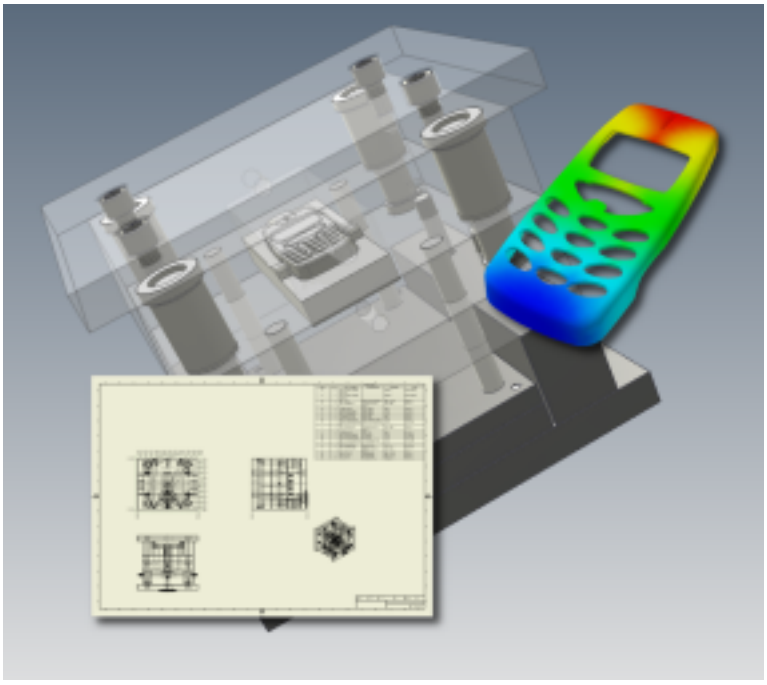
Contents

General Mold Workflow

1

About this tutorial



Explore the general mold design workflow.

Skill level **Level 3**- Specific Interest

Time required	60 Minutes
Prerequisites	■ Know how to set the active project and navigate the model space with the various view commands. See the Inventor Help topic *Getting Started* for further information.
Tutorial files used	Mobile Top.ipt, Mold.ipj

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Navigation

Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Tutorial objectives

In this tutorial, you open an Inventor part file in the mold application and use it to step through the process of mold base design. After completing this tutorial, you will be able to:

- Generate the core and cavity.
- Create the feeding system, including runners, gates, sprues, sprue bushings, and locating rings.
- Select a mold base.
- Create ejector pins.
- Create the cooling system, including cooling channels and cooling components.
- Generate a drawing from your mold design.

Next (page 3)

Getting Started

In this section, you create a mold design assembly and add the plastic part to mold.

1. Click **Inventor**  and select **Manage** > **Projects**.
2. Browse to the Tutorial Files/Mold folder and set the project to **Mold.ipj**.
3. In the **Projects** dialog box, click **Configure Content Center Libraries**  .
4. In the **Configure Libraries** dialog box, scroll to the Inventor Mold libraries and ensure that the **In Use** checkbox is enabled. If the metric library is not enabled, you cannot able to add the mold base later in the tutorial.
5. Click **OK** on the **Configure Libraries** dialog box.
6. Click **Done** on the **Projects** dialog box.
7. Click **Inventor**  .
8. Select **New** to open the **New File** dialog box and start a new mold assembly.
9. From the **Metric** tab of the **New File** dialog box, click **Mold Design (mm).iam**, and then click **OK** on the **New File** dialog box.
10. In the **Create Mold Design** dialog box, click **OK** to accept the defaults. The Mold Design environment is now available.

TIP Do not check the box titled Interactive Tutorials. It provides an alternate method to navigate to the tutorials.

NOTE For subsequent work, it is important to choose a descriptive name that easily identifies the mold file. For this tutorial, you use the supplied default. The About this tutorial (page 154) tutorial will further explain project files and file schemes.

11 In the Mold Design browser, right-click the mold icon and choose **Prompt for File Names** in the context menu. You can control the names and locations of system generated files.

   NOTE The enabled or disabled state of the file name setting is persistent between sessions.

12 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Plastic Part**  .

13 In the **Plastic Part** dialog box, select the **Mobile Top.ipt** part file and click **Open** to add the plastic part to the mold assembly.

14 Left-click in the graphics window to place the part with the default alignment option.

15 Click OK to accept the default file names and location on the file naming dialog box.

The Inventor part file, which is now placed in the Mold Design environment, will be used to guide you through the process of mold base design.

User Inventor Mold Design to place multiple moldable parts in the mold assembly as a family of parts. Or, you can use patterning functionality in the mold layout to create rectangular, circular, or variable placement patterns.

