

Chapter 16



Introduction to Injection Mold Design

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the component required in mold design*
- *Analyze the part fill*
- *Create core and cavity for the model*
- *Specify location for gates and runners*
- *Create mold base for the component*
- *Create cooling channel for mold*
- *Create ejection system for mold*

INTRODUCTION TO INJECTION MOLD DESIGN

Injection molding is a manufacturing process in which parts are created by injecting molten material into the mold. You can process thermoplastic as well as thermosetting polymers to create parts in injection molding machine.

An injection mold is an assembly of parts containing an impression into which plastic material is injected and cooled. This impression is formed by two parts of the mold:

- (i) Cavity
- (ii) Core

In NX 11.0, you can design a mold by using the tools available in the Mold environment.

INVOKING THE MOLD ENVIRONMENT

Before starting NX Mold Wizard, you need to add NX Mold Wizard library to NX. After that, you can use standard parts and material library to create a mold. Start NX by double-clicking on its shortcut icon. After the system has loaded all the required files to start NX, the initial interface will be displayed, as shown in Figure 16-1.

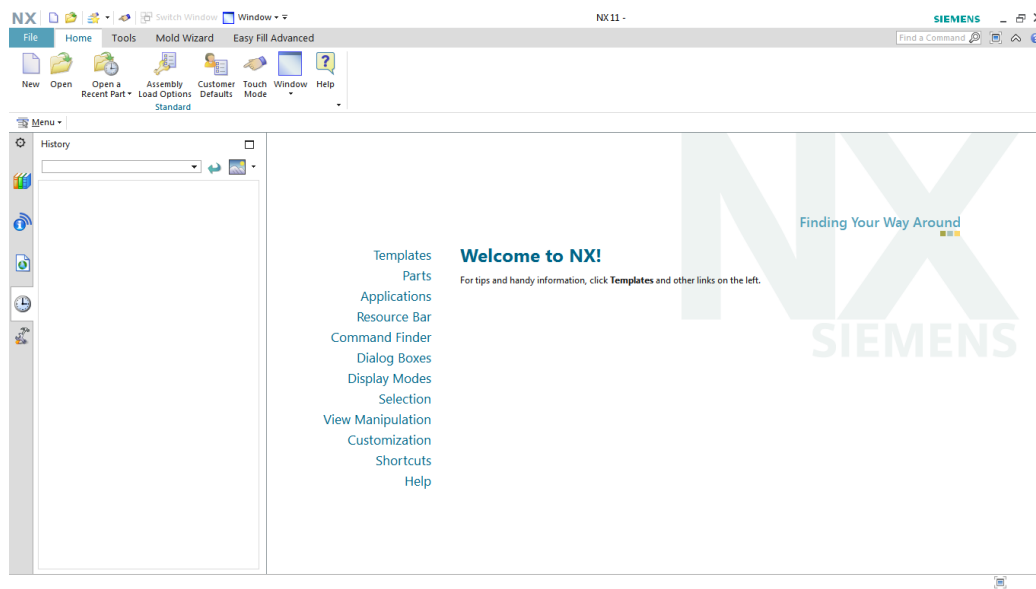


Figure 16-1 The initial interface of NX

Next, choose **File > Open** from the **Ribbon**; the **Open** dialog box will be displayed, as shown in Figure 16-2.

Now, select the file and choose the **OK** button; the file will be opened in the Modeling environment, as shown in Figure 16-3.

To invoke the NX Mold Wizard, choose the **Application** tab and then choose the **Mold** tool from the **Process Specific** group; the **Mold Wizard** tab will be displayed next to the **Application** tab, refer to Figure 16-4.

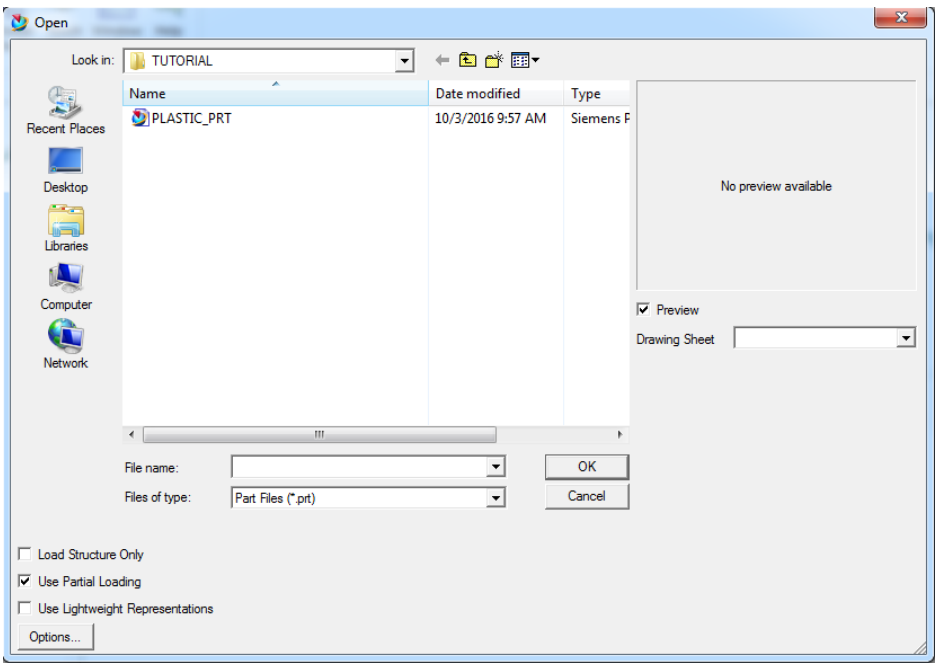


Figure 16-2 The *Open* dialog box

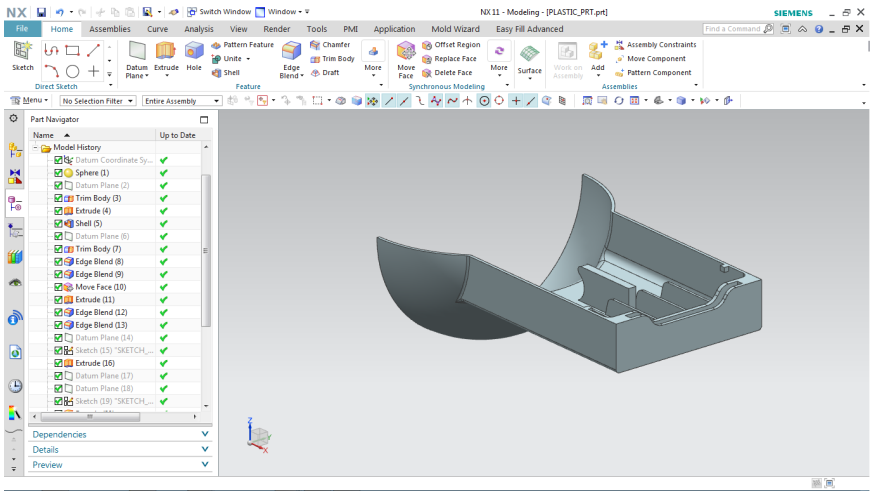


Figure 16-3 The model opened in the Modeling environment

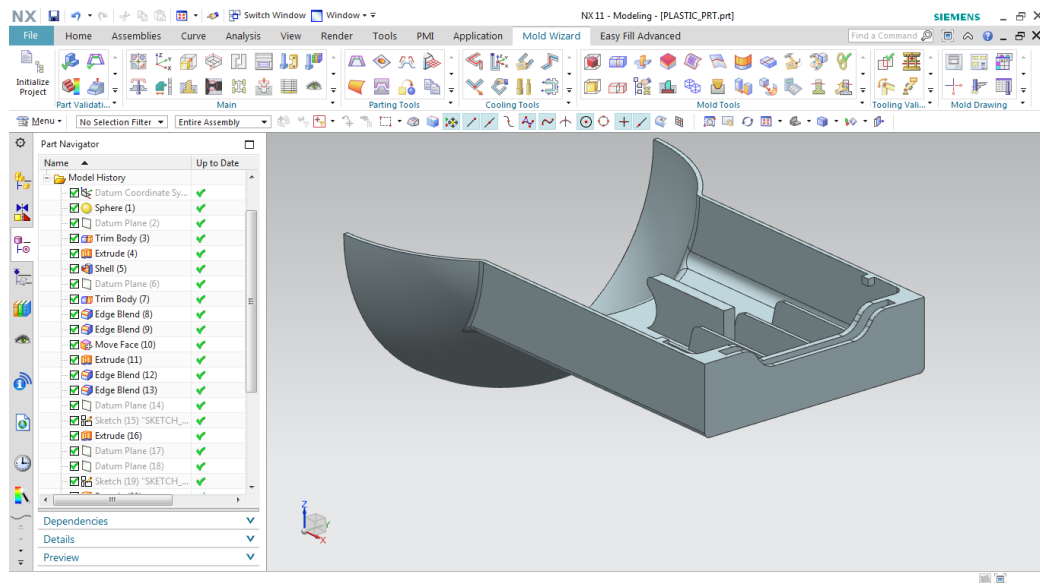


Figure 16-4 The Mold Wizard tab displayed on screen

VARIOUS METHODS FOR CREATING INJECTION MOLD DESIGN

In NX, you can use the following methods for designing an Injection Mold:

- (i) Manual
- (ii) Semi Automatic
- (iii) Automatic

These methods are discussed next.

Manual Method

In manual method, you will not use Mold Wizard but will use Part Modeling, Surfacing, and Assembly environments.

Semi Automatic Method

In semi automatic method, you will use a combination of Mold Wizard, Part, Surfacing, and Assembly environments.

Automatic Method

In automatic method, you will use only Mold Wizard environment for designing an injection mold.

ANALYSIS OF PART

For analyzing the part in NX, the Easy Fill Advanced plugin is being used in this Textbook. You can also use Run Flow Analysis for analyzing the part but due to some constraints in the results, the Easy Fill Advanced plugin is preferred for analysis.

As you install the Easy Fill Advanced plugin, the **Easy Fill Advanced** tab becomes available in NX interface. Figure 16-5 shows the **Easy Fill Advanced** tab.



Figure 16-5 The Easy Fill Advanced tab

Before analyzing the part using the Easy Fill Advanced plug in, you should check the undercut, draft angle, and wall thickness of the part using the **Mold Design Validation** and **Check Wall Thickness** tools.

Mold Design Validation

Ribbon: Mold Wizard > Part Validation gallery > Mold Design Validation



This tool helps to check the quality of the molded part. To check the undercut, draft angle, and mold part quality, invoke the **Mold Design Validation** tool from the **Part Validation** gallery of the **Mold Wizard** tab; the **Mold Design Validation** dialog box will be displayed, as shown in Figure 16-6. The options in this dialog box are discussed next.

Component Validation

The check boxes available in this area help you to find out interference between the electrodes and the overlapping of core and cavity sheets.

Product Quality

The check boxes available in this area help you to find out the undercut and the draft angle of the molded part.

Parting Validation

The check boxes available in this area help you to find out the faces that needs to be split and also the patch surfaces that are overlapping.

As you choose the **Execute Check-Mate** button from the **Parameters** rollout of the dialog box, the **HD3D Tools** window will be displayed, as shown in Figure 16-7. Depending upon the check box selected in the **Mold Design Validation** dialog box, the resulting parameter will be displayed in the **HD3D Tools** window. In this window, the status of the result is shown by symbols. Following are the symbols with their meanings:



This symbol shows the result passed with information.



This symbol shows the result passed with warning.



This symbol shows the result passed.

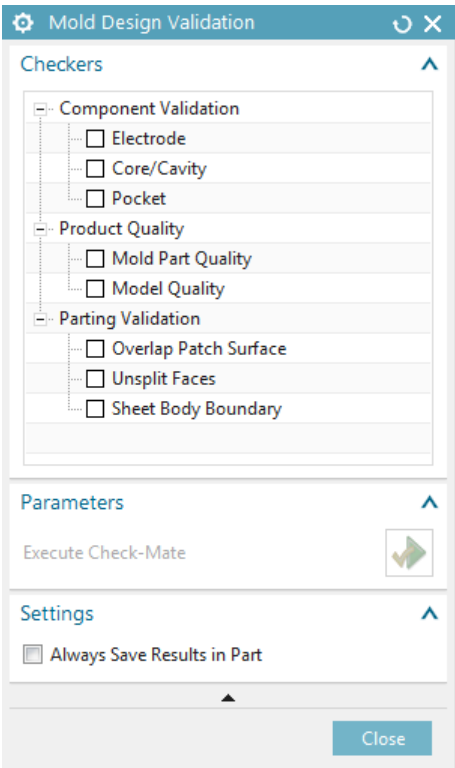


Figure 16-6 The Mold Design Validation dialog box

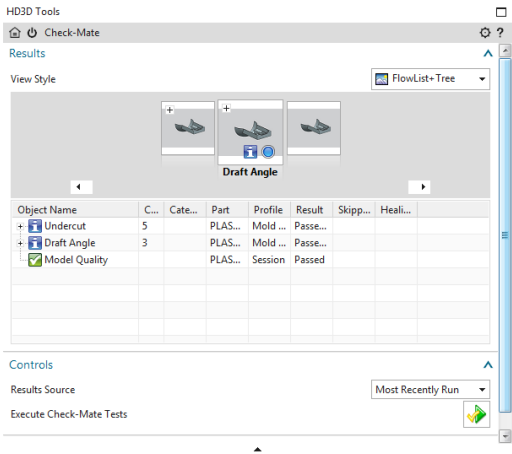



Figure 16-7 The HD3D Tools window

Check Wall Thickness

Ribbon:	Mold Wizard > Part Validation gallery > Check Wall Thickness
Menu:	Analysis > Molded Part Validation > Check Wall Thickness

 A part with uniform thickness allows the mold cavity to fill more easily since the molten plastic does not have to be forced through varying restrictions as it fills. If the walls are not uniform then the thin section cools first as compared to thick section which leaves warpage or sink mark in part. You can analyze the thickness using the **Check Wall Thickness** tool. To analyze the thickness of part, invoke the **Check Wall Thickness** tool from the **Part Validation** gallery of the **Mold Wizard** tab; the **Check Wall Thickness** dialog box will be displayed, as shown in Figure 16-8, and you will be prompted to select solid body to analyze. Select the solid body and choose the **Calculate Thickness** button from the **Process Results** rollout; the result will be displayed, as shown in Figure 16-9. Choose the **OK** button to close the dialog box.

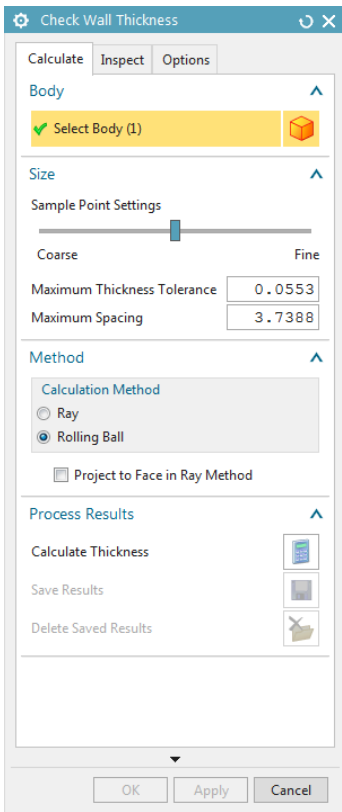


Figure 16-8 The Check Wall Thickness dialog box

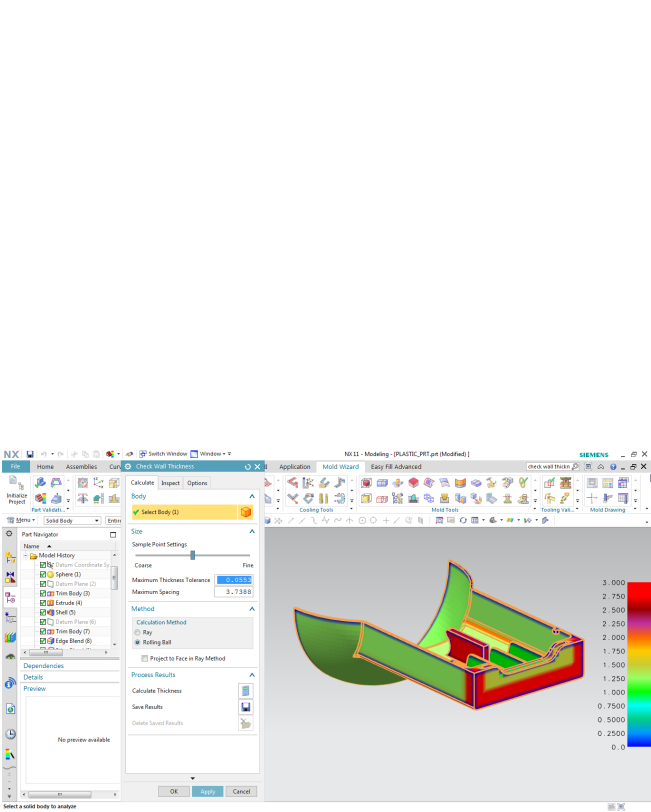


Figure 16-9 The part with thickness analysis

Easy Fill Advanced

The tools in the Easy Fill Advanced plugin help you to simulate filling, packing, and runner balancing of injection molding. Next, you need to understand why this plugin is preferred to use over NX run flow analysis.

Parameter	Easy Fill	Easy Fill Advanced
Simulation capabilities	Filling	Filling and Packing
Runner balance	N/A	YES
Gate Number	Multiple	Multiple
Results availability	10	36 (filling+packing)
Section Results	N/A	YES

This tab comprises of various tools which are discussed next.

Set Working Folder

Ribbon: Easy Fill Advanced > Setting > Set Working Folder

Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Set Working Folder



This tool helps you to locate the analysis result in the specified folder. To set the working folder, choose the **Set Working Folder** tool from the **Setting** group of the **Easy Fill Advanced** tab; the **Working Folder** dialog box will be displayed, as shown in Figure 16-10.

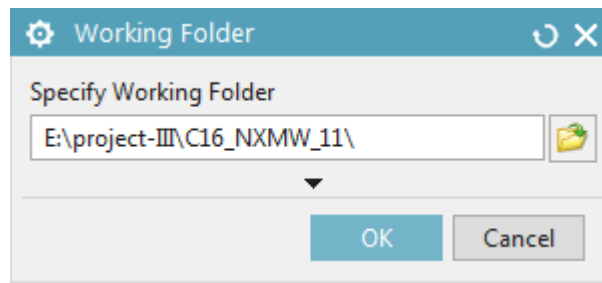


Figure 16-10 The Working Folder dialog box

To locate the folder location, choose the **Browse** button from the **Working Folder** dialog box; the **Open** dialog box will be displayed, as shown in Figure 16-11. From this dialog box, you can set the directory of the analysis result. After specifying the directory of the analysis result, choose the **OK** button to close the dialog box.

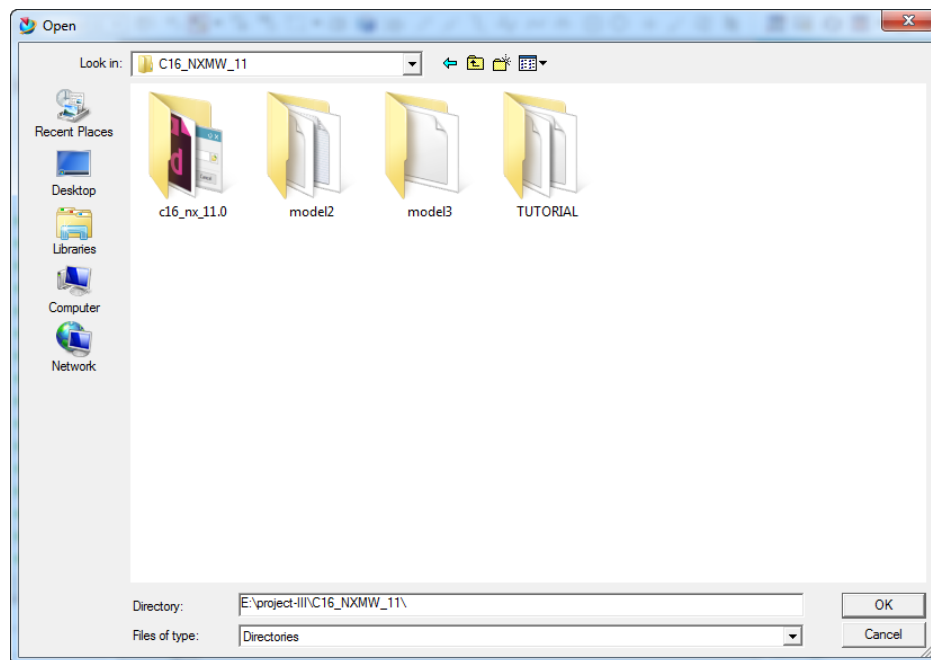



Figure 16-11 The Open dialog box

Set Cavity

Ribbon:
Menu:

Easy Fill Advanced > Setting > Set Cavity
Analysis > Molded Part Validation > Easy Fill Advanced > Set Cavity

 This tool helps you to specify the cavity of the component and assign material to the component. To specify the cavity, choose the **Set Cavity** tool from the **Setting** group in the **Easy Fill Advanced** tab; the **Select Cavity** dialog box will be displayed, as shown in Figure 16-12. Also, you will be prompted to specify solid body as cavity. Select the component from the drawing window. Next, choose the **Push button to select material** button from the **Material Setting** rollout; the **Moldex3D Material Wizard** dialog box will be displayed, refer to Figure 16-13. You can specify the name of material, producer name, and grade name in this dialog box. After specifying the material, producer, and grade name, choose the **OK** button from the **Moldex3D Material Wizard** dialog box. Next, choose the **OK** button from the **Select Cavity** dialog box.

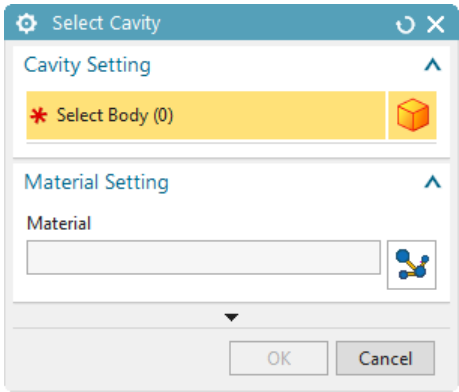


Figure 16-12 The *Select Cavity* dialog box

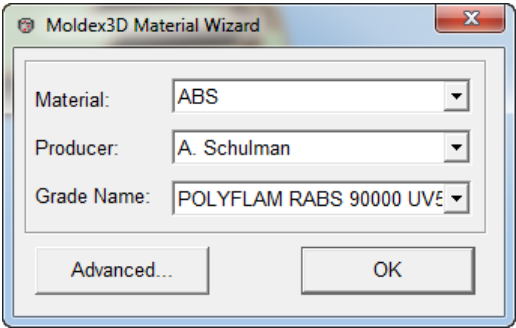



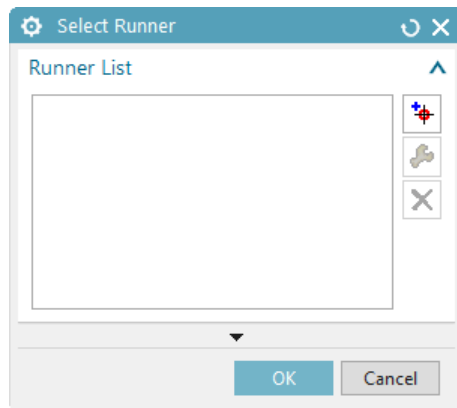
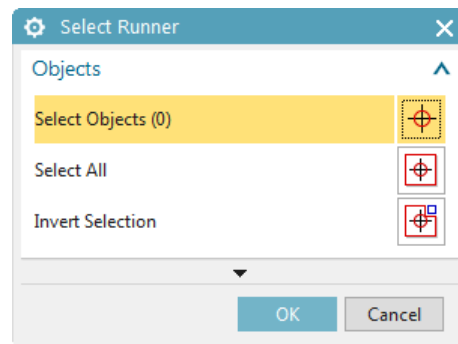
Figure 16-13 The *Moldex3D Material Wizard* dialog box

Set Runner

Ribbon:
Menu:

Easy Fill Advanced > Setting > Set Runner
Analysis > Molded Part Validation > Easy Fill Advanced > Set Runner

 This tool helps you to define the runner system of the solid bodies. To set the runner system, choose the **Set Runner** tool from the **Setting** group in the **Easy Fill Advanced** tab; the **Select Runner** dialog box will be displayed, as shown in Figure 16-14. Choose the **Specify Runner** button from the **Runner List** rollout; the **Select Runner** dialog box will be displayed, refer to Figure 16-15. Also, you will be prompted to specify the curve or solid body to specify as Runner. Select the solid body or curve from the drawing window. Choose the **OK** button to close the dialog box; the **Settings** dialog box will be displayed. You can select the type of runner from the **Type** drop-down of the **Attribute** rollout. Choose the **OK** button to close the dialog box. You will notice that type of runner and name of runner will be displayed in the **Runner List** rollout. Choose the **OK** button to close the dialog box.

Figure 16-14 The **Select Runner** dialog boxFigure 16-15 The **Select Runner** dialog box**Note**

If you select a body for setting runner, then you can only define the type of runner but if you select a curve, then you can define the type of runner and its cross-section parameters.

Set Melt Entrance

Ribbon: Easy Fill Advanced > Setting > Set Melt Entrance
Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Set Melt Entrance



Melt entrance is the path through which molten material comes into the runner. To set the Melt Entrance for the runner, choose the **Set Melt Entrance** tool from the **Setting** group of the **Easy Fill Advanced** tab; the **Melt Entrance Setting** dialog box will be displayed, as shown in Figure 16-16.

