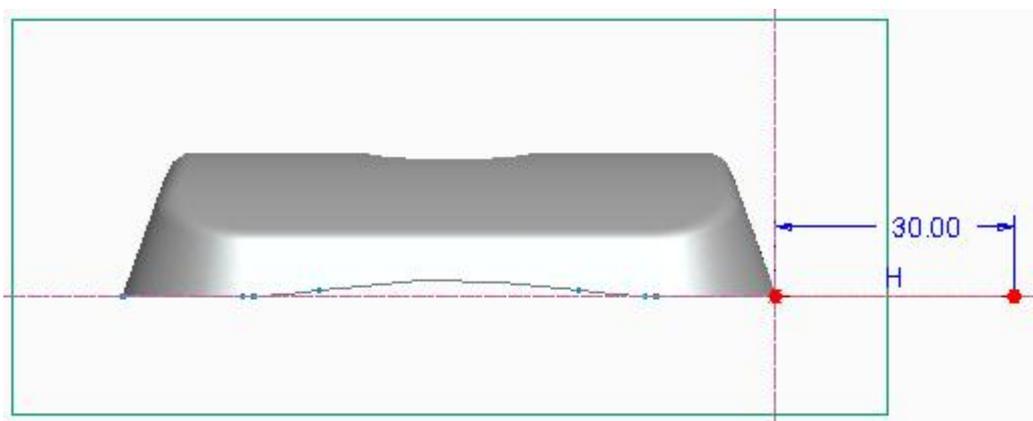
So pick the swept surface feature and right-click then select  in the pop up menu.



Pick  tab on the dashboard.



Modify the length of the sketched line to **30** as shown below.



Pick  to save the sketch and exit.

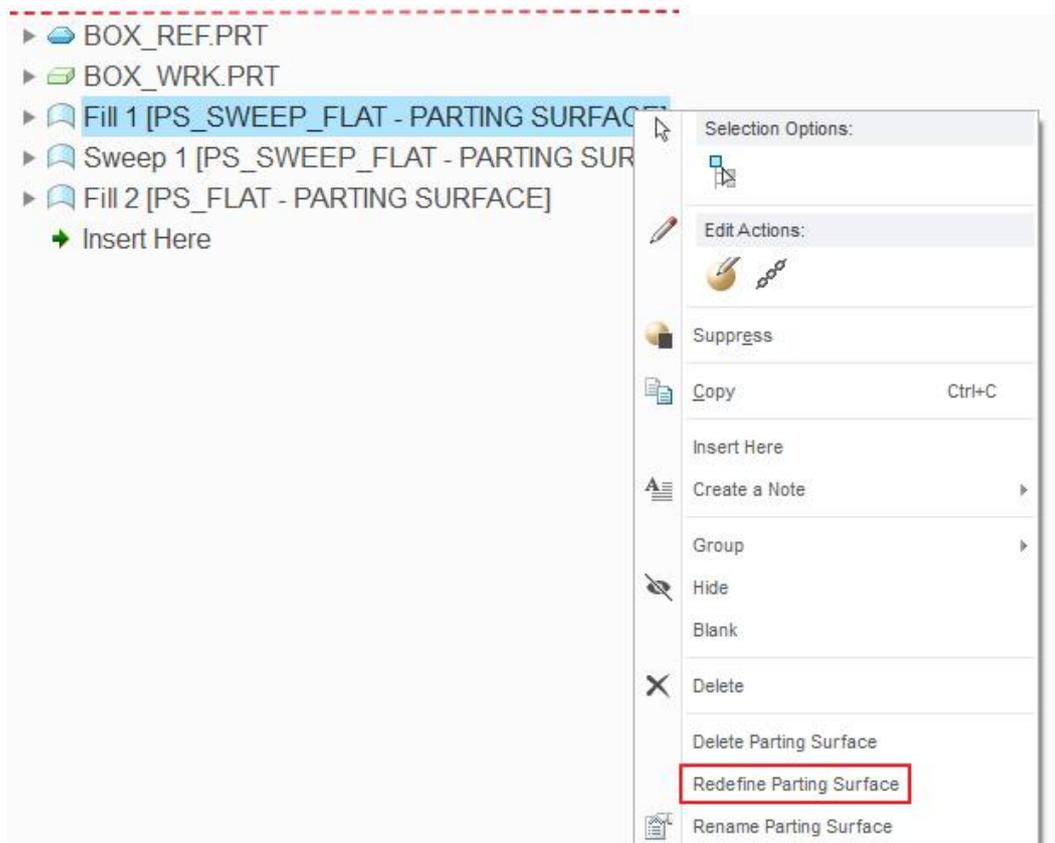
Pick  or middle-click to apply the change to the surface feature. The model will appear as shown below.