For more information about these techniques, refer to the following tutorial: About this tutorial (page 47)

Previous (page 1) | Next (page 4)

Defining the Mold Parameters

In this section, you define the mold parameters for orientation and material type.

Initially, these options are the only ones available when creating a mold assembly. When the moldable part is placed, an associated Orientation node is automatically generated. This node cannot be deleted, but the Adjust Orientation command can be used to modify it.

With the Adjust Orientation command, you can rotate the part around an axis, or align an edge or surface of the part with an axis. In some mold designs, the default direction of the mold opening, which aligns with the positive Z axis, is not correct. The direction of the plastic part and the direction of the

mold opening are often different. The Adjust Orientation command adjusts these directions so they are consistent.

1. On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Adjust Orientation**  . The **Adjust Orientation** command performs an analysis of the model surfaces. All green surfaces have a draft angle between the surface and the Z axis, which is greater than zero degrees. All blue surfaces have a draft angle between the surface and the Z axis, which is less than zero degrees.
2. Rotate the model to see that no further orientation adjustments are required in this example. On the **Adjust Orientation** dialog box, click **Cancel** to continue without changing anything.
3. On the **Mold Layout** panel, click **Select Material**  . This command enables you to select from a database of more than 7,750 materials.
4. From the list of Manufacturers, select **SABIC Innovative Plastics US, LLC**.
5. From the list of Trade names, select **Cycoloy C2950 HF-71676**.
6. In the Selected material section of the dialog box, click **Details**. Many different material characteristics are listed in the tabs for the selected material, including material shrinkage values.
7. Click **OK** to close the **Thermoplastics Material** dialog box.
8. Click **OK** to accept the selected material and close the **Select Material** dialog box. Notice that the model automatically adjusts its size to take into account the material that is being used. Now that the mold parameters have been defined, additional mold options are available.

Previous (page 3) | Next (page 5)

Defining the Workpiece

In this section, you define the workpiece to use to create the core and cavity for the mold.

1. On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Core/Cavity**  .

2 From the **Core/Cavity** tab, click **Parting Design panel** > **Define Workpiece Setting** .

3 Maintain **Rectangular** as the Workpiece Type. In the **Z_total** field, enter **50 mm**.

NOTE Increasing the total Z dimension adjusts the dimensions of the workpiece to facilitate machining and installation.

4 Click **OK** on the **Define Workpiece Setting** dialog box to accept the parameters and close the dialog box.

5 Accept the default file name and location on the file naming dialog box.

6 The workpiece appears around the model, as shown in the following image.

Previous (page 4) | Next (page 6)

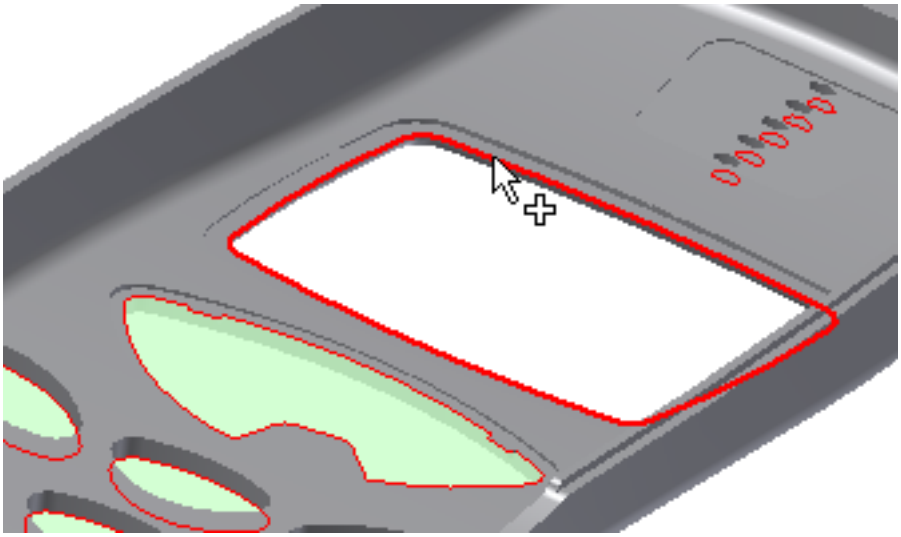
Defining the Parting Surface

In this section, you define the parting surface for the mold assembly.