Next, choose the **Specify Melt Entrance Position** button from the **Melt Entrance List** rollout; the **Point** dialog box will be displayed, refer to Figure 16-17. By using this dialog box, you can place the Melt entrance position. Choose the **OK** button to close the dialog box. You will notice the Melt Entrance name in the **Melt Entrance List** rollout. Choose the **OK** button to close the dialog box.

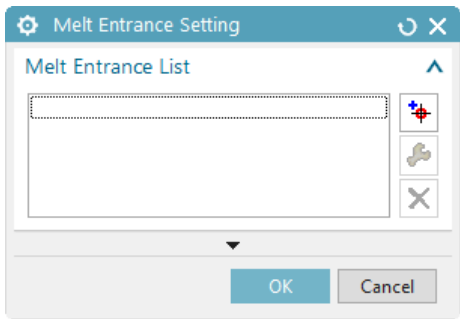


Figure 16-16 The Melt Entrance Setting dialog box

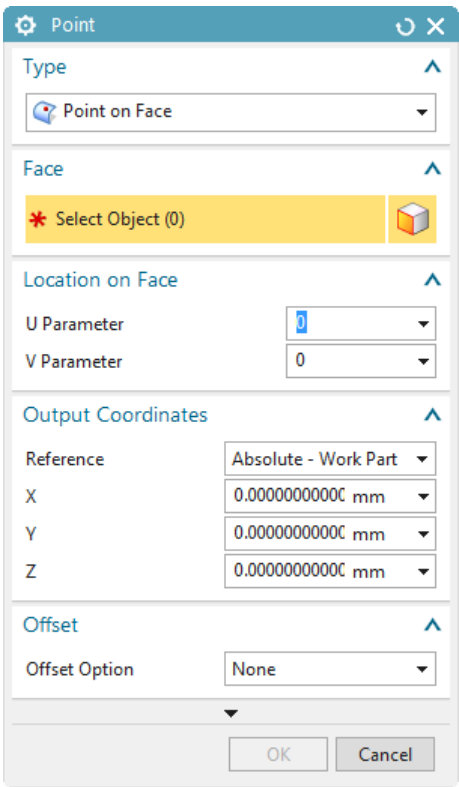


Figure 16-17 The Point dialog box

Set Parting Direction

Ribbon: Easy Fill Advanced > Setting > Set Parting Direction
Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Set Parting Direction



Parting direction of the component plays an important role in mold design procedure. So, you need to decide the parting direction carefully. To set the parting direction, choose the **Set Parting Direction** tool from the **Setting** group in the **Easy Fill Advanced** tab; the **Vector** dialog box will be displayed, refer to Figure 16-18. Select the direction of the partition by selecting the option from the **Type** drop-down list of the rollout. After specifying the correct orientation of the axis, choose the **OK** button to close the dialog box.

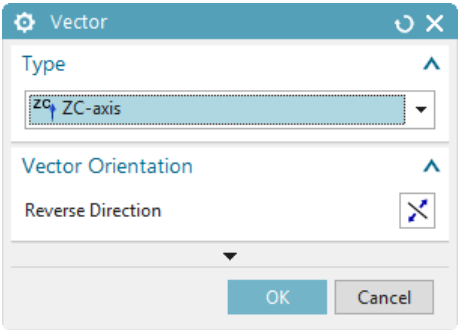


Figure 16-18 The Vector dialog box

Gate Wizard

Ribbon: Easy Fill Advanced > Wizard > Gate Wizard

Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Wizard > Gate Wizard



This tool helps you to create a gate for analysis of flow. To create a gate, choose the **Gate Wizard** tool from the **Wizard** group in the **Easy Fill Advanced** tab; the **Create Gate** dialog box will be displayed, refer to Figure 16-19. Choose the **Point Dialog** button from the **Position of Gate** rollout; the **Point** dialog box will be displayed. Using this dialog box, specify the location of the gate and then choose the **OK** button to close the dialog box. Select the type of gate from the **Type of gate** drop-down of the **Attribute** rollout. Specify the dimensions of the gate cross-section from the **Cross-Section Parameters** rollout.

In NX, the Easy Fill Advanced plug-in supports various types of gates:

- (i) Pin Gate
- (ii) Sprue Gate
- (iii) Edge Gate
- (iv) Fan Gate
- (v) Lapped Edge Gate
- (vi) Tunnel Gate
- (vii) Cashew Gate
- (viii) Tunnel Gate with ejector pin
- (ix) Cashew Gate with ejector pin

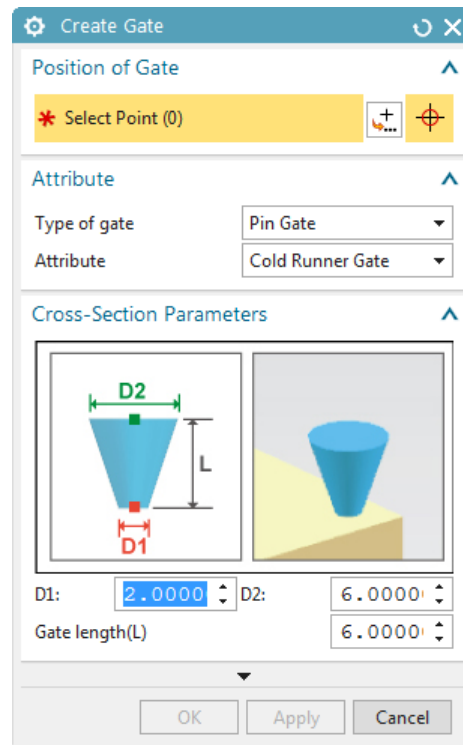


Figure 16-19 The *Create Gate* dialog box

You can also set them as a Cold Runner or a Hot Runner gate. Depending upon the selection of gate, the dimension parameters of gate also get changed. You can define the size of the gate by entering value in the edit boxes of these parameters. After entering the values, choose the **OK** button to close the dialog box.

Runner Wizard

Ribbon: Easy Fill Advanced > Wizard > Runner Wizard

Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Wizard > Runner Wizard



This tool helps you to create the runner system for flow analysis of material. You need to select only the gate location and its size in the model. As a result, the runner wizard automatically calculates its size, shape, and layout. This tool helps you not only in creation of runner but also in designing sprue and melt entrance. With the help of this tool, you can design runner system more efficiently. To create a runner, choose the **Runner Wizard** tool from the **Wizard** group in the **Easy Fill Advanced** tab; the **Runner Wizard** dialog box will be displayed, refer to Figure 16-20.

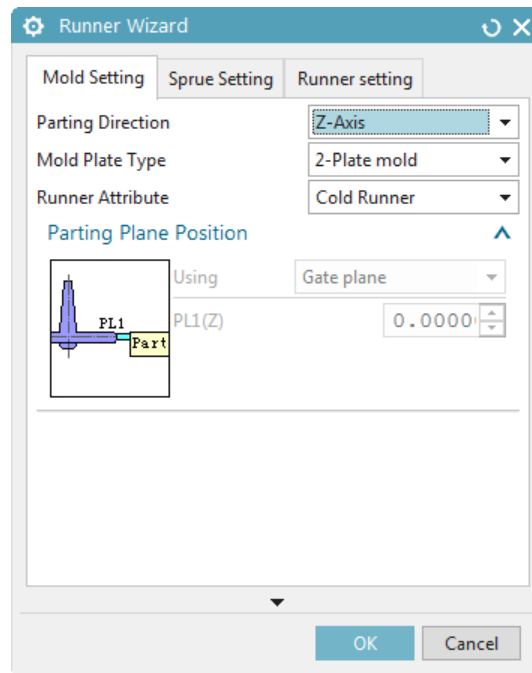


Figure 16-20 The *Runner Wizard* dialog box

Mold Setting

This tab allows you to define the parting direction, mold plate type, and runner attribute. The options of this tab are discussed next.

Parting Direction: The options in this drop-down list are used to specify the orientation of the runner. The orientation may be along X-axis, -X-axis, Y-axis, -Y-axis, Z-axis, -Z-axis or you can customize it along any vector direction.

Mold Plate Type: The options in this drop-down list are used to specify the type of mold. According to the selection of the mold plate type, the parameters of parting line or plane get changed. The following options are available in this drop-down list:

- (i) 2-Plate
- (ii) 3-Plate

Runner Attribute: The options in this drop-down list are used to define the type of runner. The following options are available in this drop-down list:

- (i) Cold Runner
- (ii) Hot Runner

Sprue Setting

This tab allows you to define the sprue position, sprue length, and sprue diameter. The options in this tab are discussed next.

Sprue Position Rollout: The options in this rollout are used to position the sprue.

Sprue Geometry Parameters Rollout: The options in this rollout are used to specify the size parameters of sprue.

Runner Setting

This tab allows you to specify the cross-section and size parameters of runner using the **Runner Geometry Parameters** rollout.

After specifying the parameters of runner, choose the **OK** button to close the dialog box.

Melt Entrance Wizard

Ribbon: Easy Fill Advanced > Wizard > Melt Entrance Wizard
Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Wizard > Melt Entrance Wizard



This tool helps you to specify molten material entrance in the model. To specify melt entrance, choose the **Melt Entrance Wizard** tool from the **Wizard** group in the **Easy Fill Advanced** tab. As you choose this tool, the melt entrance automatically gets placed at gate or runner location.

Start Analysis

Ribbon: Easy Fill Advanced > Setting > Start Analysis
Menu: Analysis > Molded Part Validation > Easy Fill Advanced > Start Analysis



This tool helps you to apply the settings for the analysis of the component. To start the analysis of the component, choose the **Start Analysis** tool from the **Setting** group in the **Easy Fill Advanced** tab; the **Start Analysis** dialog box will be displayed, refer to Figure 16-21. You can specify the process parameters, type of analysis, and the mesh level in this dialog box. Choose the **OK** button to start the analysis. As you choose the **OK** button, the **Easy Fill Advanced Monitor** window will be displayed. This window shows the progress of analysis. When the progress of analysis reaches 100%, close this window by choosing the **Close** button.

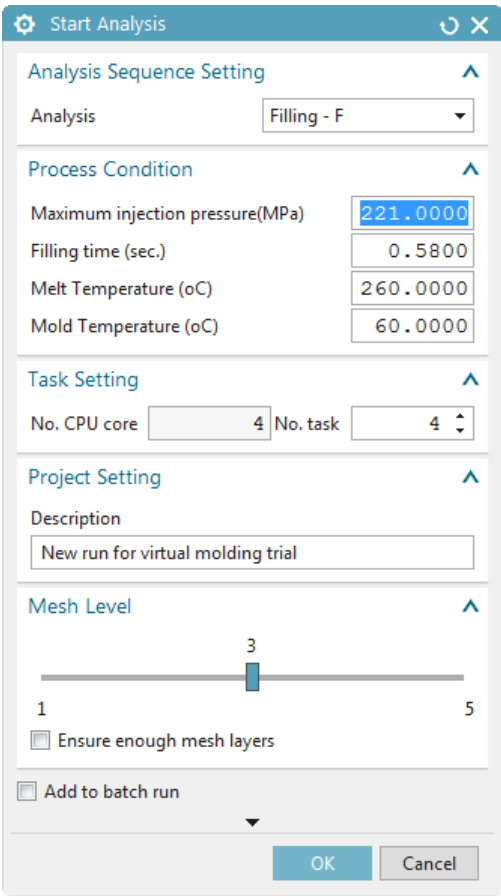


Figure 16-21 The *Start Analysis* dialog box


Show Result

Ribbon:

Menu:

Easy Fill Advanced > Setting > Show Result

Analysis > Molded Part Validation > Easy Fill Advanced > Show Result

 This tool helps you to visualize and analyze the result. To invoke this tool, choose the **Show Result** tool from the **Setting** group of the **Easy Fill Advanced** tab; the **Show Result** dialog box will be displayed, as shown in Figure 16-22. Select result from the **Analysis Result** rollout. When you select the result from this rollout, the dialog box gets modified, refer to Figure 16-22. From the **Display Result** rollout, you can select the **Filling** and **Packing** options. By default, the **Filling** option is selected. You can visualize the results by selecting appropriate option from the **Result Type** drop-down list. Choose the **Close** button to close the dialog box.

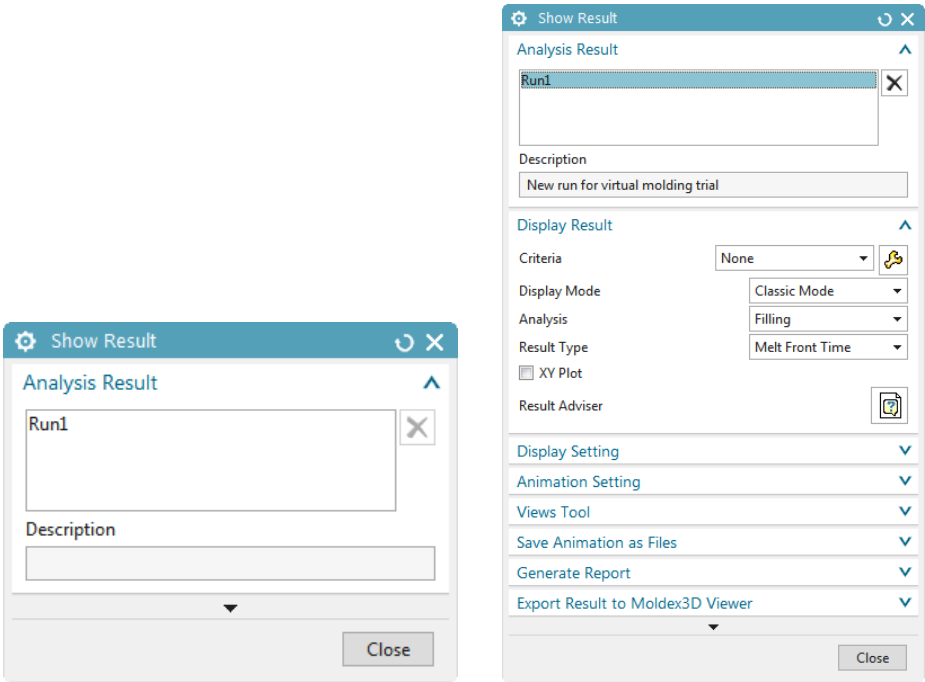


Figure 16-22 The *Show Result* dialog box before and after selecting the result

MOLD WIZARD

Mold Wizard environment helps you to design injection mold to reduce the design cycle time. The Mold Wizard tab contains tools that help you to design injection mold automatically. Before using this wizard, you should ensure that the library related to mold is installed. The tools in this wizard are discussed next.

Initialize Project

Ribbon: Mold Wizard > Initialize Project

The **Initialize Project** tool is used to create a new mold design project or to add a new model to create a family mold. Choose the **Initialize Project** tool from the **Mold Wizard** tab; the **Initialize Project** dialog box will be displayed, as shown in Figure 16-23.

To specify the path of the project, choose the **Browse** button from the **Project Settings** rollout; the **Open** dialog box will be displayed, as shown in Figure 16-24.

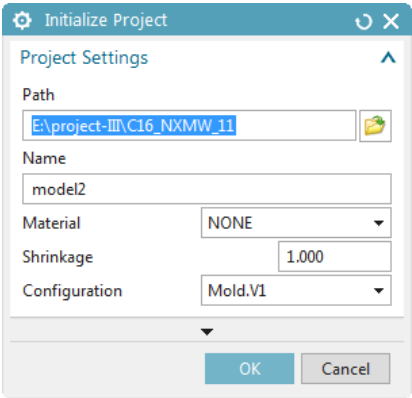


Figure 16-23 The *Initialize Project* dialog box

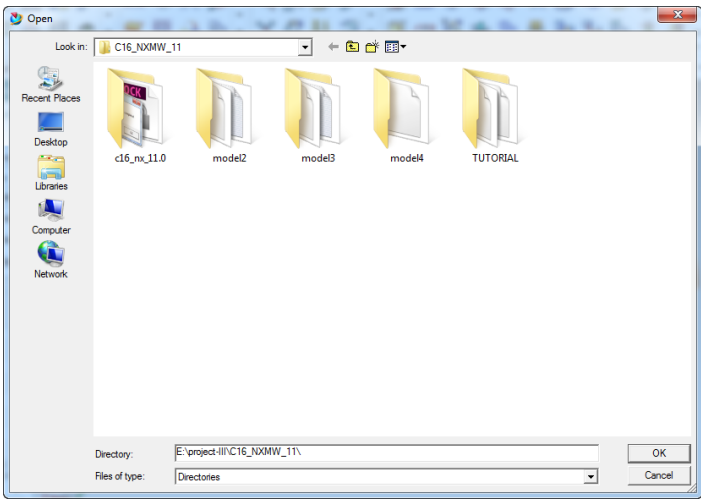


Figure 16-24 The *Open* dialog box

Specify the path of the project and then choose the **OK** button to close the dialog box. Select the name of the material from the **Material** drop-down list of the **Initialize Project** dialog box. Next, choose the **OK** button to close the dialog box; the **New IRAY+ Ray Traced Studio Rendering** window will be displayed. Choose the **OK** button to close the window.

Mold CSYS

Ribbon: Mold Wizard > Main gallery > Mold CSYS



The **Mold CSYS** tool is used to reposition the original component to the correct orientation according to the mold assembly or to the injection molding machine. While positioning the component, you should consider following points:

- Orient the component so that ejection occurs in z-axis of the mold base.
- Position the component in such a way that parting plane lies on X-Y plane.

To orient the component, invoke the **Mold CSYS** tool from the **Main** gallery of the **Mold Wizard** tab; the **MOLD CSYS** dialog box will be displayed, refer to Figure 16-25.

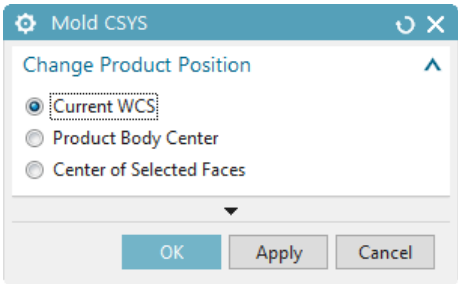


Figure 16-25 The *Mold CSYS* dialog box

In the **Change Product Position** rollout of this dialog box, there are three options to change the position of the component. These options are discussed next.

Current WCS

Select this radio button to reposition the component from the current WCS position and orientation to the position and orientation of the mold base.

Product Body Center

Select this radio button to reposition the WCS to the center of the component body. When you select this radio button, the **Lock XYZ Position** rollout will be displayed. The options in this rollout help you to lock the position of X,Y, and Z coordinates of the component.

Center of Selected Faces

Select this radio button to reposition the component from the center of one or more selected faces to the origin of the mold base. As you select this radio button, the **Lock XYZ Position** rollout gets available. The options in this rollout help you to lock the position of X,Y, and Z coordinates of the component.

Shrinkage

Ribbon: Mold Wizard > Main gallery > Shrinkage



When a molten plastic filled inside a mold is cooled, it solidifies and shrinks to some extent. This process is known as Shrinkage.

The **Shrinkage** tool helps you to apply the scale factor on the product model. While defining the scale factor, you should consider the following points:

- Type of molding material
- Wall thickness of the molded part
- Cavity surface temperature
- Gate shape
- Presence of additive material in the molding materials

To apply shrinkage on the model, choose the **Shrinkage** tool from the **Main** gallery of the **Mold Wizard** tab; the **Scale Body** dialog box will be displayed, refer to Figure 16-26.

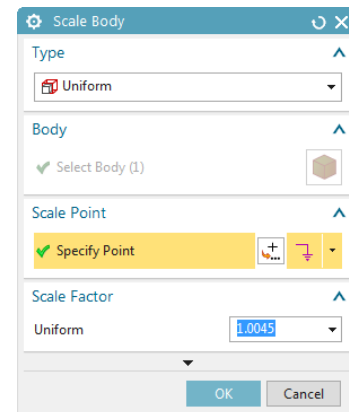


Figure 16-26 The *Scale Body* dialog box

Type Rollout

The drop-down list in this rollout is used to specify the type of shrinkage that you need to apply to the component.

Uniform

This option is used to scale the body uniformly in all directions.

Axisymmetric

This option is used to scale the body symmetrically with a specified scale factor about specified axis.

General

This option is used to scale the body with different scale factors along the X,Y and Z directions.

After scaling the body, choose the **OK** button to close the dialog box.

Check Regions

Ribbon: Mold Wizard > Parting Tools gallery > Check Regions



The **Check Regions** tool is used to analyze the draft angle of faces, set the color in the faces according to the draft values, and find undercut in the model. It also helps to find the core and cavity regions in the model. To analyze the core and cavity region of the model, invoke this tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Check Regions** dialog box and the **Parting Navigator** tree will be displayed, refer to Figure 16-27. Choose the **Calculate** button in the **Calculate** rollout and then choose the **Region** tab. In this tab, choose the **Set Regions Color** button from the **Define Regions** rollout; the color of the model gets changed. These colors represent core, cavity, and undefined region in the model.

The **Cavity Region** radio button is selected by default in the **Assign to Region** rollout. Select the undefined region of the model and then choose the **Apply** button. The undefined region gets the color of the cavity region. Choose the **OK** button to close the dialog box.

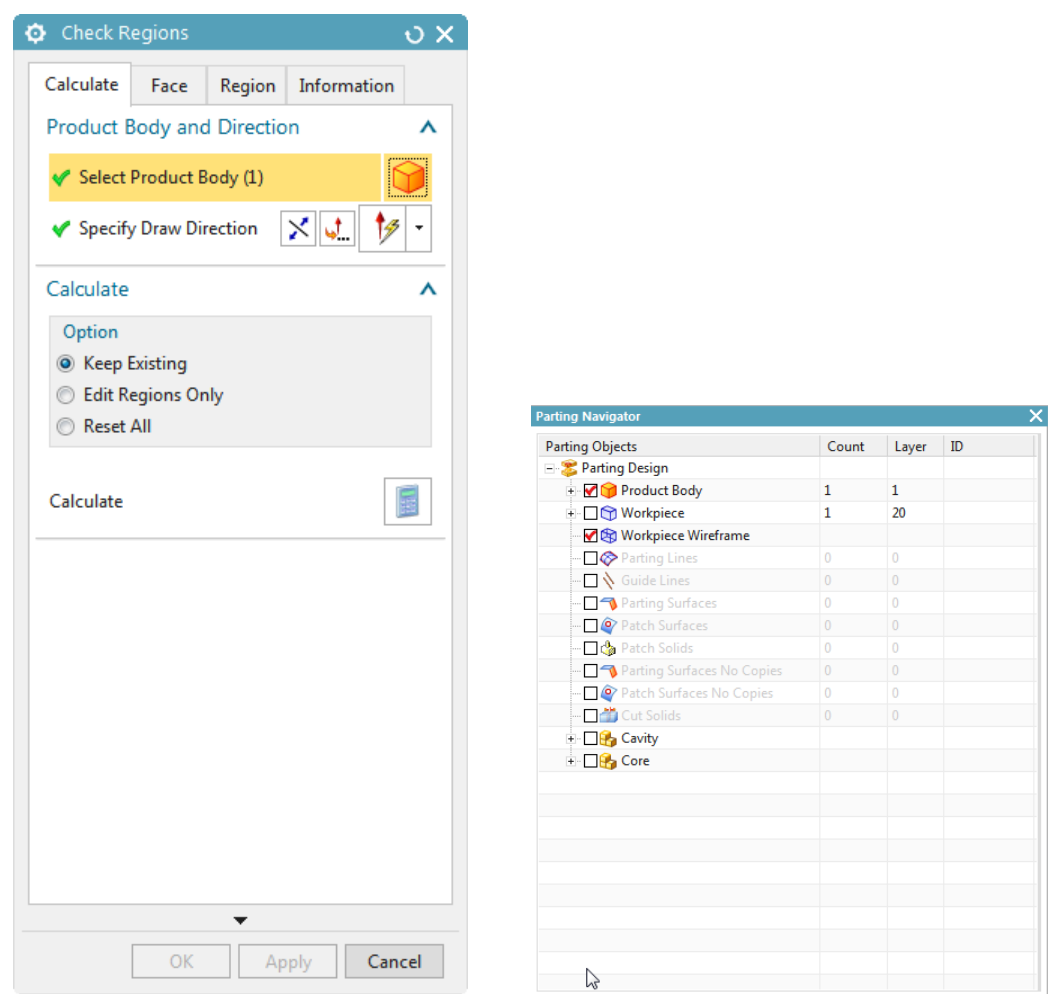



Figure 16-27 The *Check Regions* dialog box and the *Parting Navigator* tree

Patch Surface

Ribbon: Mold Wizard > Parting Tools gallery > Patch Surface

 This tool is used to create sheets to close the openings in the model. To patch a surface, choose the **Patch Surface** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Edge Patch** dialog box will be displayed, refer to Figure 16-28. In NX Mold Wizard, you have three options in the **Type** drop-down list of the **Loop Selection** rollout: **Face**, **Body**, and **Traverse**.