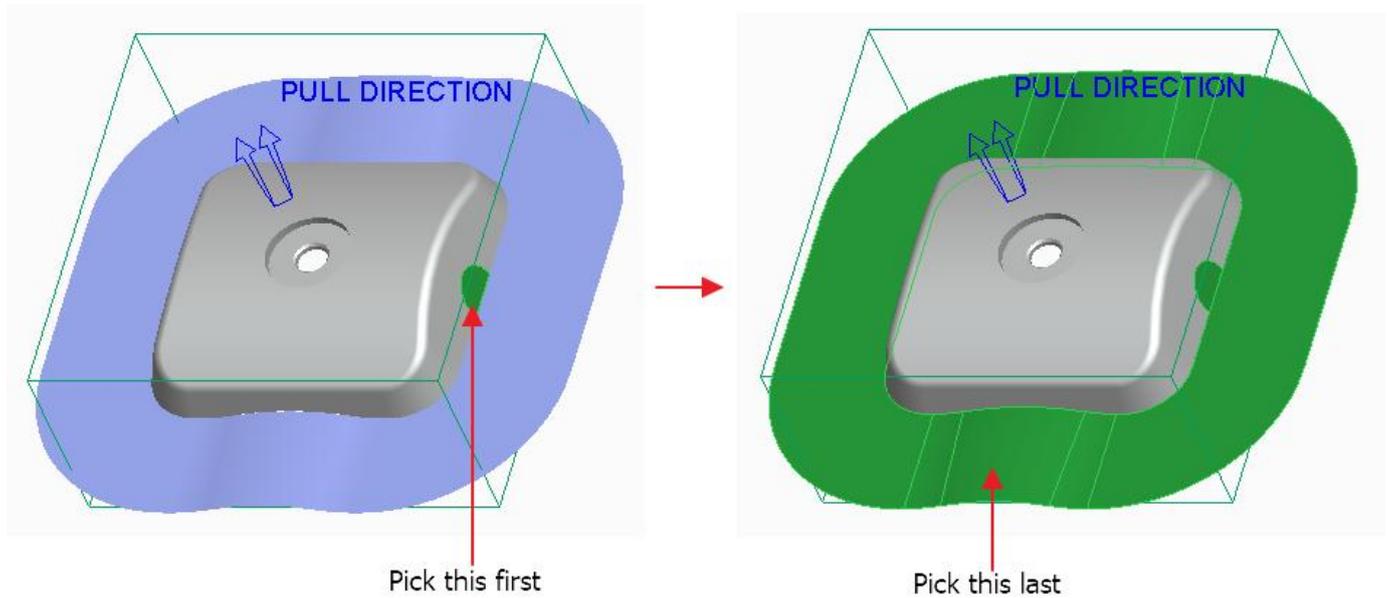
Notice that now parting surface completely intersects the workpiece.

Now we will merge the quilts of PS_SWEEP_FLAT parting surface together.

So right-click the first feature of **PS_SWEEP_FLAT** parting surface and pick **Redefine Parting Surface**.



First pick the Fill surface and then, while holding down the Ctrl key, pick the swept surface as shown below.