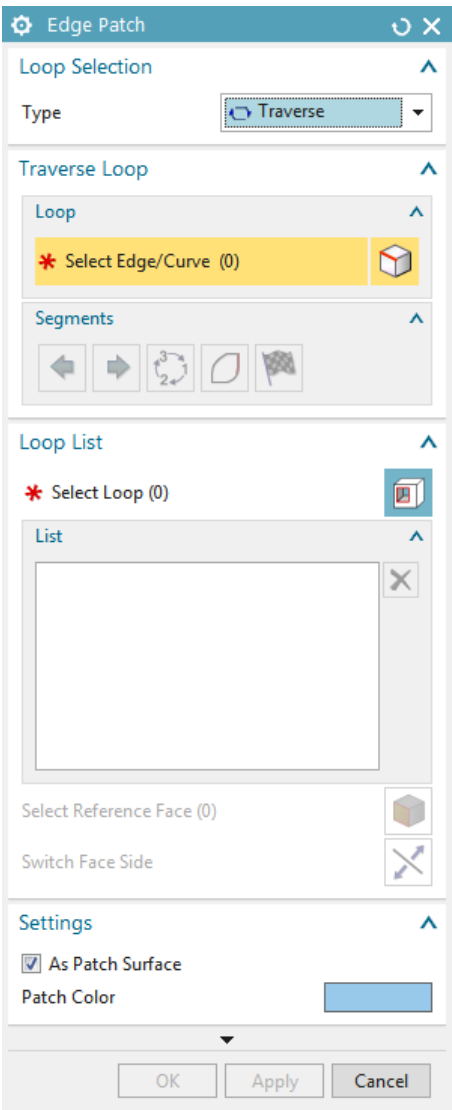



Figure 16-28 The *Edge Patch* dialog box

Define Regions


Ribbon: Mold Wizard > Parting Tools gallery > Define Regions

 This tool is used to create core and cavity region sheets, parting lines, and other region sheets (for sliders and lifters). To create core and cavity region sheets, choose the **Define Regions** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Define Regions** dialog box will be displayed, refer to Figure 16-29.

In the **Define Regions** rollout, you can examine the number of faces selected for cavity, core, and undefined faces which help you to identify the number of faces belonging to core and cavity. The procedure to create regions for core and cavity will be discussed in Tutorial 1.

Design Parting Surface

Ribbon: Mold Wizard > Parting Tools gallery > Design Parting Surface

 This tool is used to create a parting surface of the component. To create a parting surface, invoke the **Design Parting Surface** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Design Parting Surface** dialog box will be displayed, refer to Figure 16-30. You can use the tools available in the **Create Parting Surface** rollout to create parting surfaces. The tools in this rollout are: **Extrude**, **Swept**, **Enlarged Surface**, **Bounded Plane**, **Extend Sheet**, **Ribbon Surface**, and **Guided Extension**.

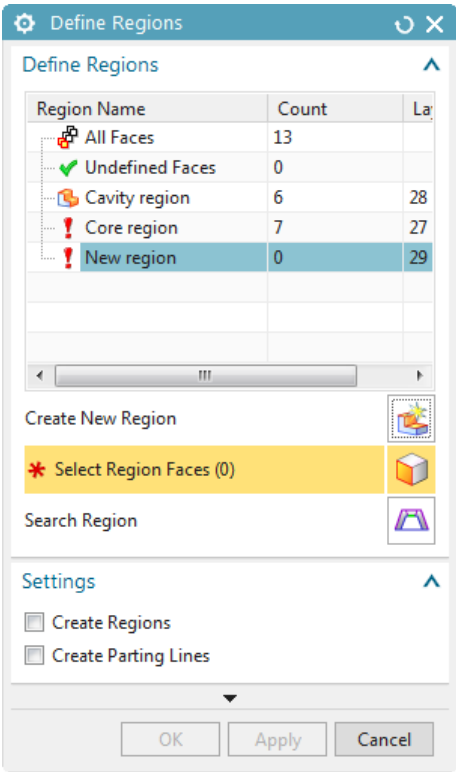


Figure 16-29 The Define Regions dialog box

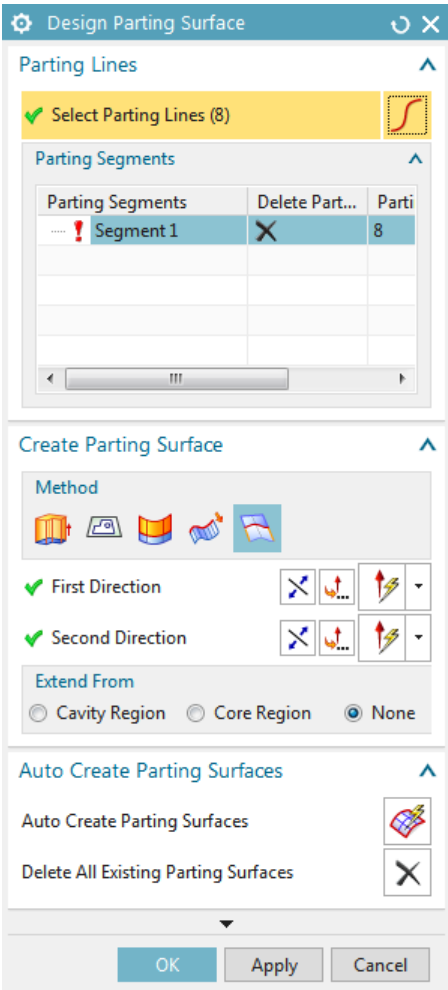

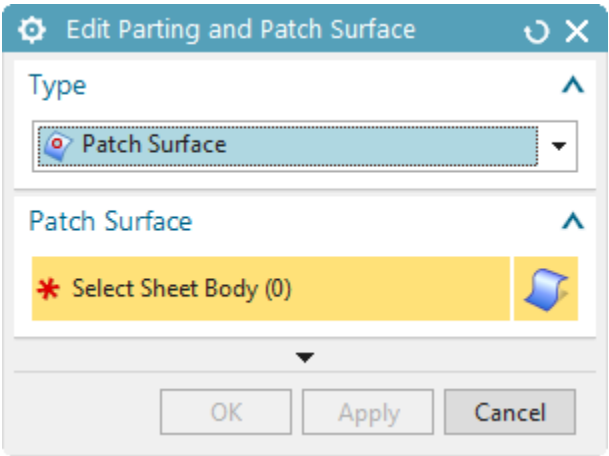


Figure 16-30 The Design Parting Surface dialog box

Edit Parting and Patch Surface

Ribbon: Mold Wizard > Parting Tools gallery > Edit Parting and Patch Surface


 This tool is used to delete parting and patch sheets. To edit parting and patch surface, choose the **Edit Parting and Patch Surface** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Edit Parting and Patch Surface** dialog box will be displayed, refer to Figure 16-31. You can select the type of surface to be edited from the **Type** drop-down list of the **Type** rollout.



*Figure 16-31 The **Edit Parting and Patch Surface** dialog box*

Workpiece

Ribbon: Mold Wizard > Main gallery > Workpiece

 This tool helps you to define the size of core and cavity. To define the size, choose the **Workpiece** tool from the **Main** gallery of the **Mold Wizard** tab; the **Workpiece** dialog box will be displayed, refer to Figure 16-32. The options in this dialog box are discussed next.

Type Rollout

The options in the drop-down list available in this rollout are used to define whether you want to create core and cavity for single component or multiple components.

Product Workpiece

This option helps you to create the core and cavity for single component.

Combined Workpiece

This option helps you to create the core and cavity for multiple components.

Workpiece Method Rollout

The options in the drop-down list in this rollout are used to define the way you want to create the workpiece. The options available in the drop-down list are:

- (i) User Defined Block
- (ii) Cavity-Core
- (iii) Cavity Only
- (iv) Core Only

Dimensions Rollout

The options in this rollout help you to define the size of the workpiece. After specifying the size of the workpiece, choose the **OK** button to close the dialog box.

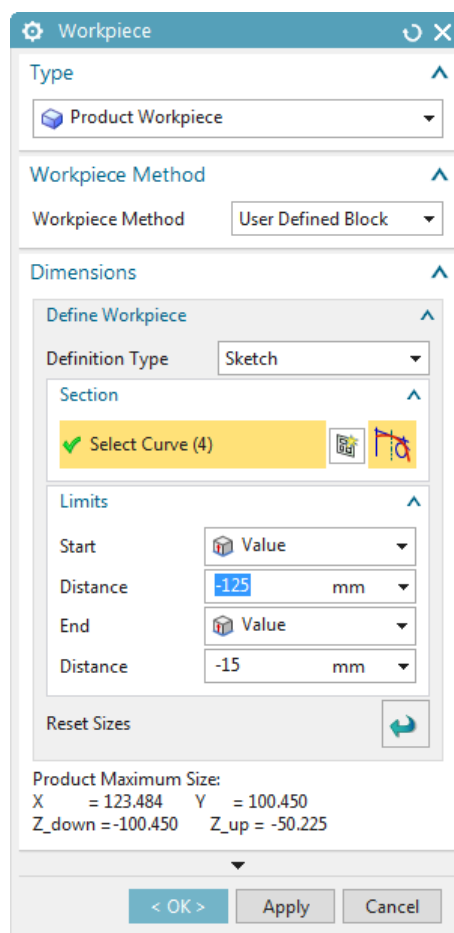



Figure 16-32 The *Workpiece* dialog box

Cavity Layout

Ribbon: Mold Wizard > Main gallery > Cavity Layout

 This tool helps you to create the layout of core and cavity workpiece. To do so, choose the **Cavity Layout** tool from the **Main** gallery of the **Mold Wizard** tab; the **Cavity Layout** dialog box will be displayed, refer to Figure 16-33. Also, the workpiece will be selected automatically, as shown in Figure 16-34. The options available in various rollouts of the **Cavity Layout** dialog box are discussed next.

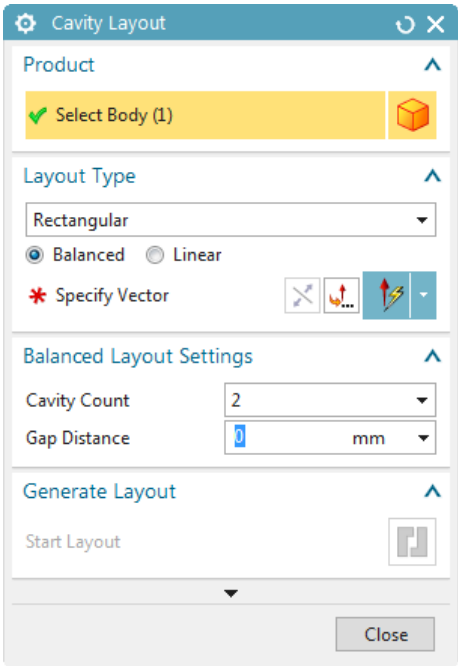


Figure 16-33 The Cavity Layout dialog box

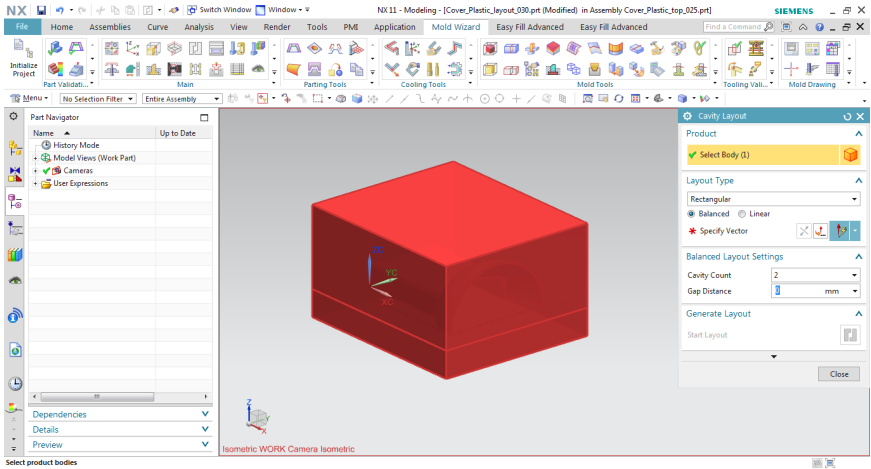


Figure 16-34 The workpiece selected automatically

Product Rollout

The option in this rollout is used to select the workpiece.

Layout Type Rollout

The options in this rollout are used to specify the type of orientation to be created. The options available in this rollout are discussed next.

Layout Type

The options in this drop-down list are used to define the layout type for the cavity. The **Rectangular** option in this drop-down list is selected by default and helps you to create rectangular layout of the workpiece. You can also choose the **Circular** option from this drop-down list to create a circular layout of the component. The **Specify Vector** area in the **Layout Type** rollout helps you to define the directional placement of the component with respect to other component. Specify the parameter of the layout in the **Balanced Layout Settings** or **Circular Layout Settings** rollout. To create the rectangular or circular layout, choose the **Start Layout** button from the **Generate Layout** rollout; the preview of the layout will be displayed in the window. Next, choose the **Edit Insert Pocket** button from the **Edit Layout** rollout; the **Insert Pocket** dialog box will be displayed, refer to Figure 16-35. Select the desired value from the **R** and **type** drop-down lists. Choose the **OK** button to close the dialog box. Next, choose the **Close** button from the **Cavity Layout** dialog box. Figures 16-36 and 16-37 show the rectangular and circular layouts respectively.

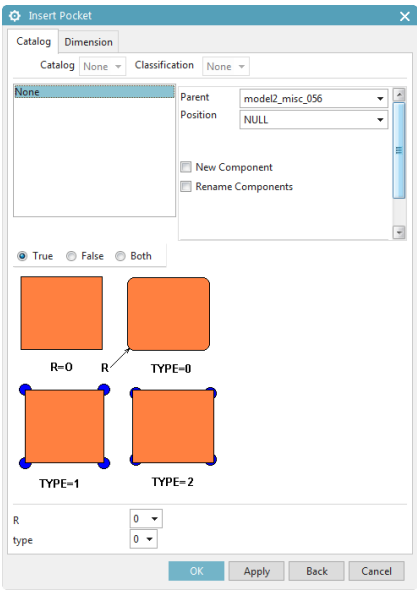


Figure 16-35 The Insert Pocket dialog box

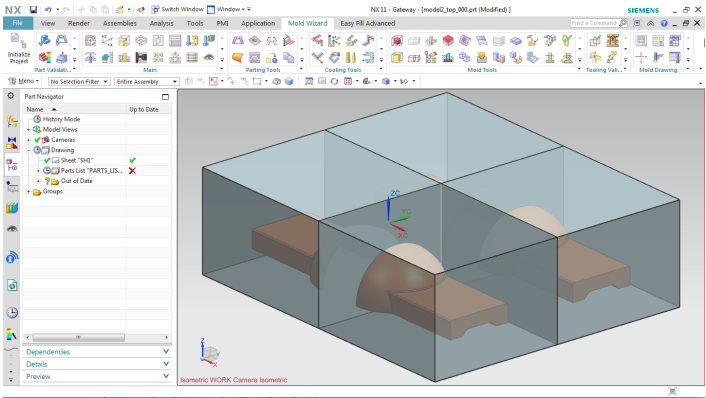


Figure 16-36 The rectangular layout of workpiece

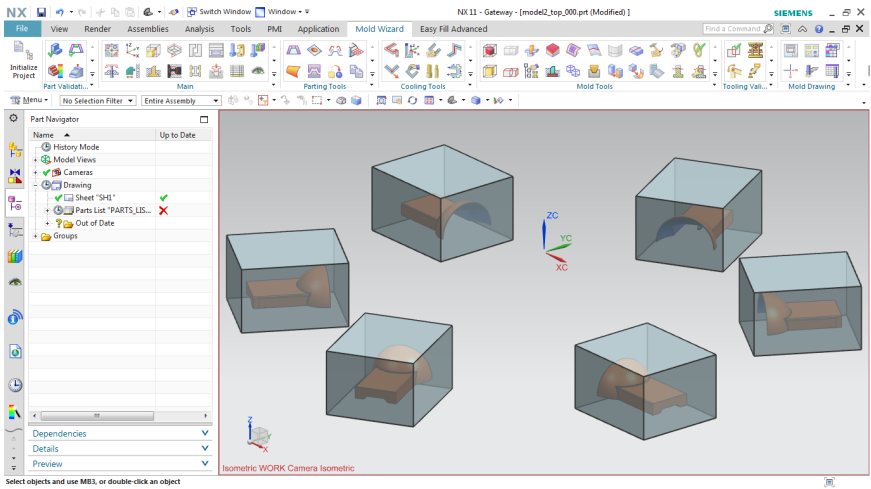



Figure 16-37 The circular layout of workpiece

Define Cavity and Core

Ribbon: Mold Wizard > Parting Tools gallery > Define Cavity and Core

 This tool helps you to create core and cavity of the component. To create core and cavity of the model, choose the **Define Cavity and Core** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Define Cavity and Core** dialog box will be displayed, refer to Figure 16-38. In the dialog box, the **Cavity region** area is selected by default in the **Select Sheets** rollout. Choose the **Apply** button; the **New Iray + Ray Traced Studio Rendering** window will be displayed. Choose the **OK** button; the **View Parting Result** window will be displayed along with the **Information** window. Ensure that the cavity is created; otherwise choose the **Reverse Normal** button. Choose the **OK** button; the **Define Cavity and Core** dialog box will be displayed again. Select the **Core region** area from the **Select Sheets** rollout. Choose the **Apply** button; the **New Iray + Ray Traced Studio Rendering** window will be displayed. When you choose the **OK** button to close the window, the **View Parting Result** window will be displayed. Ensure that the core is created; otherwise choose the **Reverse Normal** button. Choose the **OK** button to close the window; the **Define Cavity and Core** dialog box will be displayed again. Choose the **Cancel** button to close the dialog box. Click the left mouse button and then choose the **Switch Window** tool to check the core and cavity window.

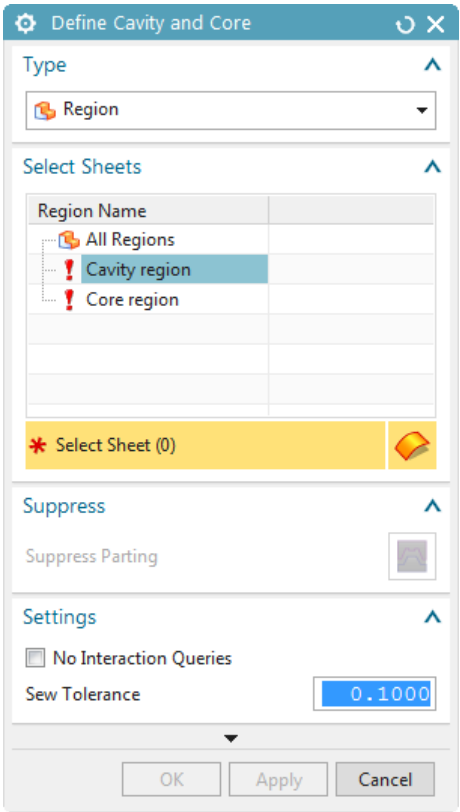



Figure 16-38 The Define Cavity and Core dialog box

Mold Base Library

Ribbon: Mold Wizard > Main gallery > Mold Base Library

 The **Mold Base Library** tool helps you to add mold plates to the core and cavity. To invoke this tool, choose the **Mold Base Library** tool from the **Main** gallery of the **Mold Wizard** tab; the **Mold Base Library** dialog box will be displayed, refer to Figure 16-39. Also, the **Information** window, refer to Figure 16-40, and the **Reuse Library** navigator on the left in the window will be displayed. Select the standard of the mold base from the **MW Mold Base Library** of the **Reuse library** navigator and specify the size in the **Details** rollout of the **Mold Base Library** dialog box and then choose the **OK** button.

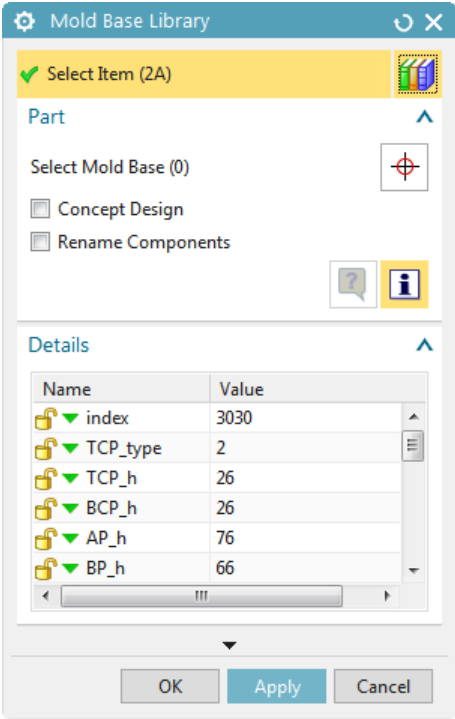


Figure 16-39 The Mold Base Library dialog box

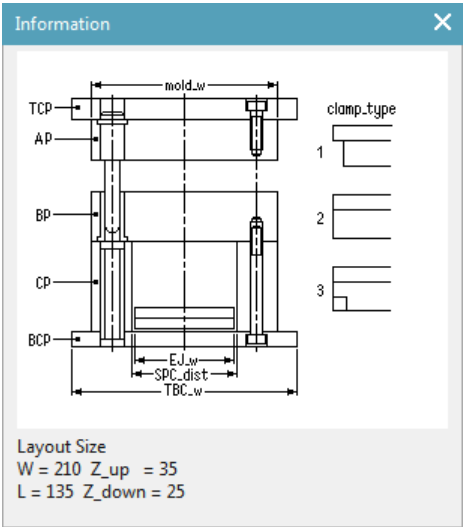



Figure 16-40 The Information window

Standard Part Library

Ribbon: Mold Wizard > Main gallery > Standard Part Library

 The **Standard Part Library** tool helps you to add the standard parts in the mold like sprue bush, locating ring, and so on. To add a standard part, choose the **Standard Part Library** tool from the **Main** gallery of the **Mold Wizard** tab; the **Standard Part Management** dialog box will be displayed, refer to Figures 16-41. Also, the **Information** window, refer to Figure 16-42, and the **Reuse Library** navigator on the left in the window will be displayed. Select the required standard from the MW Mold Base Library of the **Reuse Library** such as locating ring and sprue bush. After specifying the parameters of the parts, choose the **OK** button to close the dialog box.

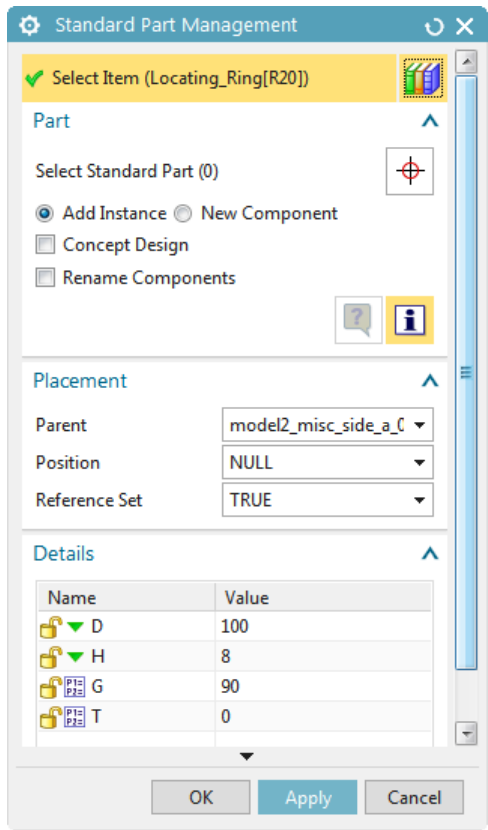


Figure 16-41 The Standard Part Management dialog box

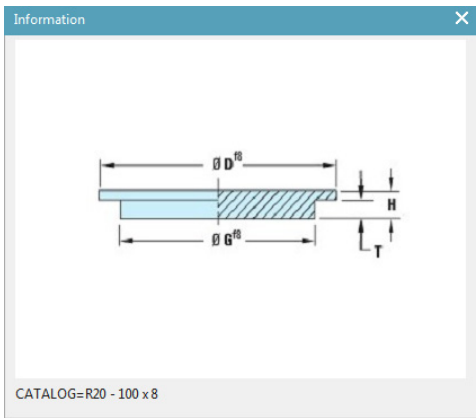


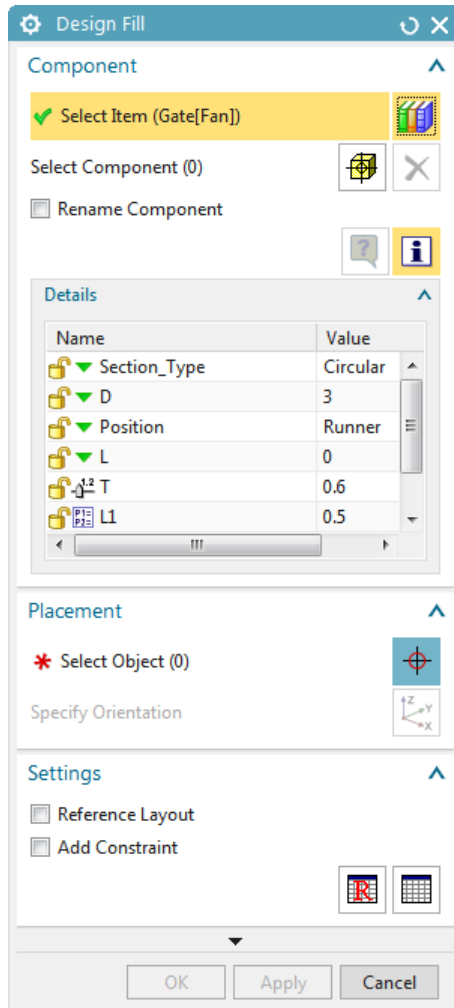
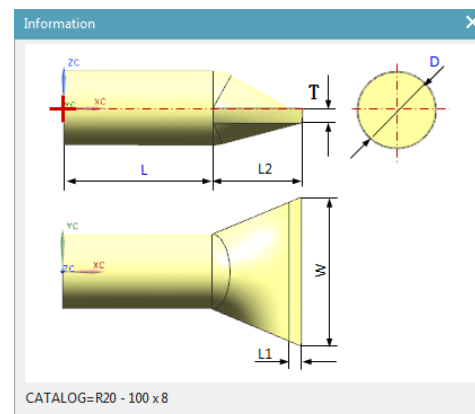
Figure 16-42 The Information window

Design Fill

Ribbon: Mold Wizard > Main gallery > Design Fill



The **Design Fill** tool helps you to create a gate which connects with core or cavity. To create a gate, choose the **Design Fill** tool from the **Main** gallery of the **Mold Wizard** tab; the **Design Fill** dialog box will be displayed, refer to Figure 16-43. Also, the **Information** window, refer to Figure 16-44, and the **Reuse Library** navigator on the left in the window will be displayed. Select the required type of gate from the **MW Mold Base Library** of the **Reuse Library**. Now, specify the parameters in the **Details** rollout of the **Design Fill** dialog box and then choose the **OK** button to close the dialog box.

Figure 16-43 The **Design Fill** dialog boxFigure 16-44 The **Information** window

Runner

Ribbon: Mold Wizard > Main gallery > Runner



The **Runner** tool helps you to create the runner which connects with sprue bush and gate. To create a runner, choose the **Runner** tool from the **Main** gallery of the **Mold Wizard** tab; the **Runner** dialog box will be displayed, refer to Figures 16-45. Also, the **Information** window, refer to Figure 16-46, and the **Reuse Library** navigator on the left in the window will be displayed. Create the sketch of the runner layout and select the type of runner cross-section. Specify the parameters of runner and then choose the **OK** button to close the dialog box.

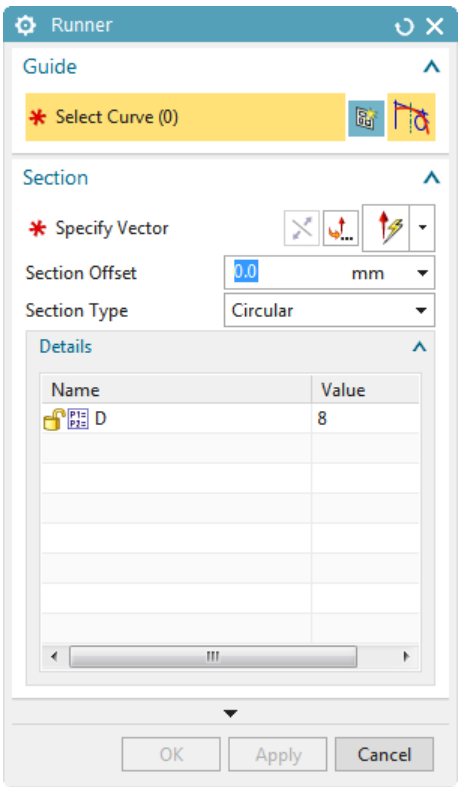


Figure 16-45 The *Runner* dialog box

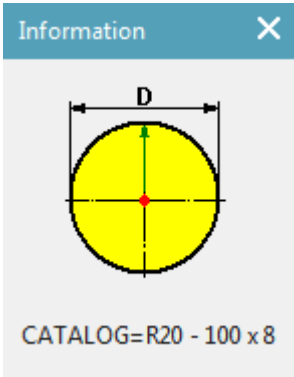



Figure 16-46 The *Information* window

Slide and Lifter Library

Ribbon: Mold Wizard > Main gallery > Slide and Lifter Library

 The **Slide and Lifter Library** tool helps you to create a slider in mold design. To create the slider, choose the **Slide and Lifter Library** tool from the **Main** gallery of the **Mold Wizard** tab; the **Slide and Lifter Design** dialog box will be displayed, refer to Figure 16-47. Also the **Information** window, refer to Figure 16-48, and the **Reuse Library** navigator on the left in the window will be displayed. Select the type of slider from the **Member Select** panel of the **Reuse Library** and specify the parameters in the **Details** rollout of the **Slide and Lifter Design** dialog box. Choose the **OK** button to close the dialog box.

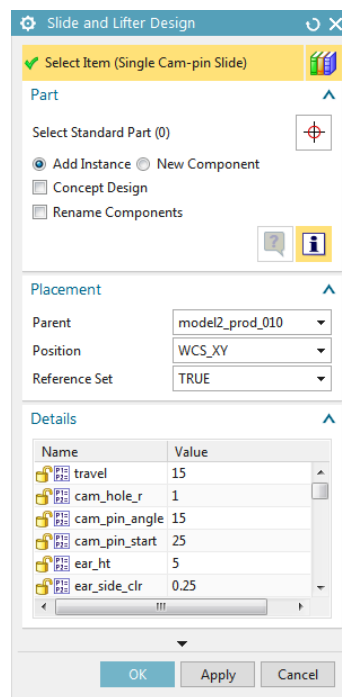


Figure 16-47 The Slide and Lifter Design dialog box

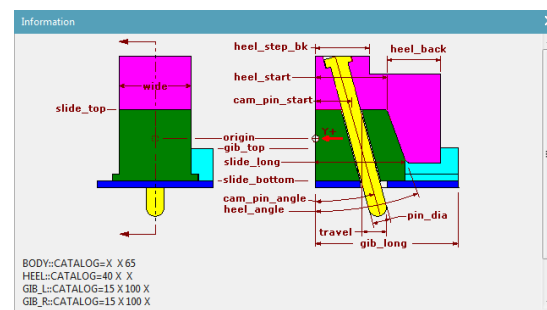


Figure 16-48 The Information window

Slide and Lifter Library

Ribbon: Mold Wizard > Main gallery > Slider and Lifter Library



The **Slide and Lifter Library** tool helps you to create lifters in mold design. To create a lifter, choose the **Slide and Lifter Library** tool from the **Main** gallery of the **Mold Wizard** tab; the **Slide and Lifter Design** dialog box will be displayed, refer to Figure 16-49. Also, the **Information** window, refer to Figure 16-50, and the **Reuse Library** navigator on the left in the window will be displayed. Select the **Lifter** from the **SLIDE_LIFT** sub-tree of the **MW Slide and Lifter Library** tree in **Reuse Library**. Select the type of lifter from the **Reuse Library** and specify the parameters in the **Details** rollout of the **Slide and Lifter Design** dialog box. Choose the **OK** button to close the dialog box.

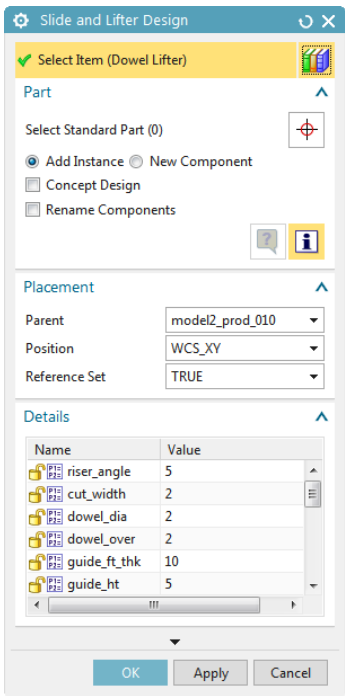


Figure 16-49 The Slide and Lifter Design dialog box

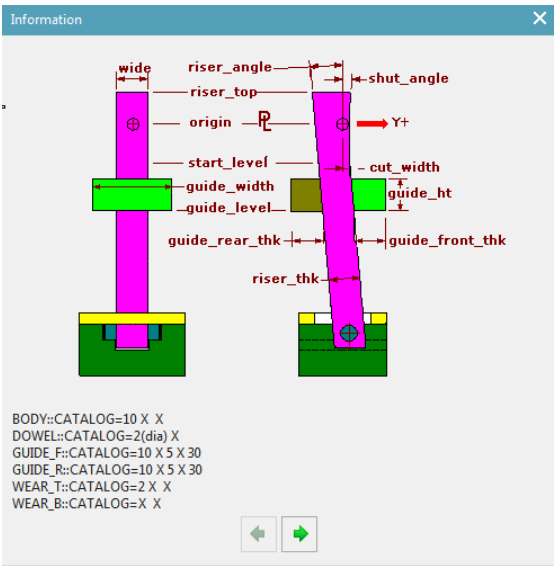


Figure 16-50 The Information window

Design Ejector Pin

Ribbon: Mold Wizard > Main gallery > Design Ejector Pin

This tool helps you to create ejector pins that are used for ejection of the component. To create an ejector pin, choose the **Design Ejector Pin** tool from the **Main** gallery of the **Mold Wizard** tab; the **Design Ejector Pin** dialog box will be displayed, refer to Figure 16-51. Also, the **Information** window, refer to Figure 16-52, and the **Reuse Library** navigator on the left in the window will be displayed. Select the type of ejector pin from the **Member Select** panel of the **Reuse Library** and specify the parameters in the dialog box. Position the pins so that the ejection takes place and then choose the **OK** button to close the dialog box, refer to Figure 16-53.

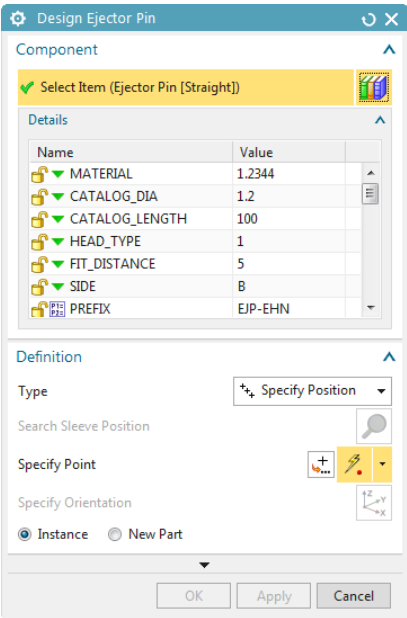


Figure 16-51 The Design Ejector Pin dialog box

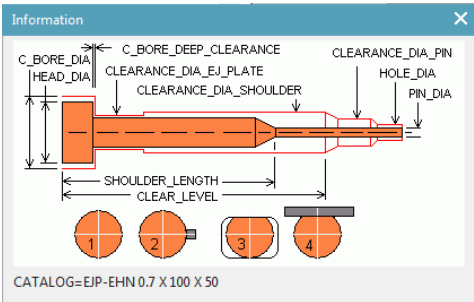


Figure 16-52 The Information window

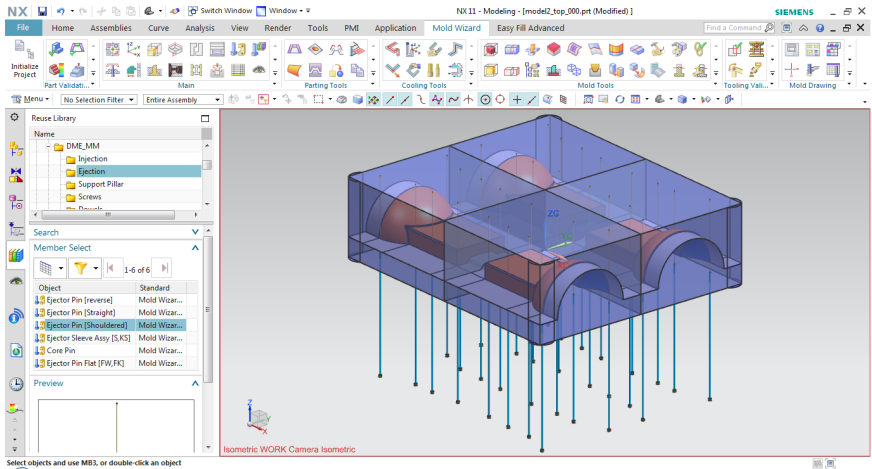


Figure 16-53 The placement of ejector pins

Ejector Pin Post Processing

Ribbon: Mold Wizard > Main gallery > Ejector Pin Post Processing



This tool helps you to trim the ejector pin. To trim the ejector pin, choose the **Ejector Pin Post Processing** tool from the **Main** gallery of the **Mold Wizard** tab; the **Ejector Pin Post Processing** dialog box will be displayed, refer to Figure 16-54. Also, you will be prompted to select the target ejector pins. Select the ejector pins that you need to trim and then choose the **OK** button; the pins will be trimmed and the dialog box will be closed.

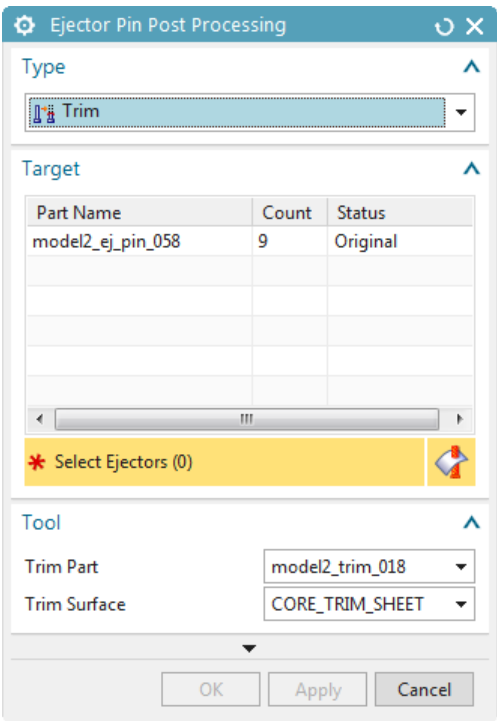



Figure 16-54 The Ejector Pin Post Processing dialog box

Sub-Insert Library

Ribbon: Mold Wizard > Main gallery > Sub-Insert Library

 This tool helps you to create and insert those components that go through wear and tear regularly or need regular replacement. Generally, the material of the insert is P20 or H13. To create the sub-insert in the core and cavity, choose the **Sub-insert Library** tool from the **Main** gallery of the **Mold Wizard** tab; the **Sub-insert Design** dialog box with the **Information** window will be displayed, refer to Figures 16-55 and 16-56. Specify whether you need **CORE SUB INSERT** or **CAVITY SUB INSERT** from the **Member Select** area of the **Reuse Library** dialog box and then specify the parameters of the insert from the **Details** rollout of the dialog box. Choose the **OK** button to close the dialog box; the **Point** dialog box will be displayed. Specify the parameter in dialog box and then choose the **OK** button. Choose the **Cancel** button to close the dialog box.

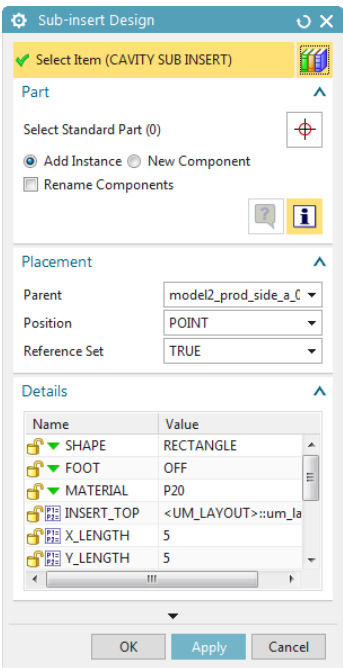


Figure 16-55 The Sub-insert Design dialog box

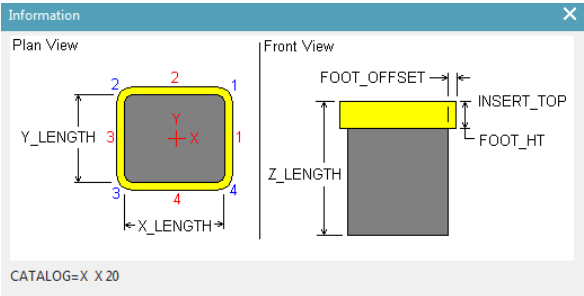



Figure 16-56 The Information Window


Pocket

Ribbon: Mold Wizard > Main gallery > Pocket

 This tool helps you to create a pocket in the mold plates and inserts. To create a pocket, choose the **Pocket** tool from the **Main** gallery of the **Mold Wizard** tab; the **Pocket** dialog box will be displayed, refer to Figure 16-57. Also, you will be prompted to select the target bodies. Select the required mold plates or insert. By default, the **Subtract Material** option is selected in the **Mode** rollout of the dialog box. After selecting the body, click in the **Select Object** area of the **Tool** rollout. Select the tool parts that are to be subtracted like ejector pin and then choose the **Apply** button from the dialog box. Repeat this step until the required subtraction has been done. Choose the **OK** button to close the dialog box.

View Manager

Ribbon: Mold Wizard > Main gallery > View Manager

 This tool helps you to manage the display of the mold model components. As you choose this tool, the **View**

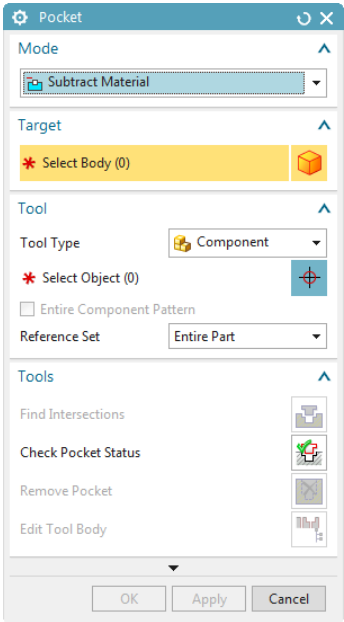


Figure 16-57 The Pocket dialog box

Manager Navigator will be displayed, refer to Figure 16-58. The functioning of this navigator is similar to assembly navigator.

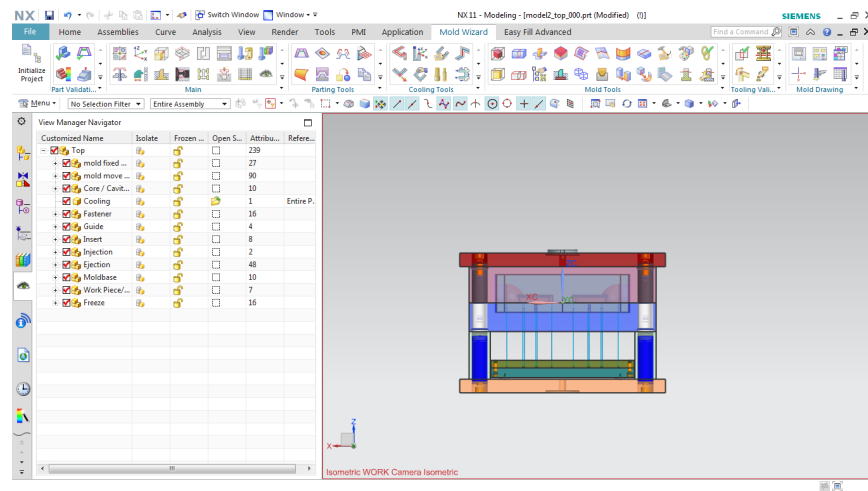



Figure 16-58 The View Manager Navigator

Cooling

Cooling is a process required to cool the component. The cycle time of cooling plays an important role in mold costing. NX provides you the **Cooling Tools** gallery in which all the tools related to cooling are available. The methods to create various types of cooling channel are discussed next.

Pattern Channel

Ribbon: Mold Wizard > Cooling Tools gallery > Pattern Channel

 This tool is used to create the cooling channels using sketches or curves. To create a channel, choose the **Pattern Channel** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Pattern Channel** dialog box will be displayed, refer to Figure 16-59. Also, you will be prompted to create the sketch or select the section geometry. After creating the sketch, specify the channel diameter in the **Settings** rollout and then choose the **OK** button from the dialog box.

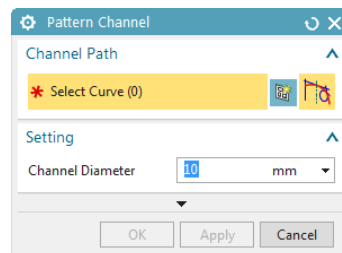


Figure 16-59 *The Pattern Channel dialog box*

Direct Channel

Ribbon: Mold Wizard > Cooling Tools gallery > Direct Channel



This tool is used to create the cooling channel or baffle by defining a specified point. To create a channel, choose the **Direct Channel** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Direct Channel** dialog box will be displayed, refer to Figure 16-60. Also, you will be prompted to select the infer point. Select the start point of the channel. Next, select the **Distance** option from the **Motion** drop-down list of the **Channel Extrusion** rollout; you will be prompted to specify the vector direction and then enter the length of the channel in the **Distance** edit box. Choose the **OK** button to close the dialog box.

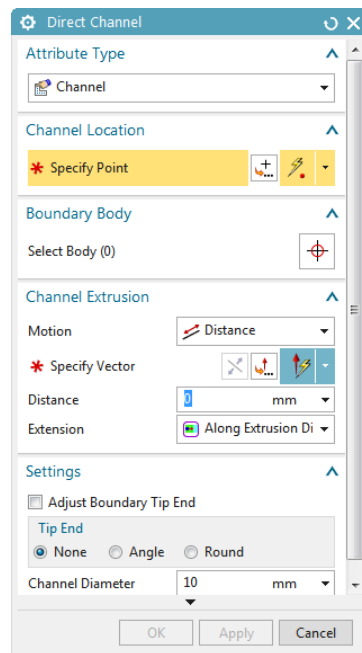


Figure 16-60 The Direct Channel dialog box

Extend Channel

Ribbon: Mold Wizard > Cooling Tools gallery > Extend Channel



This tool is used to extend the length of the cooling channel. To extend the length, choose the **Extend Channel** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Extend Channel** dialog box will be displayed, refer to Figure 16-61, and you will be prompted to select the cooling channel. Select the cooling channel and specify the length of the cooling channel in the **Distance** edit box. Choose the **OK** button; the length will be extended and the dialog box will be closed.

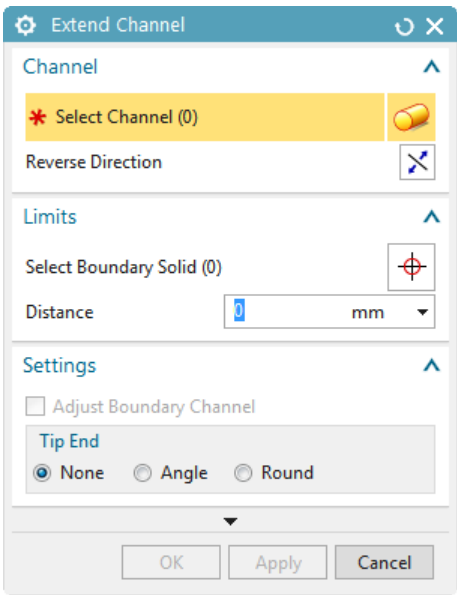



Figure 16-61 The *Extend Channel* dialog box

Connect Channels

Ribbon: Mold Wizard > Cooling Tools gallery > Connect Channels

 This tool is used to connect the cooling channels. To do so, choose the **Connect Channels** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Connect Channels** dialog box will be displayed, refer to Figure 16-62. Also, you will be prompted to select the first cooling channel. Select the first cooling channel; you will be prompted to select the second cooling channel. Select the second cooling channel and then choose the **OK** button to close the dialog box.

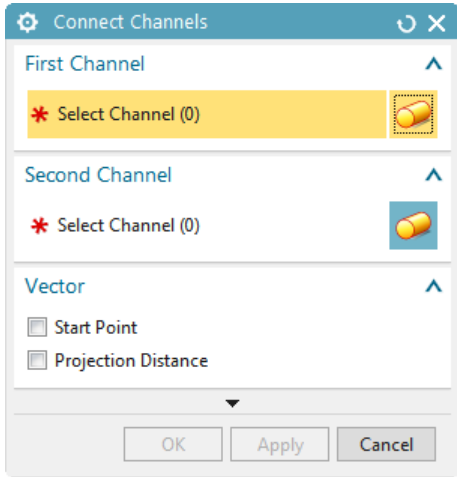


Figure 16-62 The *Connect Channels* dialog box

Adjust Channel

Ribbon: Mold Wizard > Cooling Tools gallery > Adjust Channel



This tool is used to adjust the position of the cooling channel. To do so, choose the **Adjust Channel** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Adjust Channel** dialog box will be displayed, refer to Figure 16-63. Also, you will be prompted to select the cooling channel. Select the cooling channel which you want to adjust; a dynamic triad will be displayed on the channel. Move the channel by using handles and angular handles of the triad. After positioning the channel, choose the **OK** button to close the dialog box.

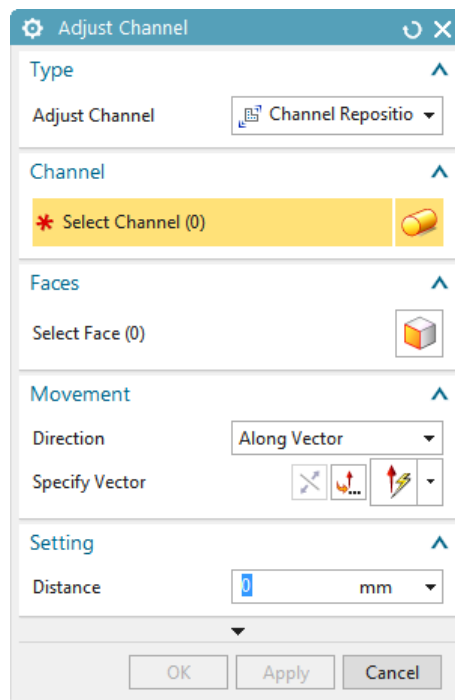


Figure 16-63 The *Adjust Channel* dialog box

Cooling Fittings

Ribbon: Mold Wizard > Cooling Tools gallery > Cooling Fittings



This tool is used to add cooling fitting components to cooling channels. To add the components, choose the **Cooling Fittings** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Cooling Fittings** dialog box will be displayed, refer to Figure 16-64, and you will be prompted to select the channel. Select the cooling channel and then choose the **OK** button; the dialog box will be closed and you will notice the o-rings in the mold cooling channel.