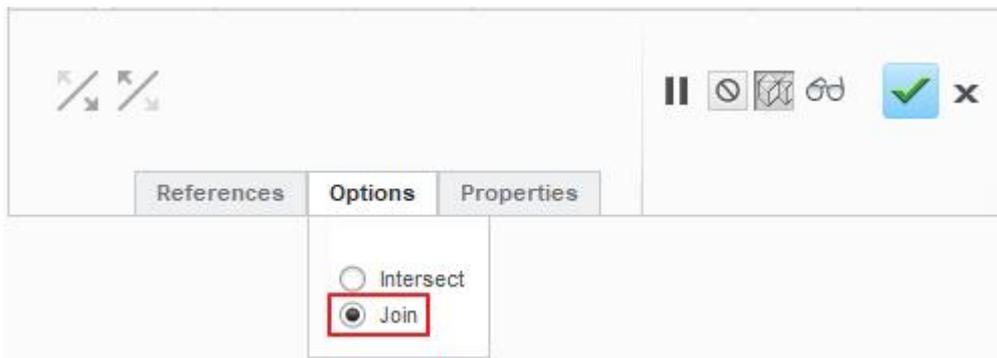
While selecting you should make sure that the first surface in the definition of the Parting Surface is selected first.

Pick merge icon () on the Parting Surface tab as shown below.



Keep in mind that Merge icon will become active only after picking the appropriate references.

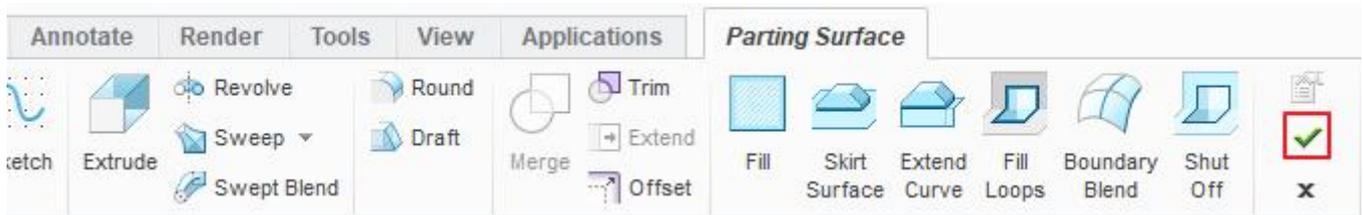
Pick **Options** tab and change the merge type to **Join**.



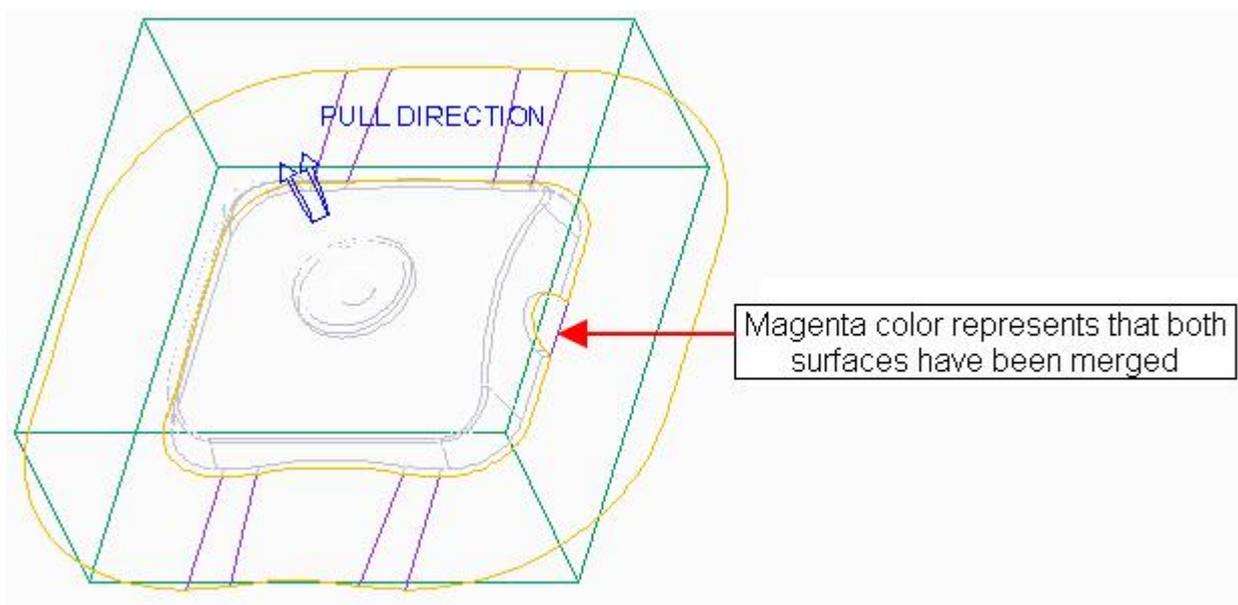
Join option is used when two surfaces have a common edge. Join option is faster than Intersect option.