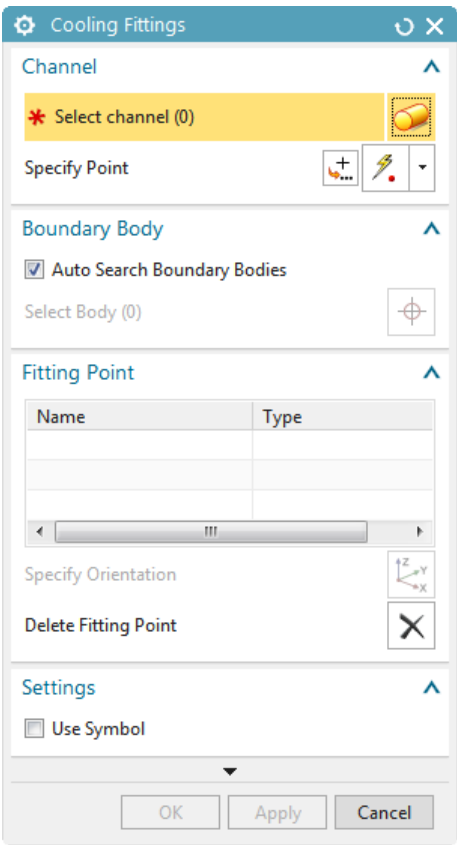


Figure 16-64 The *Cooling Fittings* dialog box

Assembly Drawings

Ribbon: Mold Wizard > Mold Drawing gallery > Assembly Drawing



This tool helps you to create the drawing of the mold assembly. To create the drawing, choose the **Assembly Drawing** tool from the **Mold Drawing** gallery of the **Mold Wizard** tab; the **Assembly Drawing** dialog box will be displayed, refer to Figure 16-65. By default, the **Visibility** option is selected in the **Type** rollout. This option helps you to select the parts of the mold you want to show in the drawing. Select the component from the drawing area and then select the **Drawing** option from the drop-down list of the **Type** rollout. Select the **Self Contained** drawing type from the **Drawing Type** rollout and select the type of template from the **Templates** rollout. Choose the **Apply** button. Select the **View** option from the drop-down list of the **Type** rollout and then select the predefined views from the **View Control** rollout. Specify the scale value in the **Scale** edit box of the **View Control** rollout. Choose the **OK** button to close the dialog box.

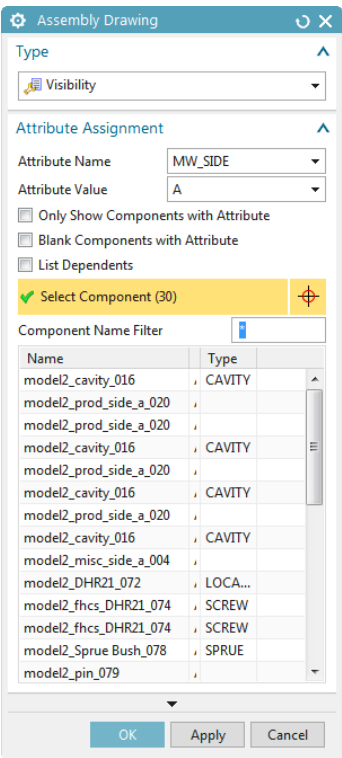



Figure 16-65 The Assembly Drawing dialog box

Component Drawing

Ribbon: Mold Wizard > Mold Drawing gallery > Component Drawing

 This tool helps you to create and manage the drawing for components of the mold assembly. To create a drawing, choose the **Component Drawing** tool from the **Mold Drawing** gallery of the **Mold Wizard** tab; the **Component Drawing** dialog box will be displayed, refer to Figure 16-66. Select the component from the list in the **Drawing** rollout and then choose the **Create Drawing** button from the **Create Drawing** area of the **Drawing** rollout; the drawing will be generated.

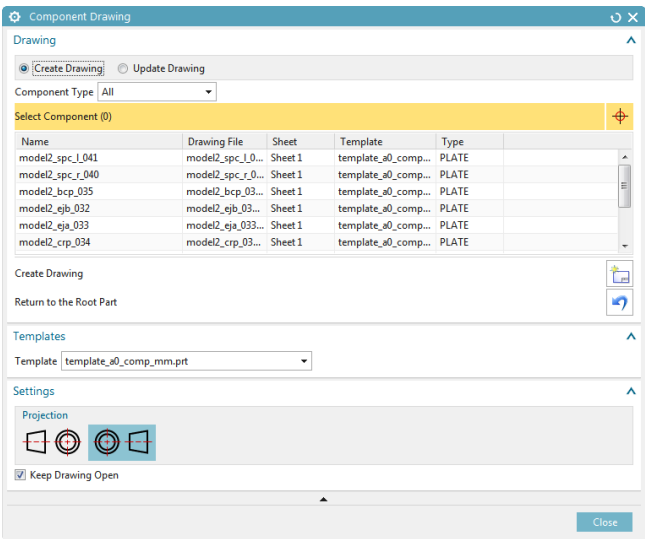


Figure 16-66 The Component Drawing dialog box

TUTORIAL

To perform the tutorial, you need to download the zipped file named as *c16_NX_11.0_input* from the Input Files section of the CADCIM website. The complete path for downloading the file is:

Textbooks > CAD/CAM > NX > NX 11.0 for Designers > Input Files

After the file is downloaded, extract the folder to the location *C:\NX 11.0* and rename it as *c16*.

Tutorial 1

In this tutorial, you will create the mold design of the plastic cover part shown in Figure 16-67. Then, you will analyze the model and add mold base to it. Next, you will create core, cavity, runner, and gate. Also, you will add lifter, ejector pin, and cooling channels to the model.

After creating the mold design, save it with the name *c16tut1.prt* at the location:
|NX|c16 (Expected time: 3 hr)

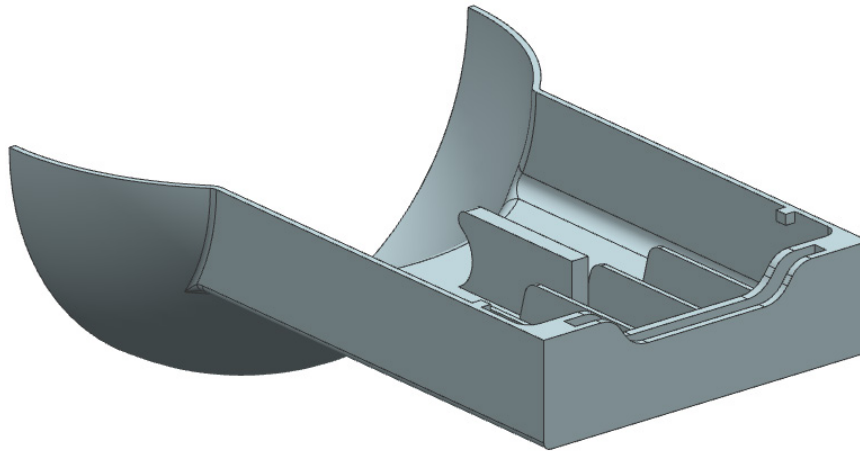



Figure 16-67 Plastic part for Tutorial 1

The following steps are required to complete this tutorial:

- a. Start NX and open the model, refer to Figure 16-68.
- b. Create a gate for analyzing the model, refer to Figure 16-74.
- c. Create runner and sprue for analyzing the model, refer to Figure 16-77.
- d. Set the parting direction.
- e. Start analysis and analyze the result, refer to Figures 16-85 to 16-90.
- f. Initialize the project.
- g. Orient the model, refer to Figures 16-94 and 16-95.
- h. Create the parting surface, refer to Figures 16-99 to 16-103.
- i. Create core and cavity, refer to Figures 16-110 and 16-111.
- j. Add the mold base, refer to Figure 16-113.
- k. Add lifter for ejection and part shape, refer to Figure 16-116.
- l. Add the register ring and sprue bush in mold, refer to Figures 16-118 and 16-120.
- m. Add the ejector pin for ejection of component, refer to Figure 16-123.
- n. Create gate and runner, refer to Figures 16-128 and 16-129.
- o. Create cooling channel for cooling the component, refer to Figure 16-132.

Starting NX and Opening a Model

First, you need to start NX and then open a new file.

1. Double-click on the shortcut icon of NX on the desktop of your computer to start NX.
2. Choose the **Open** button from the **Standard** group of the **Home** tab or choose **Menu > File > Open** from the **Top Border Bar**; the **Open** dialog box is displayed. 
3. Select the **Cover_Plastic_prt** from the **Name** list; **Cover_Plastic_prt** is displayed in the **File name** drop-down list. Then, choose the **OK** button; the model is displayed, refer to Figure 16-68.

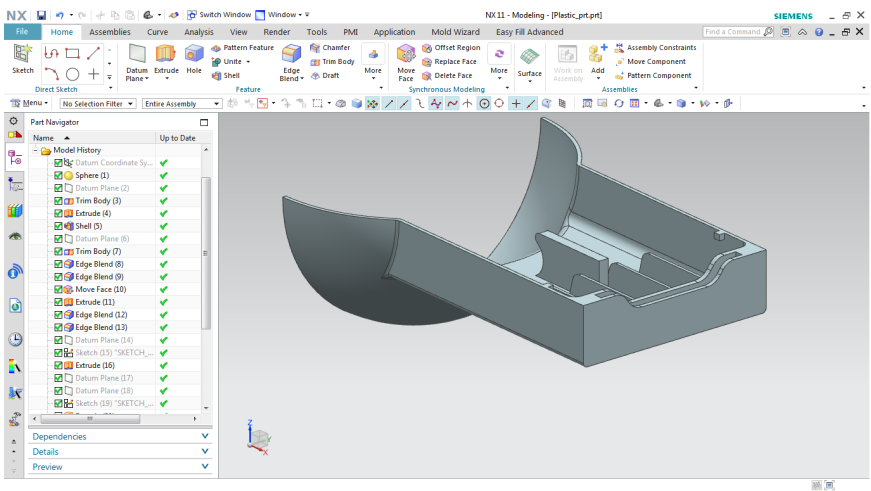


Figure 16-68 Plastic part in Modeling environment

Analyzing the Model

In this section, you will analyze the model by using the tools from the **Easy Fill Advanced** tab. Sometimes the material is not filled properly in the model. In such a case, you need to perform iterations by changing the gate, runner, coolant size, shape, or orientation.

- 1. Choose the **Easy Fill Advanced** tab. Next, choose the **Set Working Folder** tool from the **Setting** group of the tab; the **Working Folder** dialog box is displayed.
- 2. Choose the **Browse** button; the **Open** dialog box is displayed where you can set the working folder, refer to Figure 16-69.

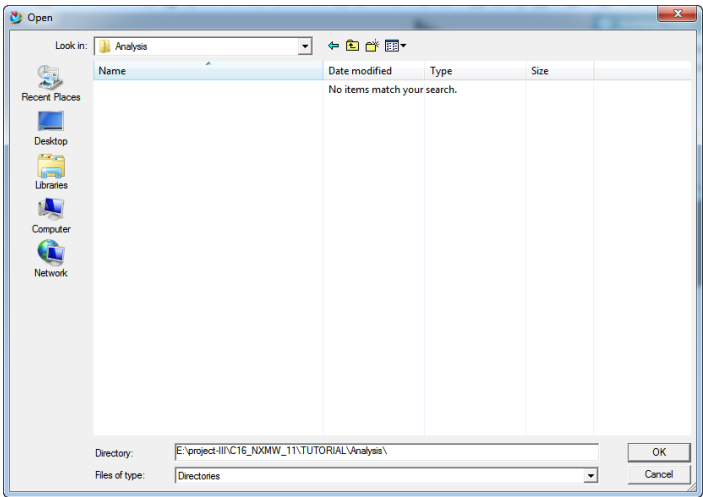


Figure 16-69 The Open dialog box

- 3. Choose the **OK** button from the **Open** dialog box and then from the **Working Folder** dialog box.

Creating the Gate

In this section, you will create a gate.

1. Choose the **Set Cavity** tool from the **Setting** group of the **Easy Fill Advanced** tab in the **Ribbon**; the **Select Cavity** dialog box is displayed, refer to Figure 16-70.

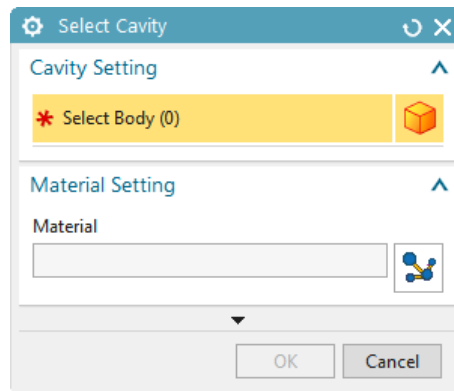


Figure 16-70 The Select Cavity dialog box

2. Select the solid body from the **Select Body** area of the **Cavity Setting** rollout to specify the solid body as cavity.
3. Choose the **Push button to select material** button in the **Material Setting** rollout; the **Moldex3D Material Wizard** dialog box is displayed, refer to Figure 16-71.

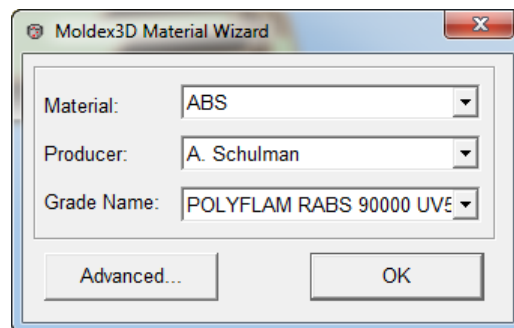


Figure 16-71 The Moldex3D Material Wizard dialog box

4. Select **ABS+PA6** from the **Material** drop-down list.
5. Select **Styrolution** from the **Producer** drop-down list.
6. Select **Terblend N NG-06** from the **Grade Name** drop-down list.
7. Choose the **OK** button from the **Moldex3D Material Wizard** and then choose the **OK** button from the **Select Cavity** dialog box.

- 8. Choose the **Gate Wizard** tool from the **Wizard** group; the **Create Gate** dialog box is displayed, refer to Figure 16-72.
- 9. Select **Fan Gate** from the **Type of gate** drop-down list to select the type of gate.
- 10. Select **Cold Runner Gate** from the **Attribute** drop-down list.
- 11. Choose the **Point Dialog** button; the **Point** dialog box is displayed, refer to Figure 16-73.

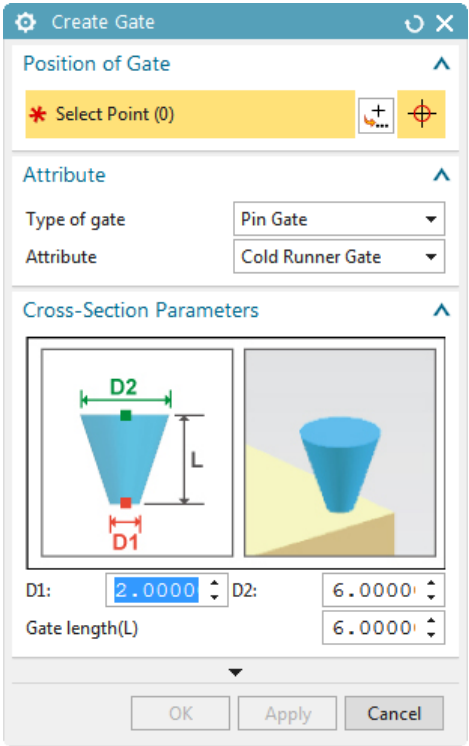


Figure 16-72 The *Create Gate* dialog box

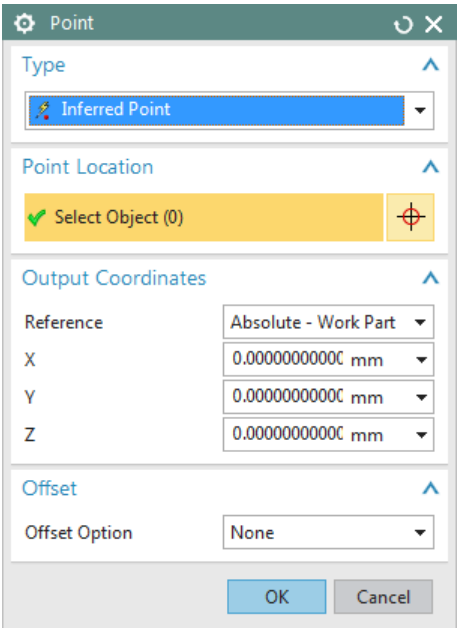


Figure 16-73 The *Point* dialog box

- 12. Select the **Point on Face** option from the **Type** drop-down list; you are prompted to select a face at point location, refer to Figure 16-74.
- 13. Enter **0.308** and **0.5** in the **U Parameter** and **V Parameter** edit boxes of the **Location on Face** rollout and choose the **OK** button from the **Point** dialog box to exit it.
- 14. Enter **2,2,4,8,180** and **5** in the **a1, a2, b1, b2, Angle** and **Gate length(L)** edit boxes of the **Cross-Section Parameters** rollout.
- 15. Choose the **OK** button to close the **Create Gate** dialog box. Refer to Figure 16-75 for model with fan gate.

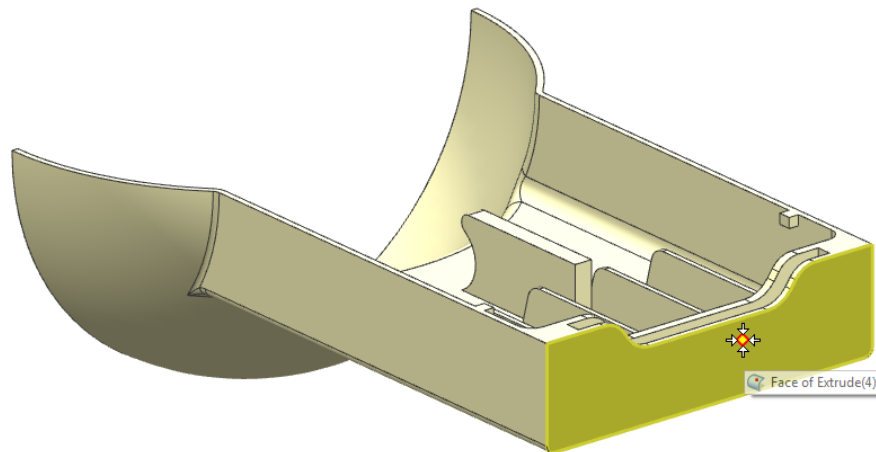


Figure 16-74 The face selected for gate placement

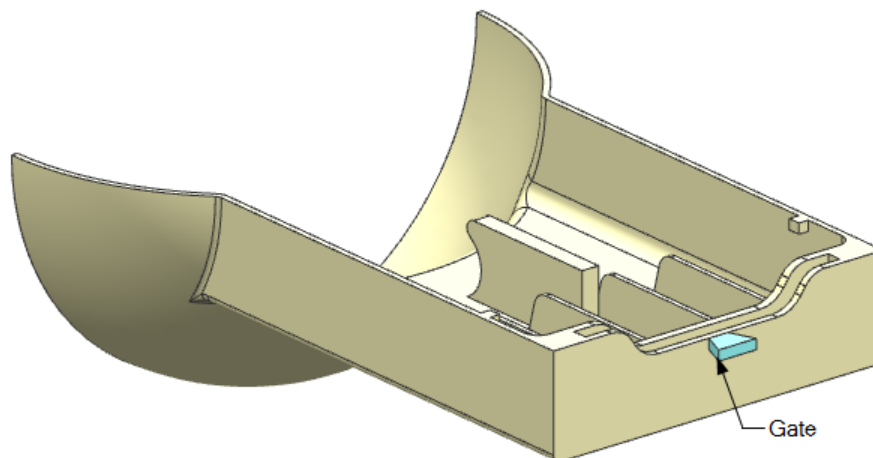


Figure 16-75 The model with fan gate

Creating the Runner and Sprue

In this section, you will create the runner and sprue.

1. Choose the **Runner Wizard** tool from the **Wizard** group; the **Runner Wizard** dialog box is displayed, refer to Figure 16-76.



By default, the **Mold Setting** tab is chosen.

2. Select **-Z-Axis** from the **Parting Direction** drop-down list, **2-Plate mold** from the **Mold Plate Type** drop-down list, and **Cold Runner** from the **Runner Attribute** drop-down list.
3. Choose the **Sprue Setting** tab and enter **10,7**, and **45** in the **D1**, **D2**, and **SH** edit boxes of the **Sprue Geometry Parameters** rollout.

4. Choose the **Runner Setting** tab and then select the **Circular** type of cross section for runner from the **Type** drop-down list. Next, enter **10** in the **D** edit box.
5. Choose the **OK** button to close this dialog box, refer to Figure 16-76.

The gate, runner, and sprue are now attached to the model, refer to Figure 16-77.

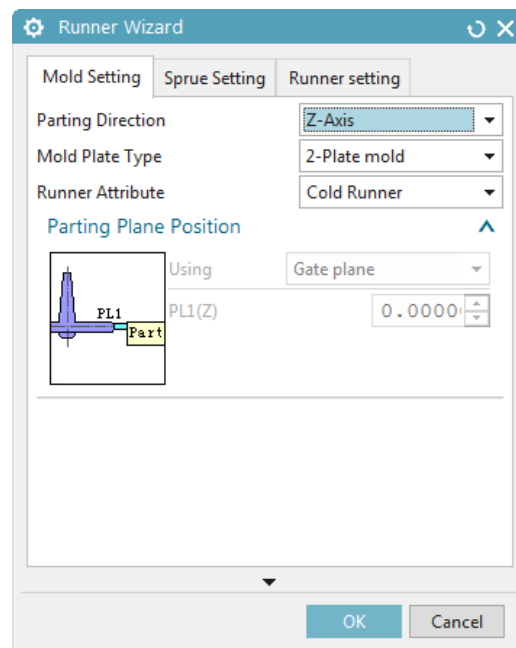


Figure 16-76 The Runner Wizard dialog box

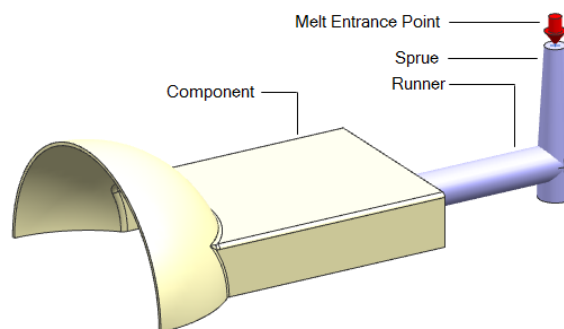


Figure 16-77 The runner and sprue with model

Setting the Parting Direction

In this section, you will going to set the direction of ejection.

1. Choose the **Set Parting Direction** tool from the **Settings** group to set the parting direction of the model; the **vector** dialog box is displayed, refer to Figure 16-78.
2. Select the **ZC-Axis** option from the **Type** rollout and choose the **OK** button to close the dialog box.

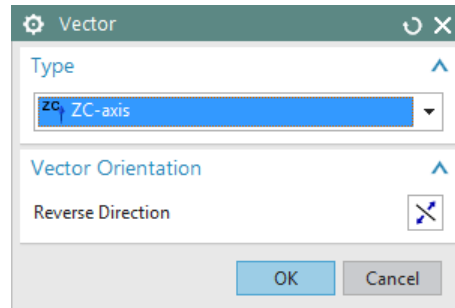


Figure 16-78 The *Vector* dialog box

Starting Analysis and Analyzing the Result

In this section, you will start the analysis and show results.

1. Choose the **Start Analysis** tool from the **Setting** group to start the analysis of the model; the **Start Analysis** dialog box is displayed, refer to Figure 16-79.

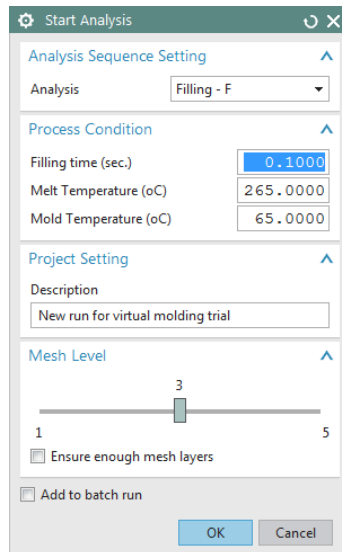


Figure 16-79 The *Start Analysis* dialog box

- 2. Select the **Filling & Packing - F P** option from the **Analysis** drop-down list of the **Analysis Sequence Setting** rollout.
- 3. Enter **220**, **0.25**, **4.20**, **250**, and **60** in the **Maximum injection pressure**, **Filling time**, **Packing time**, **Melt Temperature**, and **Mold Temperature** edit boxes, respectively, of the **Process Condition** rollout.
- 4. Choose the **OK** button; the dialog box closes and the **Moldex3D** window showing the progress of solid mesh generation is displayed, refer to Figure 16-80.

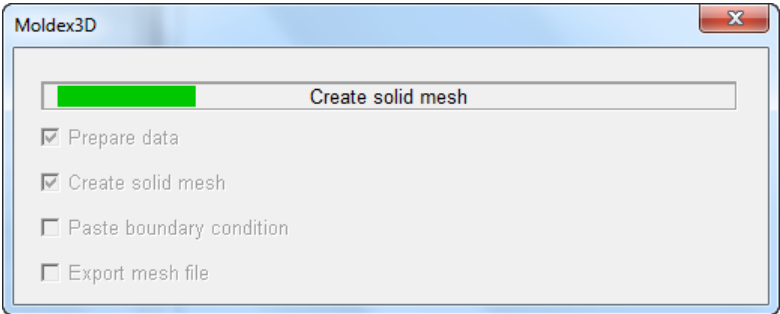


Figure 16-80 The Moldex3D window

Note that after the solid mesh is created; the **Easy Fill Advanced Project Monitor** window showing the progress of filling and packing is displayed, refer to Figure 16-81. After completion of analysis, the **Moldex3D eDesignSYNC** window gets displayed, refer to Figure 16-82.

- 5. Choose the **Close** button in the **Easy Fill Advanced Projector Monitor** window to close it.

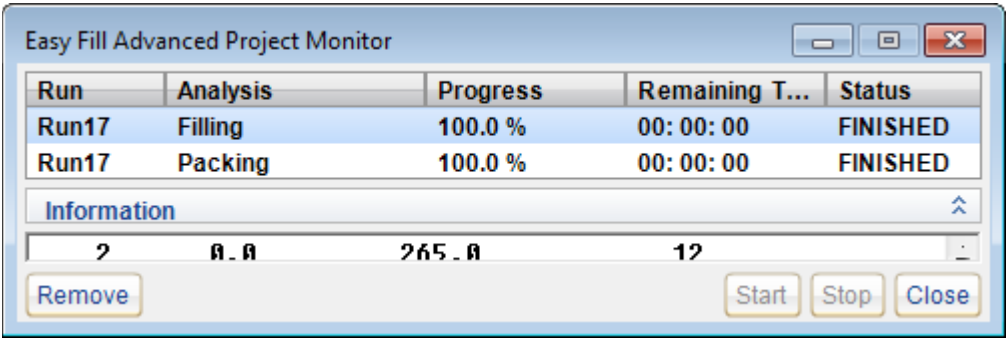


Figure 16-81 The Easy Fill Advanced Project Monitor window

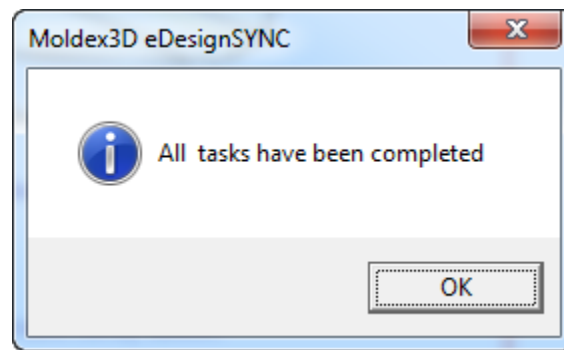


Figure 16-82 The Moldex3D eDesignSYNC window

6. Choose the **Show Result** tool from the **Settings** group; the **Show Result** dialog box gets displayed, refer to Figure 16-83.

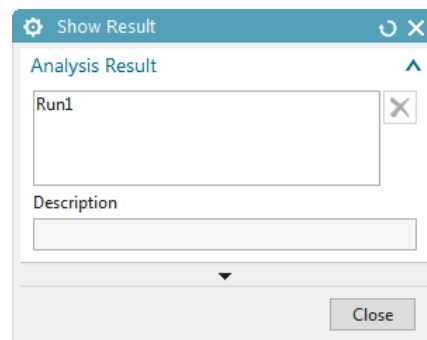


Figure 16-83 The Show Result dialog box

7. Select the result from the **Analysis Result** rollout; the dialog box gets modified, refer to Figure 16-84.

You can select the **Filling** or **Packing** option from the **Analysis** drop-down list in the **Display Result** rollout. By default, the **Filling** option is selected.

In this tutorial, you can visualize the results by selecting appropriate option from the **Result Type** drop-down list.

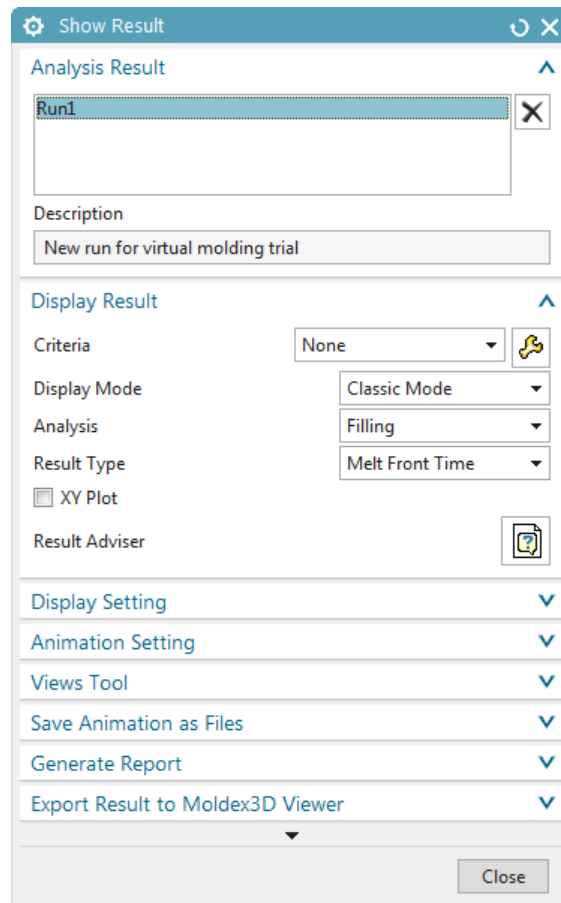


Figure 16-84 The modified Show Result dialog box

The options which can be selected for results are **Melt Front Time**, **Air Trap**, **Weld Line**, **Volumetric Shrinkage**, **Maximum Shear Stress**, and **Sink Marks Indicator**.

8. Select the **XY Plot** check box to activate the **XY Curve Type** drop-down list. Next, select the **Sprue Pressure** or **Clamping Force** option from this drop-down list; the resultant graphs are shown, refer to Figure 16-85 through Figure 16-90.

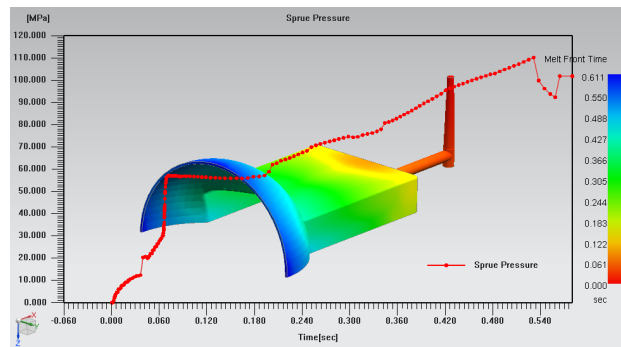


Figure 16-85 The Sprue Pressure vs Melt Front Time graph

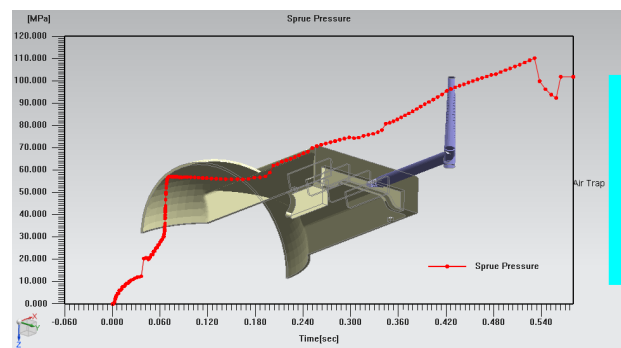


Figure 16-86 The Sprue Pressure vs Air Trap graph

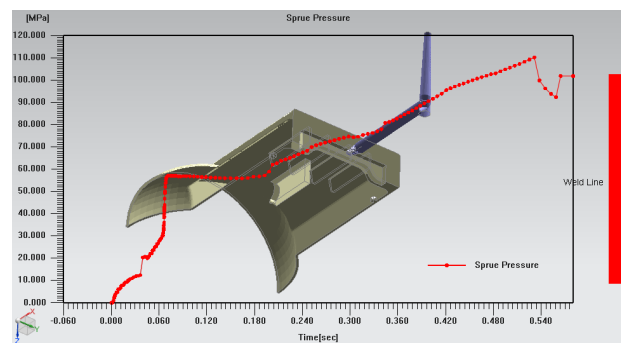


Figure 16-87 The Sprue Pressure vs Weld Line graph

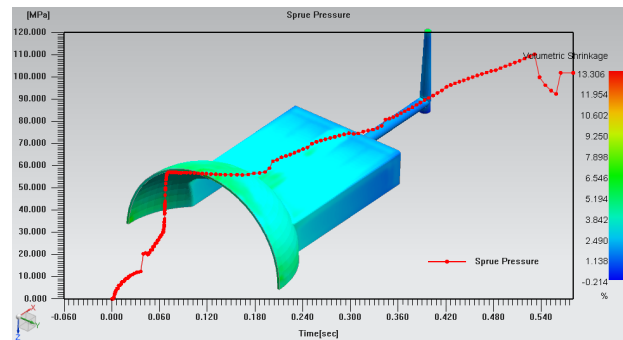


Figure 16-88 The Sprue Pressure vs Volumetric Shrinkage graph

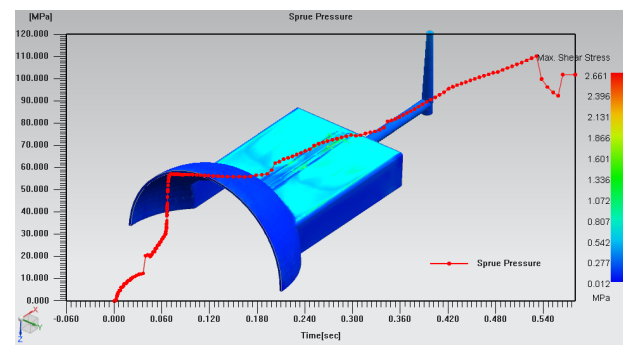


Figure 16-89 The Sprue Pressure vs Max. Shear Stress graph

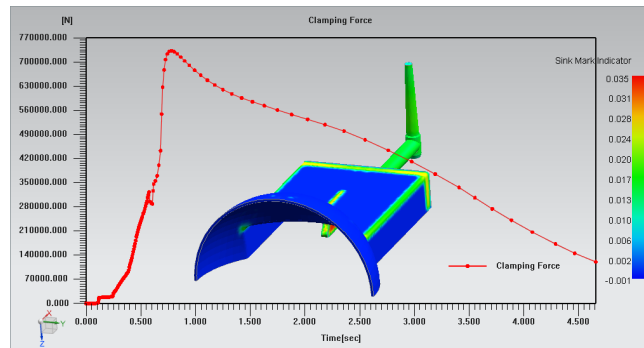


Figure 16-90 The Clamping Force vs Sink Mark Indicator graph

9. Choose the **Play** button from the **Animation Setting** rollout. Next, choose the **Close** button to close the dialog box.



10. Save the file to the specified folder by choosing the **Save All** button and closing NX.

Next, you need to open the model again. To open a file, choose the **Open** button from the **Standard** group of the **Home** tab or choose **Menu > File > Open** from the **Top Border Bar**; the **Open** dialog box is displayed. Select **Cover_Plastic** from the **Name** list; **Cover_Plastic** gets displayed in the **File name** drop-down list. Then choose the **OK** button; the model is displayed.

Initializing the Project Using the Mold Wizard Tab

In this section, you will use the **Initialize Project** tool to define the material and the path for saving the project.

1. Choose the **Mold Wizard** tab.
2. Choose the **Initialize Project** tool from the **Mold Wizard** tab; the **Initialize Project** dialog box is displayed, as shown in Figure 16-91.

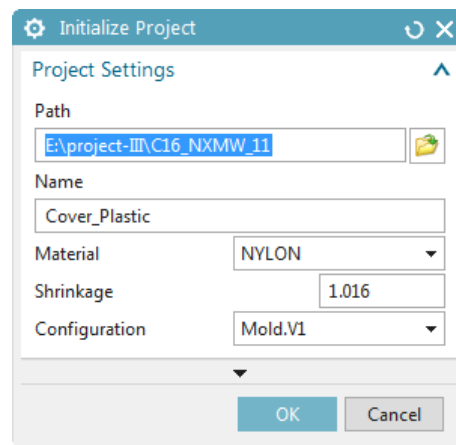


Figure 16-91 The Initialize Project dialog box

3. Choose the **Browse** button from the **Path** area of the **Project Settings** rollout; the **Open** dialog box is displayed, as shown in Figure 16-92. After setting the path of the project, choose the **OK** button to close the dialog box.
4. Select the **Nylon** material from the **Material** drop-down list; the **Shrinkage** edit box gets updated automatically depending upon the type of selection of material. Next, select the desired template from the **Configuration** drop-down list. By default, the **Mold.V1** template is selected. Choose the **OK** button to close the dialog box; the **New Iray + Ray Traced Studio Rendering** dialog box is displayed. Close this dialog box.

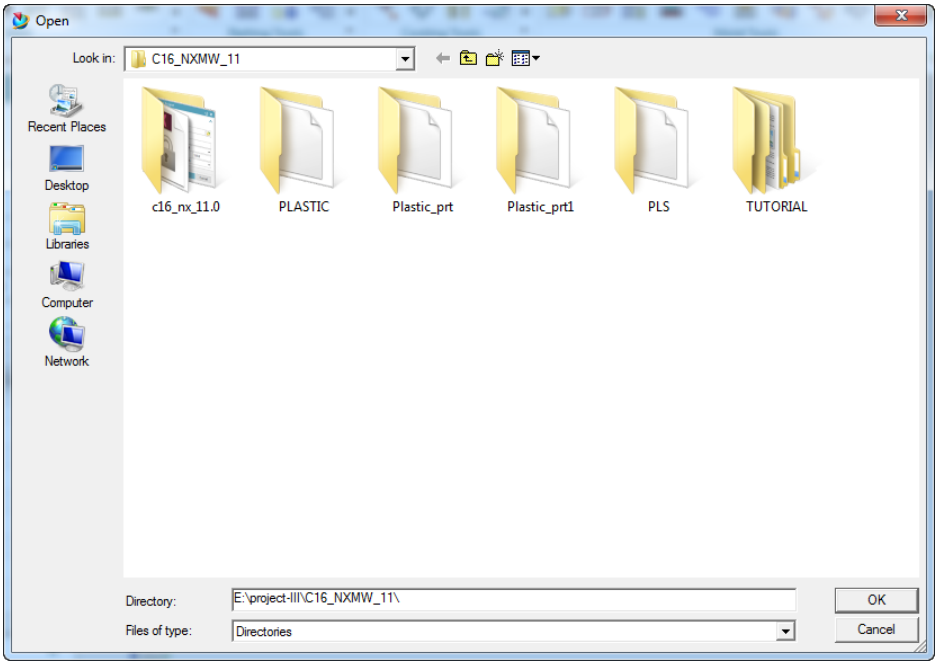


Figure 16-92 The *Open* dialog box

Reorienting the Model

In this section, you will use the **Mold CSYS** tool to reorient the model.

- 1. Choose the **Mold CSYS** tool from the **Main** group of the **Mold Wizard** tab; the **Mold CSYS** dialog box is displayed, as shown in Figure 16-93. Also, you are prompted to double-click on WCS.

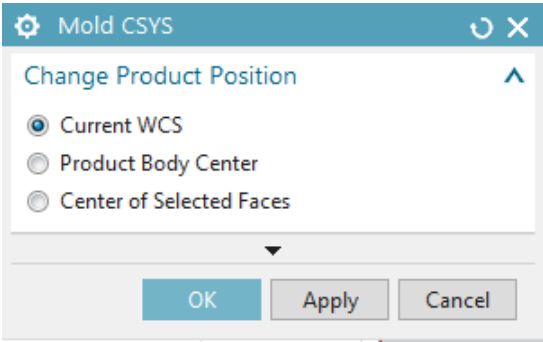


Figure 16-93 The *Mold CSYS* dialog box

2. Double-click on the WCS, refer to Figure 16-94; you are prompted to drag a handle or select a handle. Select the handle of WCS; the **WCS Dynamics** dialog box is displayed.

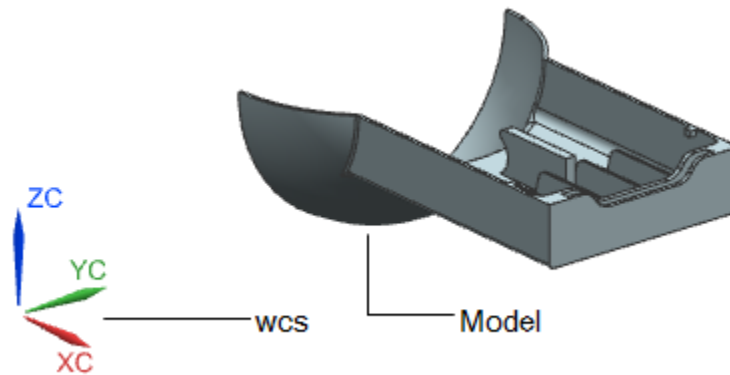


Figure 16-94 The WCS (work coordinate system)

3. Position the WCS, as shown in Figure 16-95.

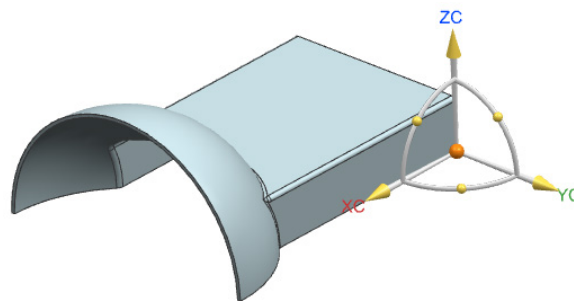



Figure 16-95 The WCS after positioning

4. Click the middle mouse button to close the **WCS Dynamics** dialog box.
5. Next, choose the **OK** button from the **Mold CSYS** dialog box to close the dialog box.

Checking the Core and Cavity Regions

You can use the **Check Regions** tool to check regions of core and cavity.

1. Invoke the **Check Regions** tool from the **Parting Tools** gallery; the **Check Regions** dialog box is displayed, refer to Figure 16-96. 
2. Choose the **Calculate** button from the **Calculate** rollout of the **Calculate** tab and then choose the **Region** tab. Choose the **Set Regions Color** button from the **Define Regions** rollout.

Note the color of the model gets changed. This color represents core, cavity, and undefined region in the model.

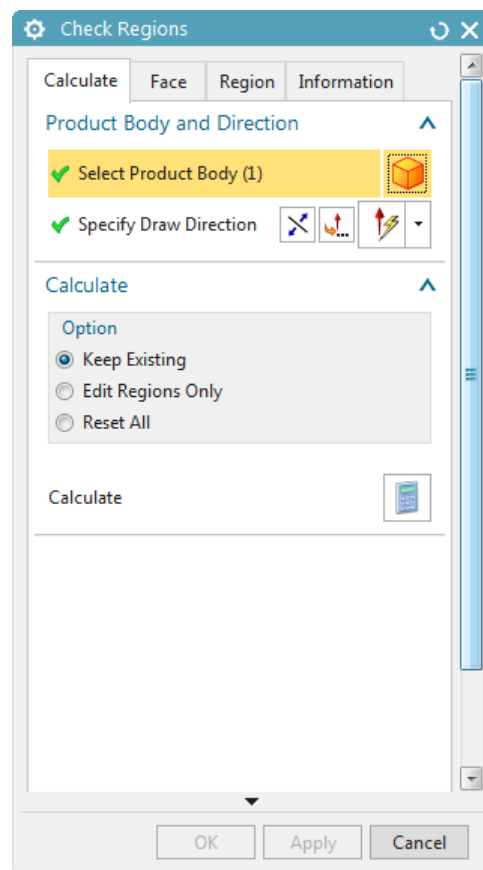


Figure 16-96 The Check Regions dialog box

3. The **Cavity Region** radio button is selected by default in the **Assign to Region** rollout. Select the undefined region of the model and then choose the **Apply** button; the undefined region gets the color of the cavity region.
4. Choose the **OK** button to close the dialog box. Modified model after using the **Check Region** tool is shown in Figure 16-97.

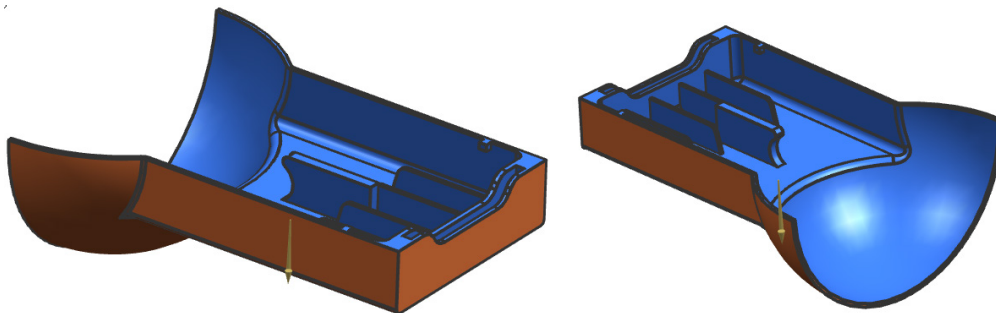


Figure 16-97 The modified model after using the Check Region tool

Creating Core and Cavity Region Sheets

To create parting lines, you will use the **Define Regions** tool.

1. Choose the **Define Regions** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Define Regions** dialog box is displayed with the **Parting Navigator** window, refer to Figure 16-98.

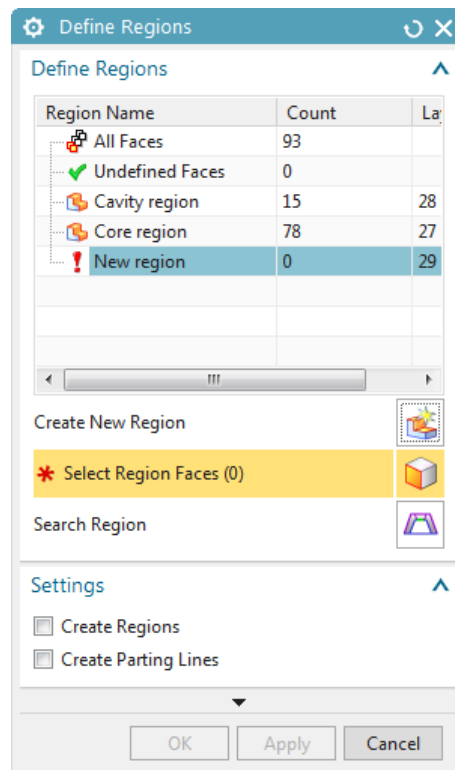


Figure 16-98 The Define Regions dialog box

2. Select **Cavity region** in the **Define Regions** rollout and then select the **Create Regions** and the **Create Parting Lines** check boxes from the **Settings** rollout.
3. Choose the **Apply** button; a green tick mark appears beside the cavity and core region.
4. Choose the **Cancel** button to close the **Define Regions** dialog box.
5. Clear all check boxes from the **Parting Navigator** window except the **Parting Lines** check box; parting lines get created in the window, refer to Figure 16-99.

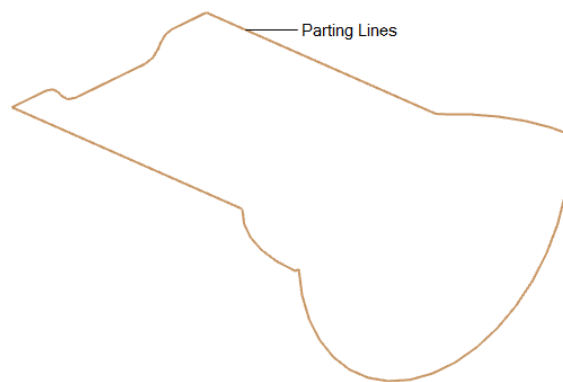


Figure 16-99 Parting Lines on the component

Creating a Parting Surface

To create a parting surface, you need to use the **Design Parting Surface** tool.

1. Choose the **Design Parting Surface** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Design Parting Surface** dialog box is displayed, refer to Figure 16-100.
2. Expand the dialog box by clicking on the down arrow.
3. Expand the **Edit Parting Segments** rollout and choose the **Edit Guide Lines** button. Next, select the parting line; the guide line is created, refer to Figure 16-101. Similarly, create other guide lines, refer to Figure 16-102.

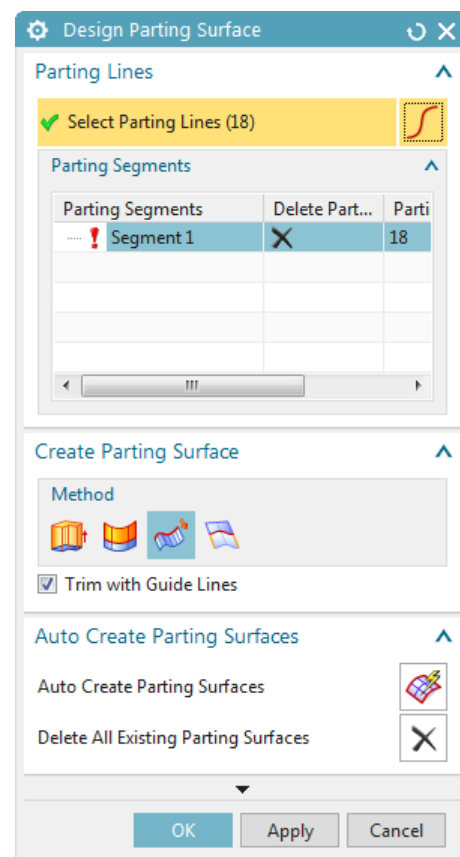


Figure 16-100 The Design Parting Surface dialog box

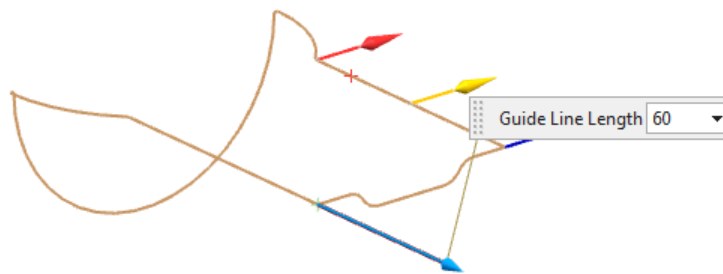


Figure 16-101 The guide line created

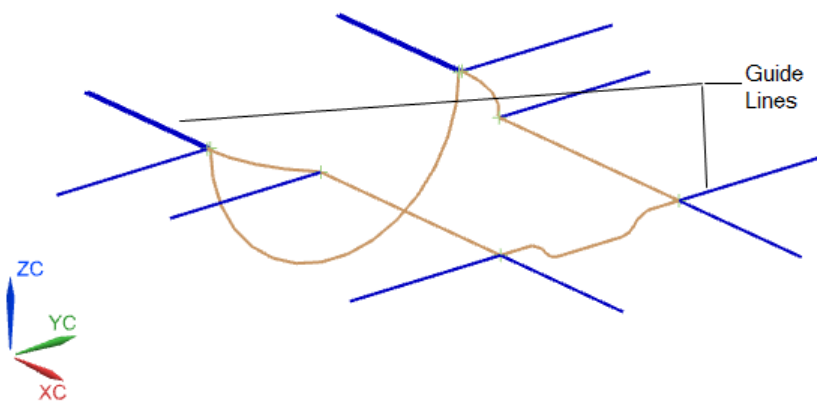


Figure 16-102 The guide lines created

4. Select **Segment 1** from the **Parting Segments** sub-rollout of the **Parting Lines** rollout. Next, choose the **Swept** button from the **Method** area of the **Creating Parting Surface** rollout. Specify the first and second directions. Enter **250** as the value in the **Extend Distance** dynamic edit box and choose the **Apply** button. Similarly, create parting surface for other portion of the part, refer to Figure 16-103.