Pick  or middle-click to complete the merge feature.

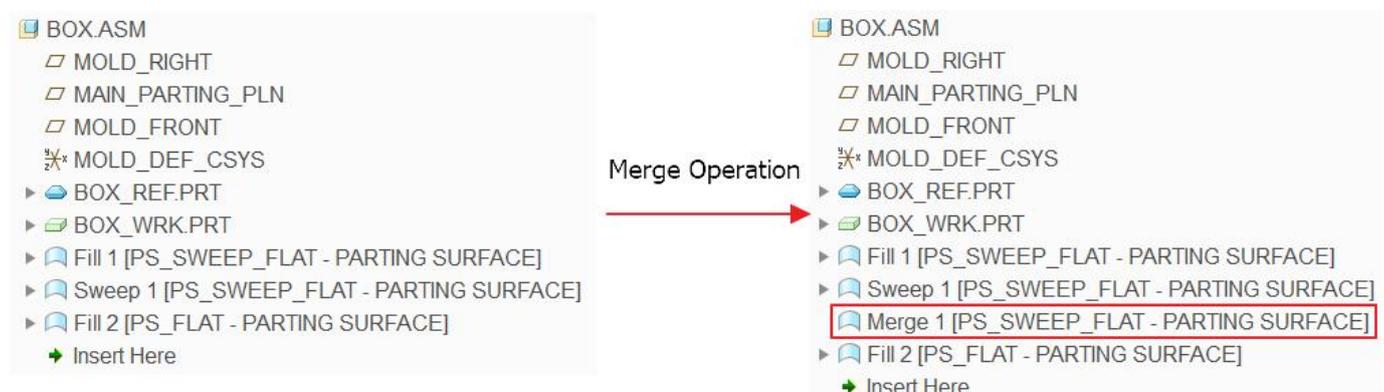
Pick  and it completes the manipulation of the parting surface.