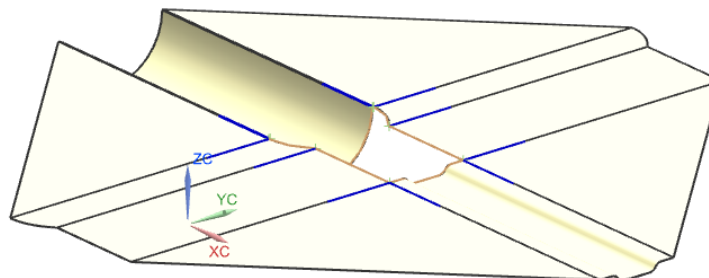


Figure 16-103 Parting surface of the component

5. Choose the **Cancel** button to close the dialog box and then choose the **Save All** button to save the file.

Creating the Workpiece

The **Workpiece** tool helps you to decide the size of the core and cavity insert.

1. Choose the **Workpiece** tool from the **Main** gallery of the **Mold Wizard** tab; the **New Iray + Ray Traced Studio Rendering** window is displayed. Choose the **OK** button; the **Workpiece** dialog box is displayed, refer to Figure 16-104. By default, the **Product Workpiece** option is selected in the **Type** drop-down list of the **Type** rollout and the **User Defined Block** option is selected in the **Workpiece Method** drop-down list. Refer to Figure 16-105 for the size of the core and cavity insert.

Note the size of insert is automatically defined.

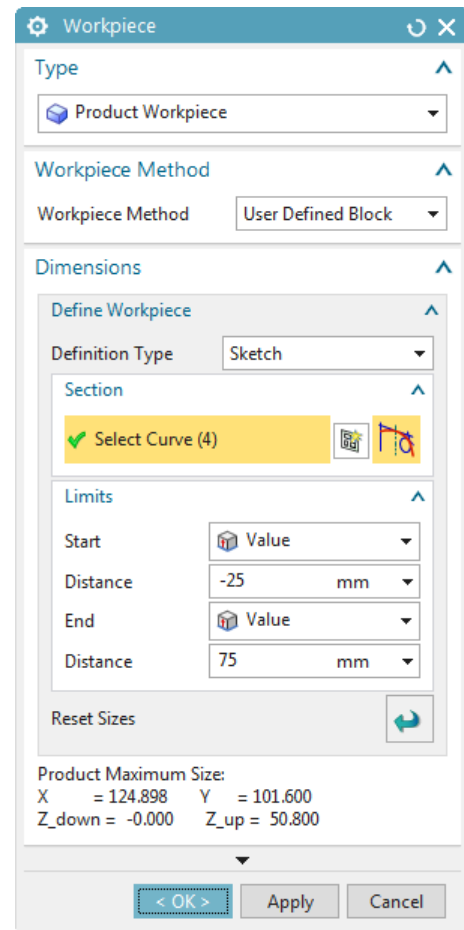


Figure 16-104 The Workpiece dialog box

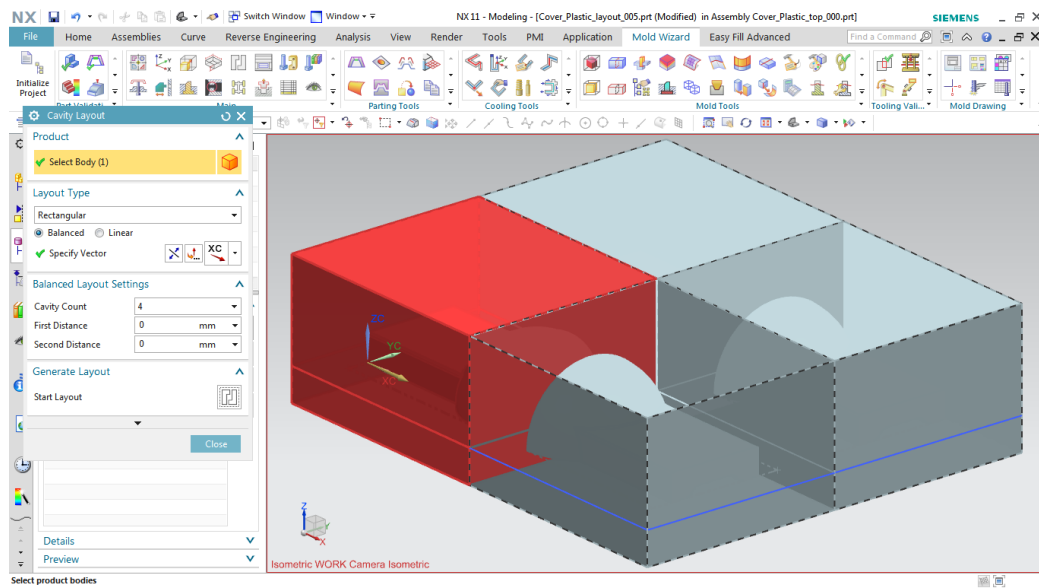


Figure 16-107 The cavity layout of the workpiece

6. Select **5** from the **R** drop-down list and **2** from the **type** drop-down list. Next, choose the **OK** button to close the dialog box.
7. Choose the **Close** button to close the **Cavity Layout** dialog box.

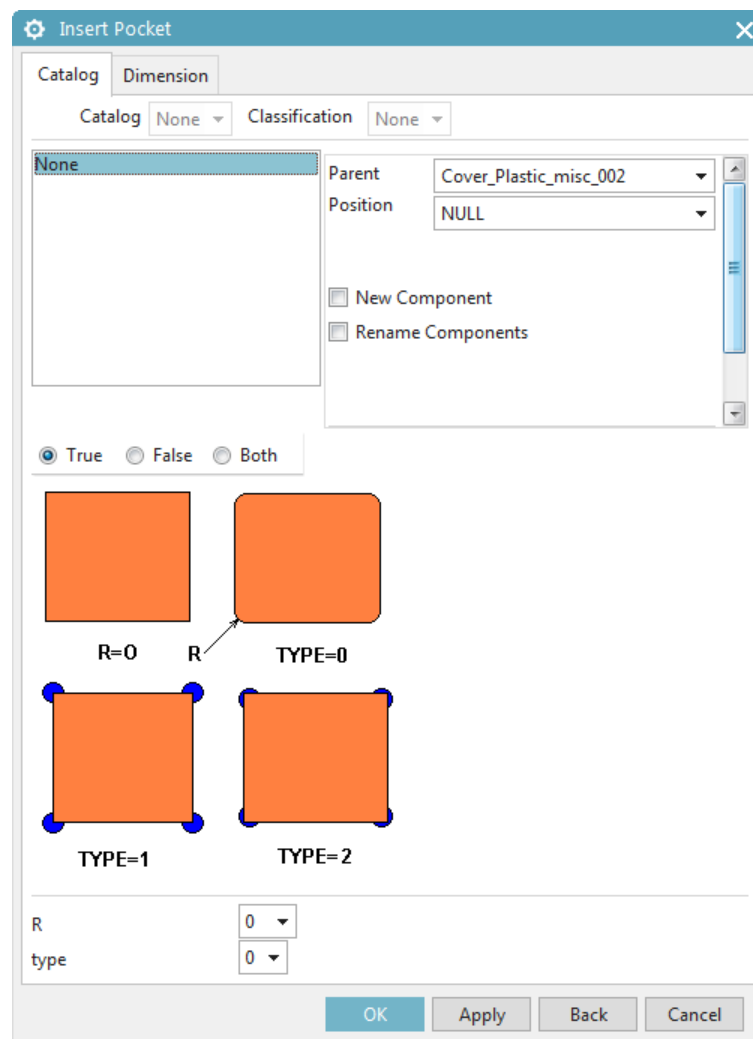


Figure 16-108 The Insert Pocket dialog box

Creating the Core and Cavity

In this section, you will create the core and cavity by using the **Define Cavity and Core** tool.

1. Choose the **Define Cavity and Core** tool from the **Parting Tools** gallery of the **Mold Wizard** tab; the **Define Cavity and Core** dialog box is displayed, refer to Figure 16-109. 

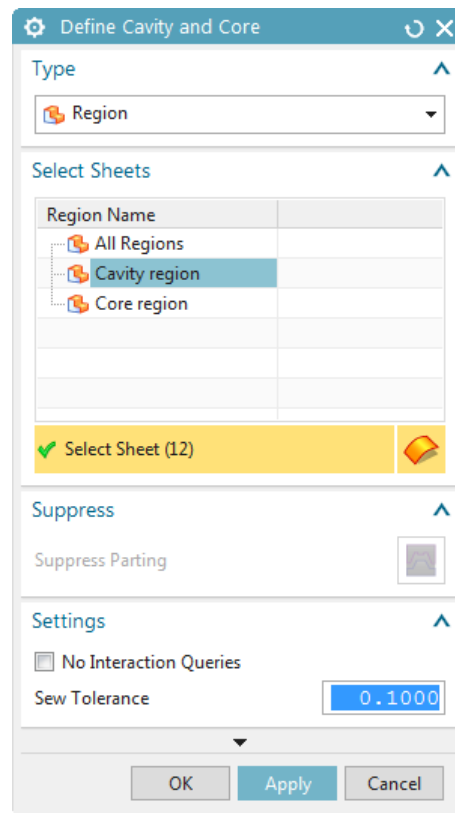


Figure 16-109 The Define Cavity and Core dialog box

By default, the **Region** option is selected in the **Type** drop-down list of the **Type** rollout and the **Cavity region** is highlighted in the **Select Sheets** rollout.

2. Choose the **Apply** button; the **New Iray + Ray Traced Studio Rendering** window is displayed. Choose the **OK** button; the **View Parting Result** dialog box and the cavity of the model get displayed, refer to Figure 16-110. Choose the **Reverse Direction** button to create other half of the cavity. Choose the **OK** button to close the dialog box; a green tick mark is displayed before the **Cavity region** in the **Select Sheets** rollout of the dialog box. Now, select **Core region** in the **Select Sheets** rollout of the dialog box and repeat the same procedure as for the cavity; the core is displayed, refer to Figure 16-111.
3. Choose the **Cancel** button to close the dialog box.
4. Select **Cover_Plastic_parting_014** in the assembly navigator and right-click on it; a shortcut menu is displayed. Choose **Cover_Plastic_top_000** from the **Display Parent** cascading menu.

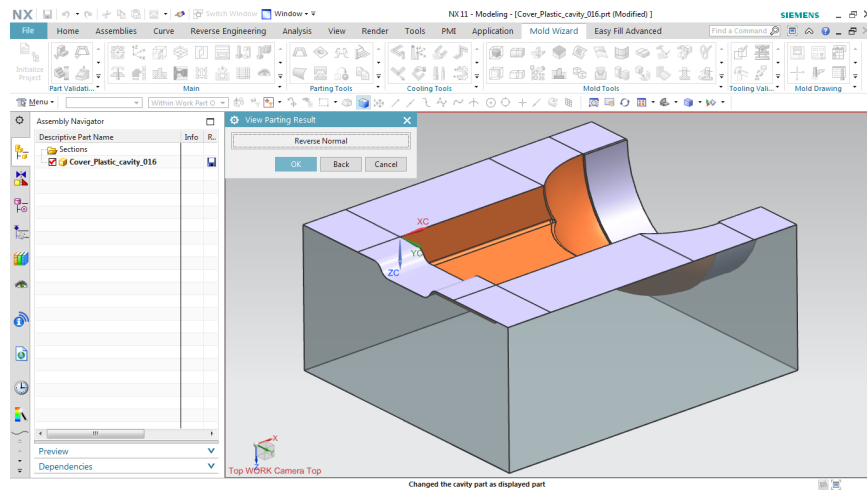


Figure 16-110 The View Parting Result dialog box and cavity of the model on screen

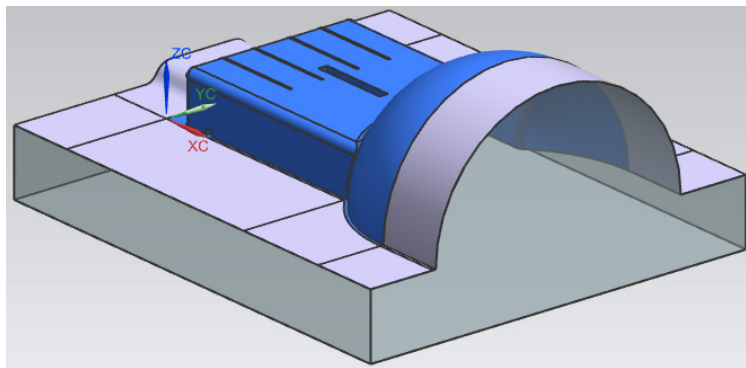


Figure 16-111 The core of the model

Adding Mold Assembly

Now, use the **Mold Base Library** tool to create the plates for the mold assembly.

1. Choose the **Mold Base Library** tool from the **Main** gallery of the **Mold Wizard** tab; the **Mold Base Library** dialog box is displayed, refer to Figure 16-112.
2. Expand the **MW Mold Base Library** from the **Main** panel. Select the **DME** folder. Right-click on **2A** in the **Member Select** panel and choose the **Insert** option; the **Mold Base Library** dialog box is displayed, refer to Figure 16-112 .
3. Select **5050**, **2**, **106**, **36**, and **106** from the **index**, **TCP_type**, **AP_h**, **BP_h**, and **CP_h** drop-down list respectively in the **Details** rollout. Next, choose the **OK** button to close the dialog box; the Mold plates are added to the assembly, refer to Figure 16-113.
4. Choose **File > Save > Save All** to save the file.

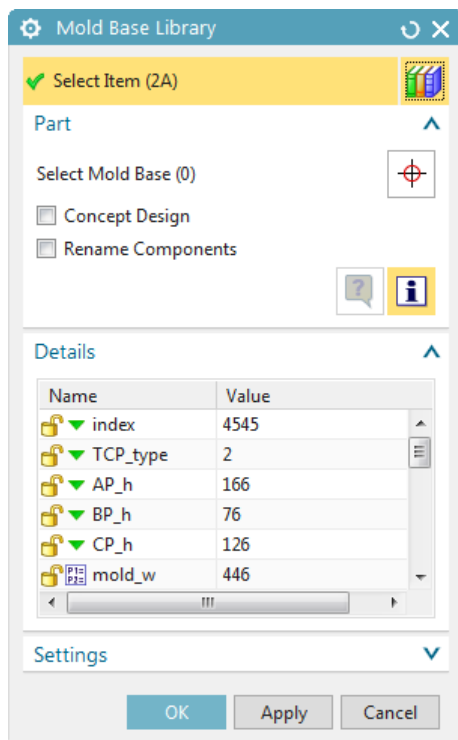


Figure 16-112 The Mold Base Library dialog box

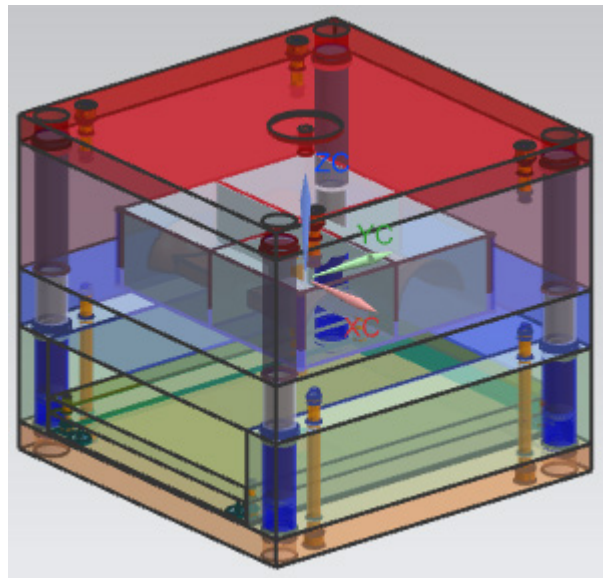


Figure 16-113 The Mold Plate assembly

Adding Lifter in Mold Assembly

Now, use the **Slide and Lifter Library** tool to create the lifter for mold assembly.

1. Choose the **Slide and Lifter Library** tool from the **Main** gallery; the **Slide and Lifter Design** dialog box is displayed with the **Information** window.
2. Select the **Lifter** folder in Main Panel and **Dowel Lifter** in the **Member Select** panel of the **Reuse Library**; the **Slide and Lifter Design** dialog box and the **Information** window gets modified, refer to Figure 16-114.

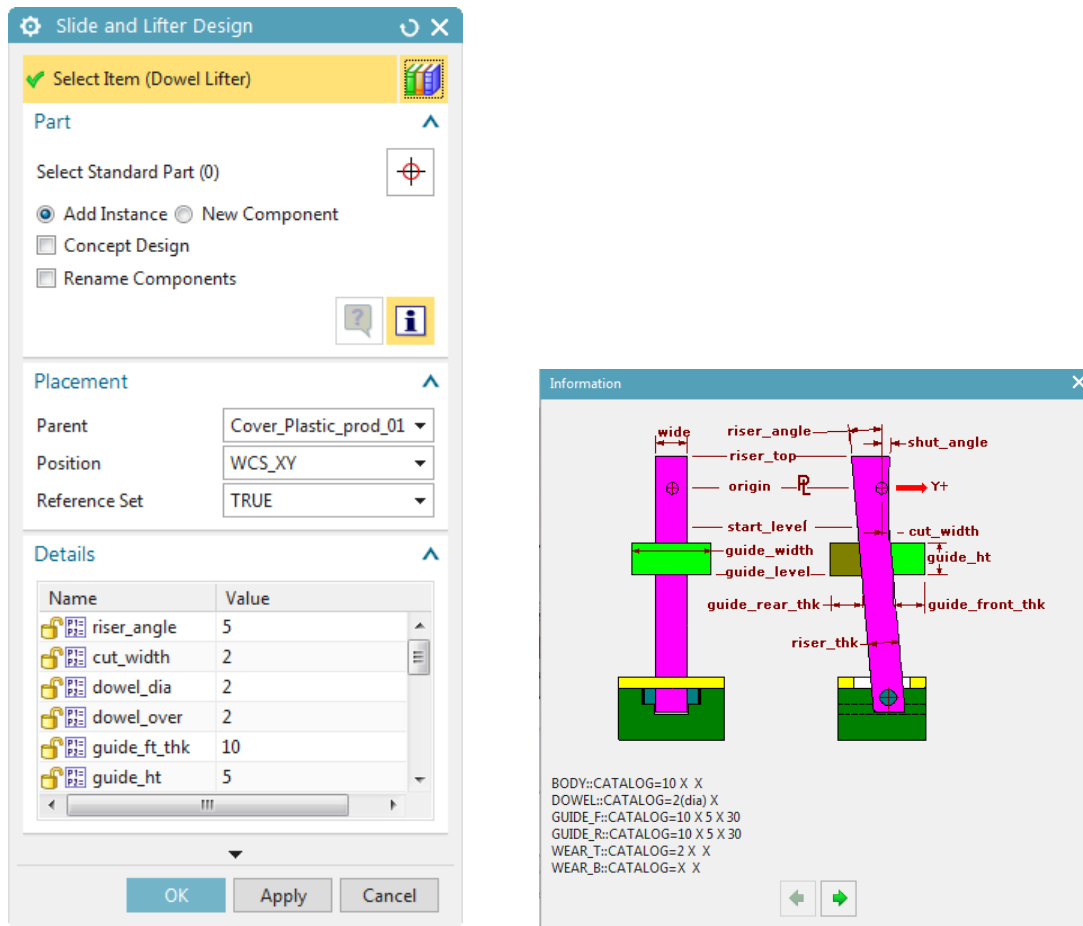


Figure 16-114 The *Slide and Lifter Design* dialog box and the *Information* window

- Specify the parameter of the Lifter in the **Details** rollout and choose the **Apply** button.
- Choose the **Select Standard Part** area of the **Part** rollout and then select the lifter which you want to place.
- Choose the **Reposition** button in the **Part** rollout; the **Move Component** dialog box is displayed, refer to Figure 16-115. You can place the lifter position using this dialog box.
- Choose the **OK** button after placing the lifter to close the dialog box.
- Choose the **OK** button to close the **Slide and Lifter Design** dialog box. Refer to Figure 16-116 for arrangement of lifter and repeat the same procedure for other lifters.

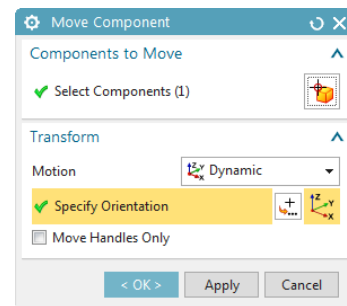


Figure 16-115 The *Move Component* dialog box

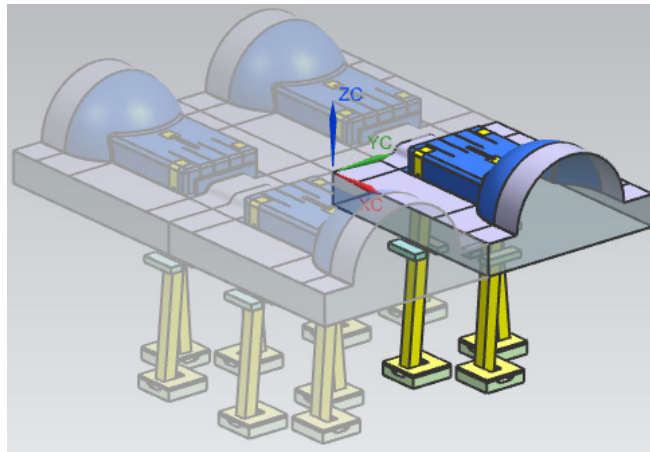


Figure 16-116 The arrangement of lifter



Note

If you create the lifter for one component then other lifters will be get placed automatically.

Adding Register Ring in the Mold Assembly

Now, you need to add the register ring in the mold assembly.

1. Choose **MW Standard Part Library** from the **Reuse Library**. Expand the **MW Standard Part Library** and then choose the type of locating ring from the **Injection** folder of the **DME_MM** Standard.
2. Double-click on the **Locating_Ring_With_Mounting_Holes[DHR21]** from the **Member Select** panel; the **Standard Part Management** dialog box is displayed with the **Information** window, refer to Figure 16-117.
3. Select **M8** and **12** from the **Type** and **H** drop-down lists respectively. Enter **100** in **D** edit box of the **Details** rollout and then choose the **OK** button; the **Information** window and the **Standard Part Management** dialog box is closed. Also, another **Information** window is displayed. Choose the **Close** button to close the window. Refer to Figure 16-118 for arrangement of register ring.