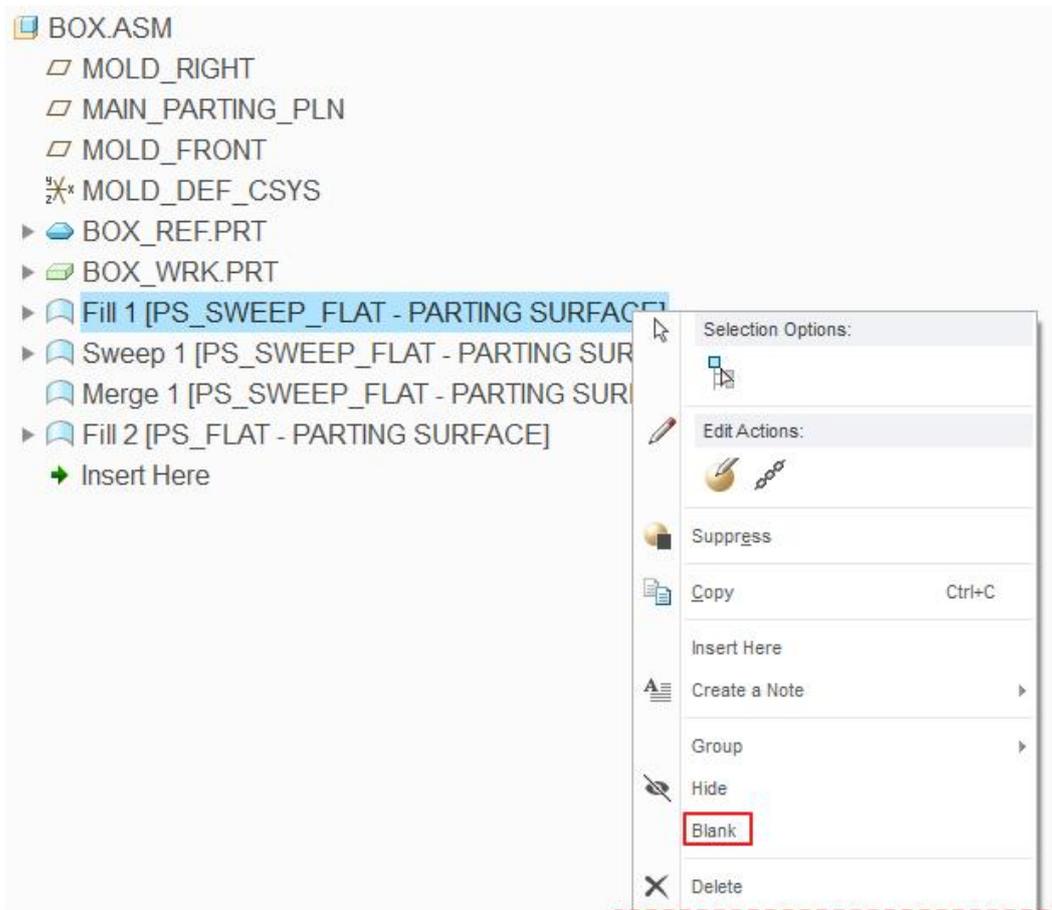
Notice that the joint between flat and swept surface appears magenta (in wireframe mode) which means that both surfaces have been merged into a single quilt.



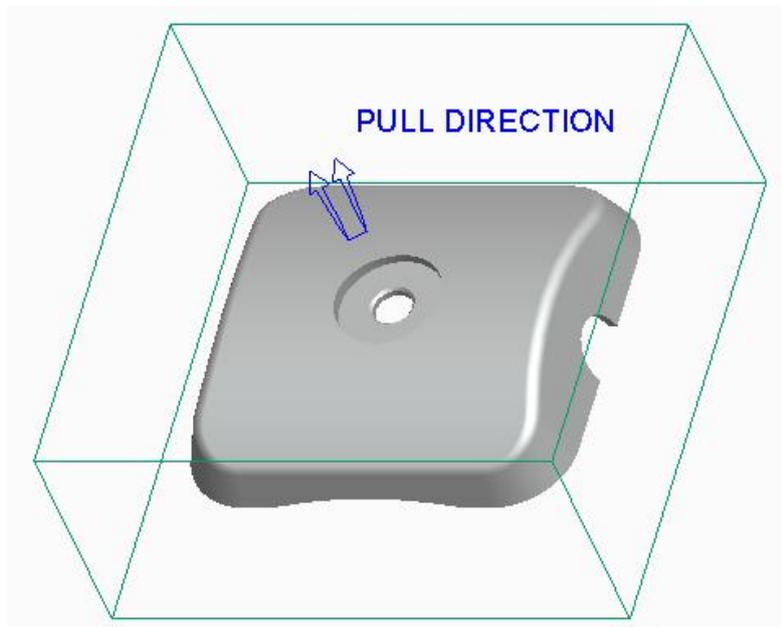
If you look in the model tree, you will notice that Merge feature has been created just after the sweep feature and before the second parting surface.



Now right-click the first feature (fill surface) of the PS_SWEEP_FLAT parting surface and pick **Blank**.



After blank operation, mold model will appear as shown below.



Now this mold model can be split easily by following the procedure in previous exercise.

Select **File > Save** to save the work done so far.

Note: The parting surface in this exercise has been defined using very simple techniques for easy understanding. In later exercises, more advanced techniques will be shown.

2. PARTING SURFACES

Further material is not included in the free download. You can buy the complete tutorials at <http://www.caeuniversity.com/buy/>. Thanks for your review of the training material.