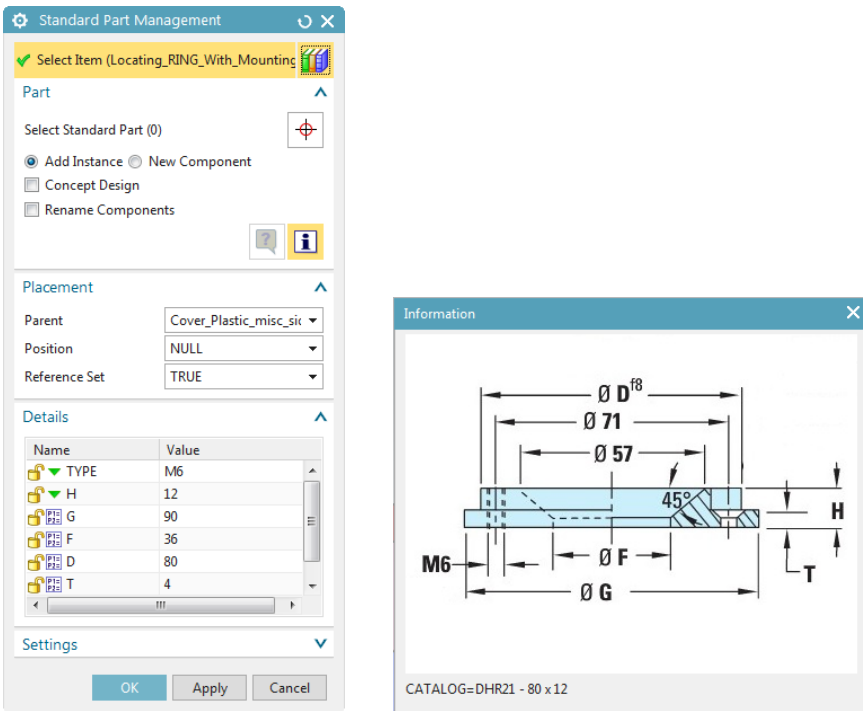


Figure 16-117 The *Standard Part Management* dialog box and the *Information* window

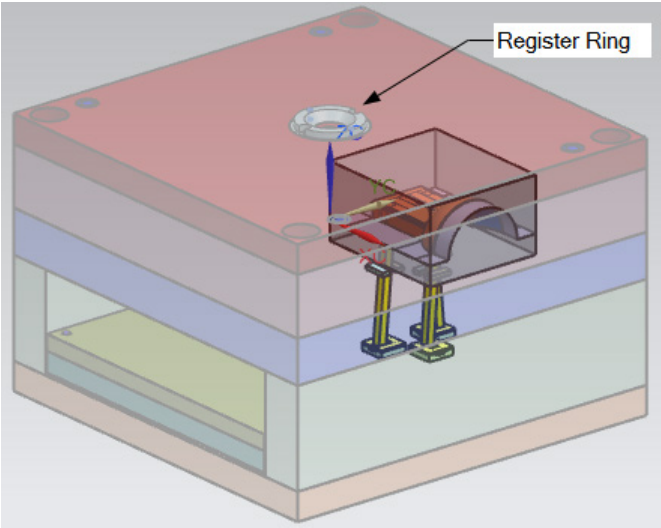


Figure 16-118 The arrangement of register ring in the mold assembly

Adding Sprue Bush in the Mold Assembly

Now, you will add the sprue bush in the mold assembly.

1. Choose the **MW Standard Part Library** from the **Reuse Library**. Expand the **MW Standard Part Library** and then choose the type of sprue bush from the **Injection** folder of **DME_MM Standard**.
2. Double-click on the **Sprue Bushing(DHR74)** from the **Member Select** panel; the **Standard Part Management** dialog box is displayed with the **Information** window, refer to Figure 16-119.
3. Select **8**, **4**, and **116** from the **D**, **O**, and **N** drop-down lists, respectively. Enter **24** in the **K** edit box of the **Details** rollout.

If the size of sprue bush is not appropriate then you can change it by double-clicking on the **Sprue Bushing (DHR74)** from the **Member Select** panel; the **Standard Part Management** dialog box is displayed. Click in the **Select Standard Part** area in the **Part** rollout and then select the sprue bush in the mold assembly. Now, you can change the size of parameters or reposition the sprue bush. Refer to Figure 16-120 for arrangement of sprue bush in the mold assembly.

4. Choose the **OK** button to close the dialog box.

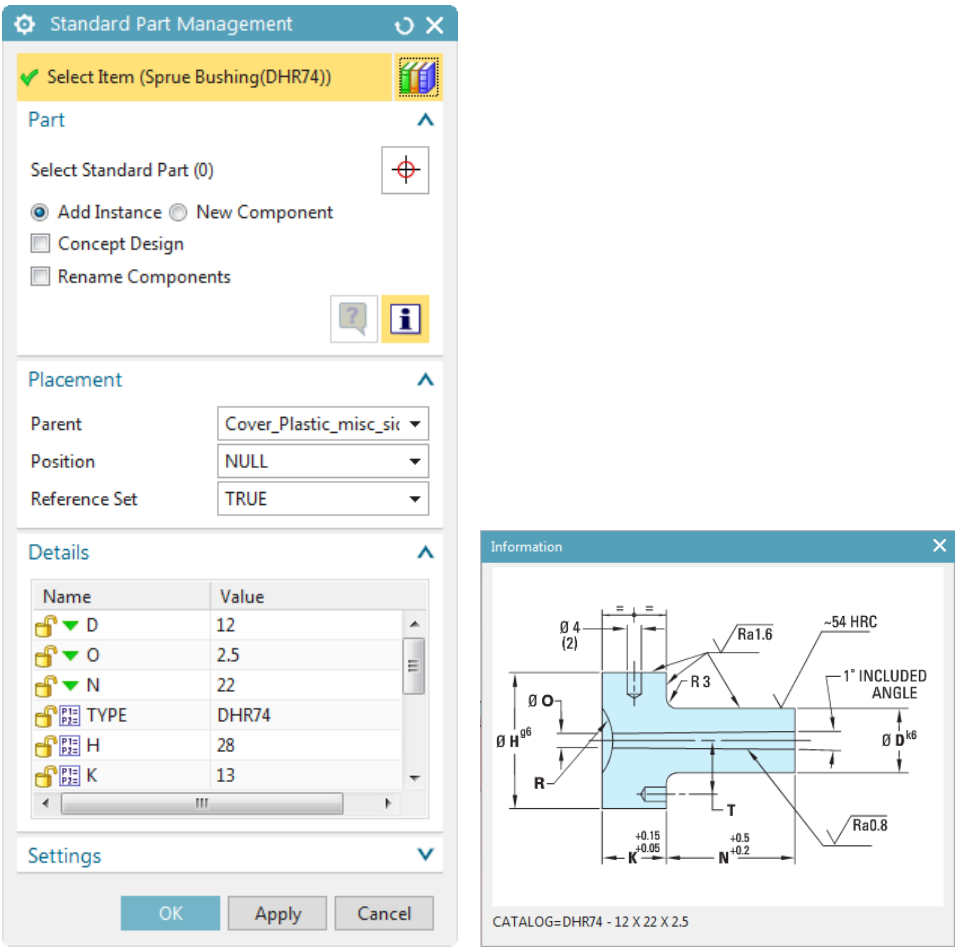


Figure 16-119 The Standard Part Management dialog box and the Information window

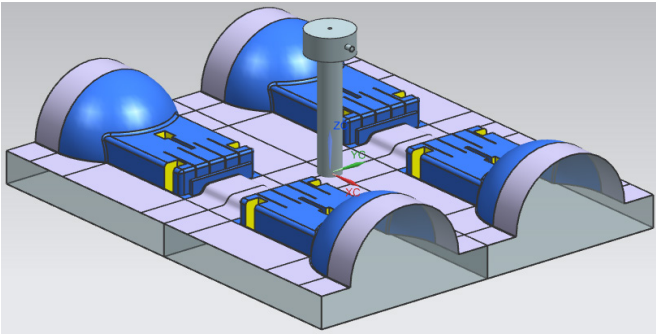


Figure 16-120 The arrangement of sprue bush in the mold assembly

Adding Ejector Pin in the Mold Assembly

Now, you will add the ejector pin in the mold assembly.

1. Choose **MW Standard Part Library** from the **Reuse Library**. Expand the **MW Standard Part Library** and then choose the desired type of ejector pin from the **Ejection** folder of the **DME_MM** Standard.
2. Double-click on the **Ejector Pin[Shouldered]** in the **Member Select** panel; the **Standard Part Management** dialog box is displayed with the **Information** window, refer to Figure 16-121.

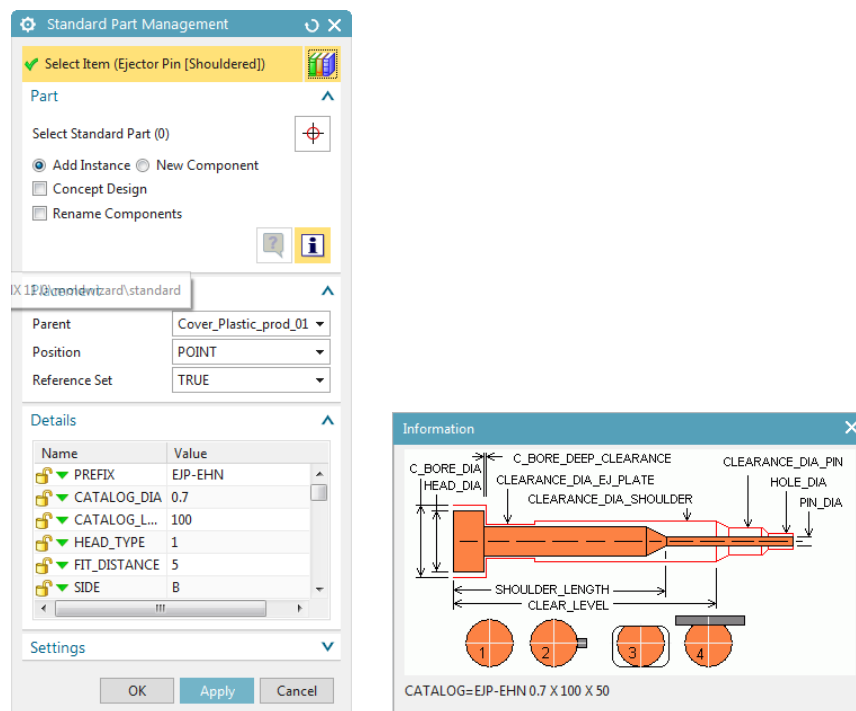


Figure 16-121 The **Standard Part Management** dialog box and the **Information** window

3. Select **EJP-EHN**, **1**, **160**, and **1** from the **PREFIX**, **CATALOG_DIA**, **CATALOG_LENGTH** and **HEAD_TYPE** drop-down lists respectively in the **Details** rollout. Choose the **Apply** button; the **Point** dialog box is displayed, refer to Figure 16-122. Specify the position of the ejector pin by placing the point. Choose the **Cancel** button to close the dialog box. Next, choose the **OK** button to close the **Standard Part Management** dialog box and the **Information** window. Refer to Figure 16-123 for arrangement of ejector pin.

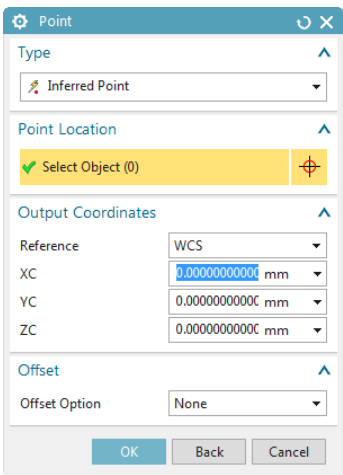


Figure 16-122 The *Point* dialog box

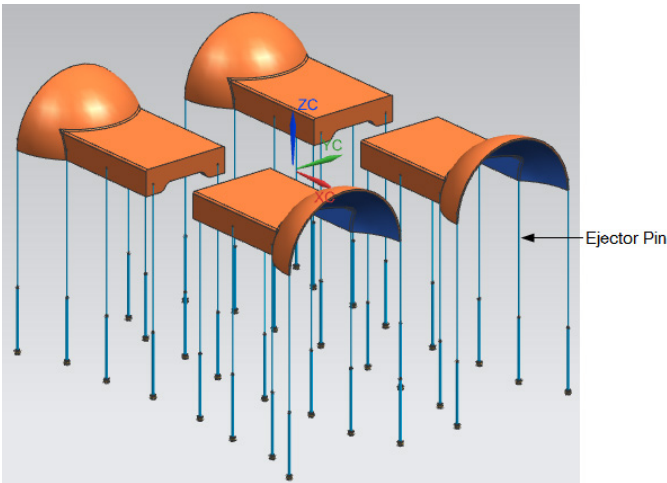


Figure 16-123 The arrangement of ejector pin in the mold assembly

Trimming the Ejector Pin

In this section, you will trim the ejector pin in the mold assembly.

1. Choose the **Ejector Pin Post Processing** tool from the **Main** gallery of the **Mold Wizard** tab; the **Ejector Pin Post Processing** dialog box is displayed, refer to Figure 16-124.
2. Select the ejector pins that you need to trim.
3. Choose the **Apply** button to trim the ejector pin. Choose the **Cancel** button to close the dialog box.

Creating the Pocket

Now, you will create the pocket for components to be placed or moved.

1. Choose the **Pocket** tool from the **Main** gallery of the **Mold Wizard** tab; the **Pocket** dialog box is displayed, refer to Figure 16-125. Also, you are prompted to select the target bodies. Select the target bodies from the mold assembly.
2. Click in **Select Object** area in the **Tool** rollout, choose the tool body for which you need to create the pocket. Next, choose the **Apply** button to create the pocket and then the **Cancel** button to close the dialog box. Refer to Figure 16-126 for pocket in the core plate to keep core insert.

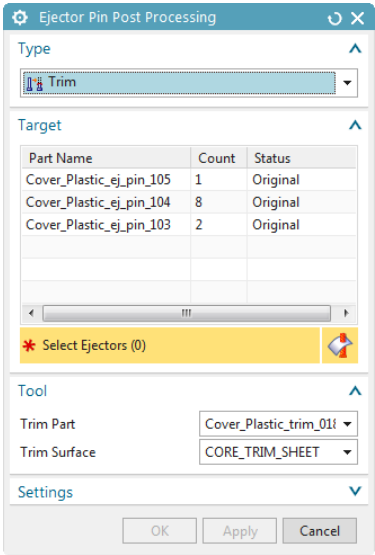


Figure 16-124 The *Ejector Pin Post Processing* dialog box

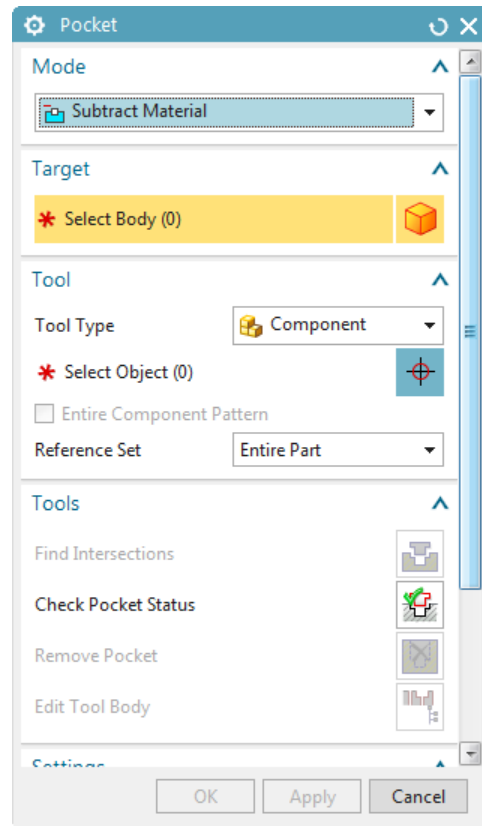


Figure 16-125 The Pocket dialog box

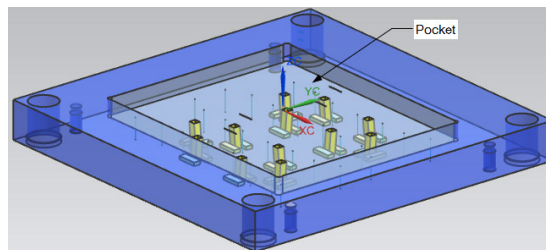

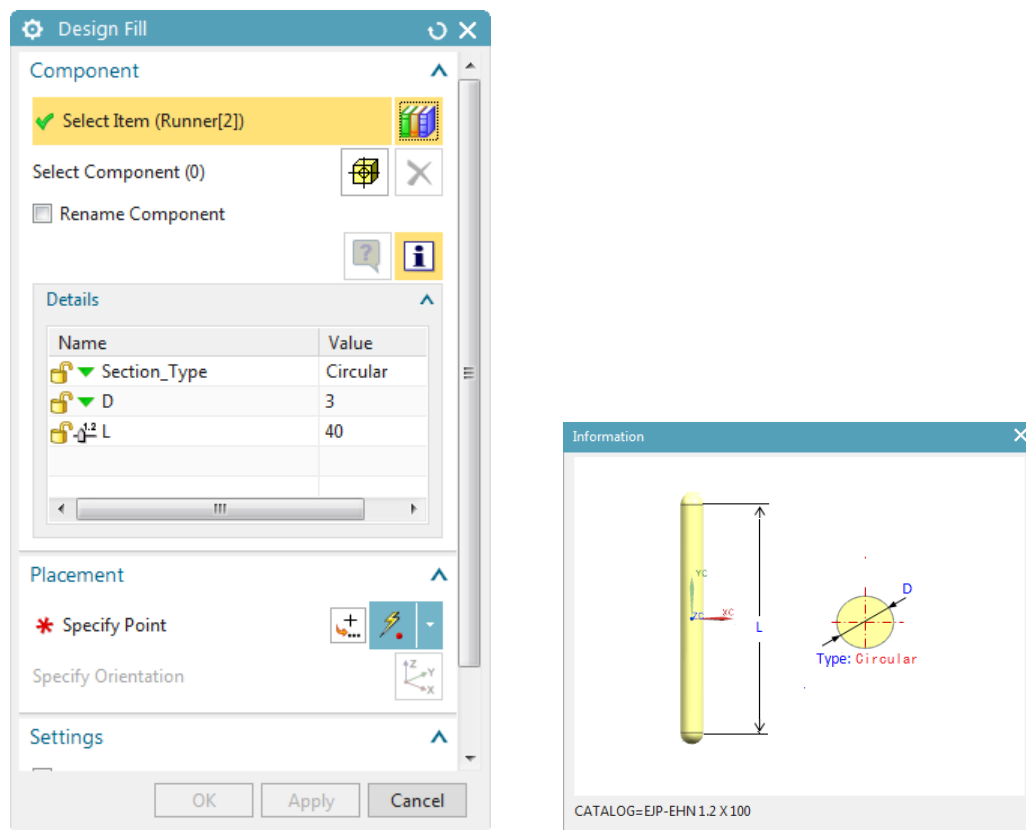


Figure 16-126 The pocket in the core plate

Creating the Gate

Now, you will create a gate for mold component.

1. Choose the **Design Fill** tool from the **Main** gallery of the **Mold Wizard** tab; the **Design Fill** dialog box is displayed with the **Information** window, refer to Figure 16-127. 
2. Select the **Fan** gate from the **Member Select** panel in the **Reuse Library**.



*Figure 16-127 The **Design Fill** dialog box and the **Information** window*

3. Specify the parameter of the gate in the **Details** rollout of the dialog box. Click in the **Specify Point** area in the **Placement** rollout and then specify the gate position and orientation, refer to Figure 16-128.

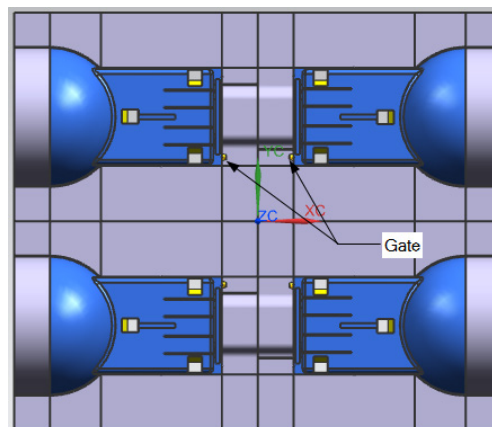



Figure 16-128 The gate position and orientation in the model

4. After placing the gate, choose the **OK** button to close the dialog box.

Creating the Runner

Now, you will create a runner for mold component.

1. Choose the **Design Fill** tool from the **Main** gallery of the **Mold Wizard** tab; the **Design Fill** dialog box is displayed. 
2. Select the **Runner[4]** from the **Member Select** panel in the **Reuse Library**.
3. Select **10** from the **D** drop-down list and enter **92** in the **L** edit box of the **Details** rollout. Click in the **Specify Point** area in the **Placement** rollout and position the runner; refer to Figure 16-129.
4. After placing the runner, choose the **OK** button to close the dialog box.

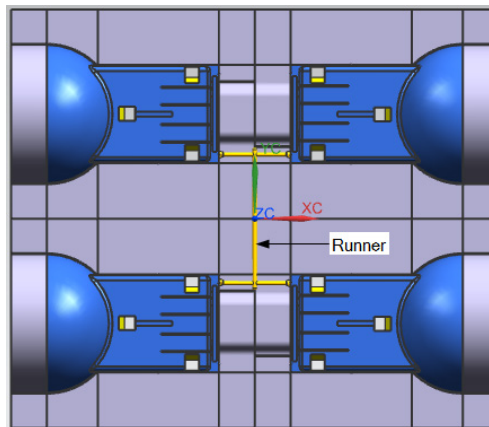



Figure 16-129 The runner position and orientation in the model

Adding the Cooling Channel

Now, you will add a cooling channel for mold component.

1. Choose the **Pattern Channel** tool from the **Cooling Tools** gallery of the **Mold Wizard** tab; the **Pattern Channel** dialog box is displayed, refer to Figure 16-130. 
2. Choose the **Sketch Section** button in the **Channel Path** rollout; the **Create Sketch** dialog box is displayed, refer to Figure 16-131.

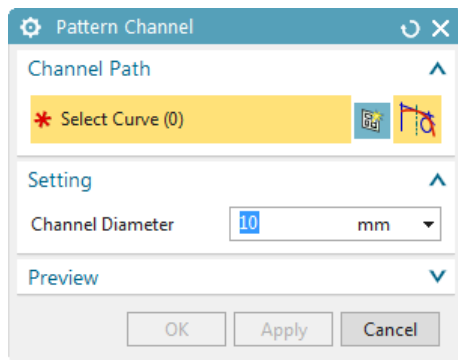


Figure 16-130 The Pattern Channel dialog box

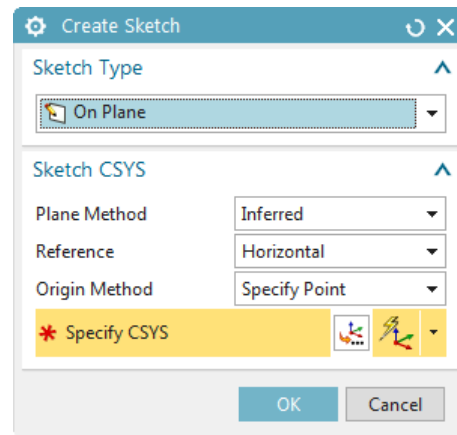


Figure 16-131 The Create Sketch dialog box

3. Create the sketch for cooling channel on a plane and then choose the **Finish** button to exit the sketch environment; the **Pattern Channel** dialog box is displayed. Specify the diameter of channel in the **Channel Diameter** edit box of the **Setting** rollout. Choose the **OK** button to close the dialog box. Refer to Figure 16-132 for understanding the position and orientation of the cooling channel.

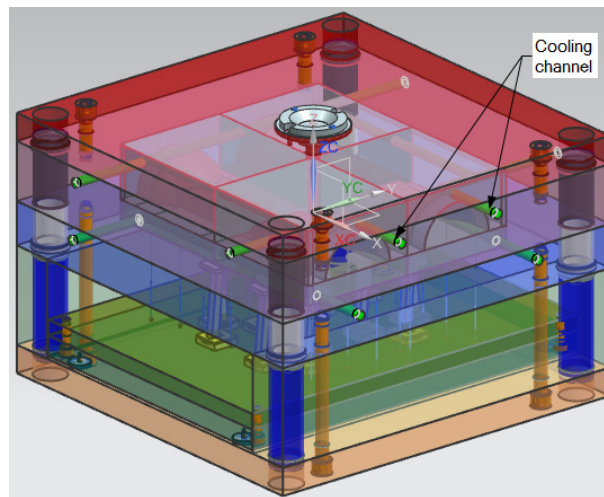


Figure 16-132 The orientation and position of cooling channel

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The _____ tool is used to define the path of the mold project.
2. The _____ tool is used to assign the core and cavity regions in the model.
3. There are _____ types of configuration in the **Initialize Project** dialog box.
4. You can change the orientation of the model by using the _____ tool.
5. The _____ tool is used to apply material on model.
6. Gate is a channel which connects sprue bush to the component. (T/F)
7. Runner is a channel which connects sprue bush to the gate. (T/F)
8. Shrinkage is used to apply scale factor to the component. (T/F)
9. The **Patch Surface** tool is used to create the sheet to close the openings in the model. (T/F)
10. The **Design Fill** tool is used to create gates. (T/F)

Review Questions

Answer the following questions:

1. Which of the following dialog boxes is displayed when you choose the **Patch Surface** tool from the **Parting Tools** gallery of the **Mold Wizard** tab?

(a) Patch Surface	(b) Edge Patch
(c) Patch	(d) None
2. Which of the following tools in NX Mold Wizard is used to create core and cavity regions?

(a) Check Regions	(b) Define Regions
(c) Patch Surface	(d) None
3. Which of the following dialog boxes is displayed when you choose the **Set Cavity** button from the **Easy Fill Advanced** tab to specify the cavity of the component?

(a) Set Cavity	(b) Select Cavity
(c) Cavity	(d) Set
4. The _____ tool is used to create a parting surface of the model.

5. The _____ tool is used to add cooling fitting components to cooling channels.
6. The _____ tool is used to adjust the position of the cooling channel.
7. The _____ tool is used to add a mold base.
8. The **Design Gate** tool is used to create the runner. (T/F)
9. The **Assembly Drawing** tool helps you to create the drawing of the mold assembly. (T/F)
10. The **Slider and Lifter Library** tool helps you to create slider in mold design. (T/F)

EXERCISE

To perform the exercise, you need to download the zipped file named as *c16_NX_11.0_input* from the Input Files section of the CADCIM website. The complete path for downloading the file is:

Textbooks > CAD/CAM > NX > NX 11.0 for Designers > Input Files

After the file is downloaded, extract the folder to the location *C:\NX 11.0* and rename it as *c16*.

Exercise 1

Create the Mold Design of the model shown in Figure 16-133.



Figure 16-133 Trimetric view of the Phone Case cover

Answers to Self-Evaluation Test

1. Initialize Project, 2. Check Regions, 3. Three, 4. Mold CSYS, 5. Initialize Project, 6. F, 7. T, 8. T, 9. T, 10. T