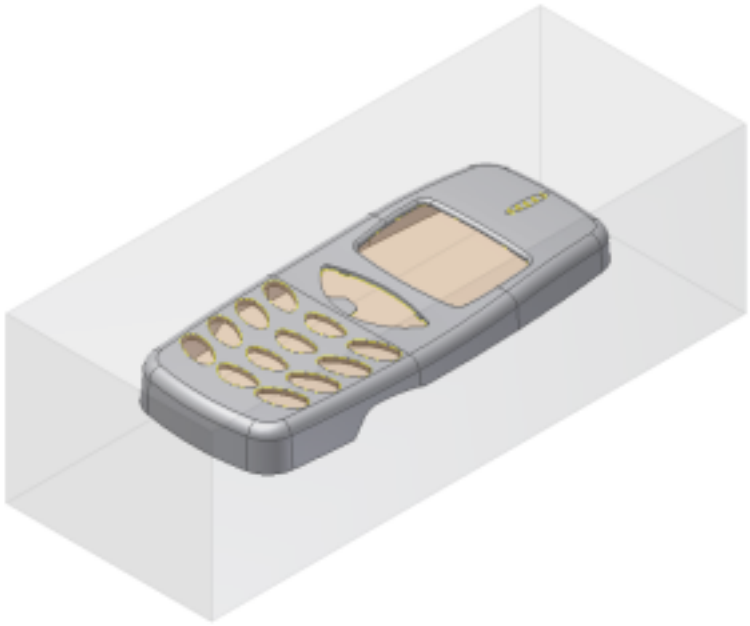
Creating a satisfactory parting surface is a crucial step in designing a successful injection mold. The right parting surface helps ensure that the design of the core and cavity suitable. The plastic part can be taken out of the mold. In Inventor Mold Design, the parting surface is divided into two parts: the patching surface and the runoff surface.

Patching surfaces are used to fill in all internal openings in a part, such as holes and slots. Many patching surfaces can exist in one part. The runoff surface is used to split the workpiece, and is also where the part is taken out of the mold. Only one runoff surface can exist on a part.

1. On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Create Patching Surface**  . Select the Auto Detect button to detect all internal patches automatically and list them by name in the Creating Patching Surface dialog box. The patches are highlighted on the model in green.
2. Scroll through the list of patches and notice that 19 patches have been created. In some situations, the automatically detected patches create surfaces that are not suitable. Some patch surfaces may also be missing in the model. In this case, the largest opening is not patched correctly.
3. Scroll to the top of the list of patches. Individually select Patch 2. Notice that it did not successfully patch the largest opening in the mobile phone cover. Patch 2 is to be deleted. The opening will be patched manually.
4. Select Patch 2 in the **Create Patching Surface** dialog box and press **Delete** on the keyboard. A dialog box confirming the deletion appears. Ensure that patch 2 is selected and click **Yes**.
5. Scroll to the bottom of the list and select **Click to add**. Patch 20 is added to the list.
6. Rotate the model and select the inside loop to patch the large opening as shown in the following image.



7 Click **OK** to generate all patching surfaces. Click the Home view. The model appears as shown in the following image.



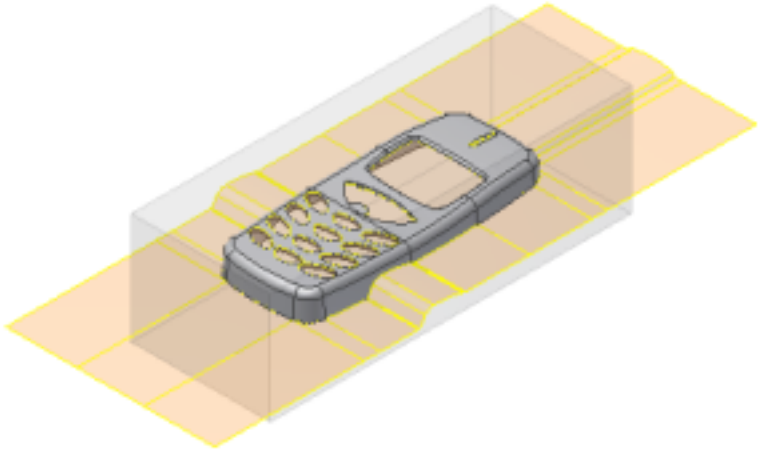
In real-world mold design sessions, you use a combination of automatic and manual surfacing to patch openings. In addition to using **Click to add**, you can also create manual patches with the Use Existing Surface or Create Planar Patch commands. Click the following link to open an animation that shows creating patching surfaces manually using these two workflows.

Manually generating patching surfaces

Click the play button in the following image to play an animation that shows creating patching surfaces manually.

1. On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Create Runoff Surface** . Click Auto Detect to generate a preview of the runoff loop as shown in the following image.

1. Click **OK** on the **Create Runoff Surface** dialog box to accept the default surfaces.



Previous (page 5) | Next (page 10)

Generating the Core and Cavity

In this section, you generate the core and cavity.

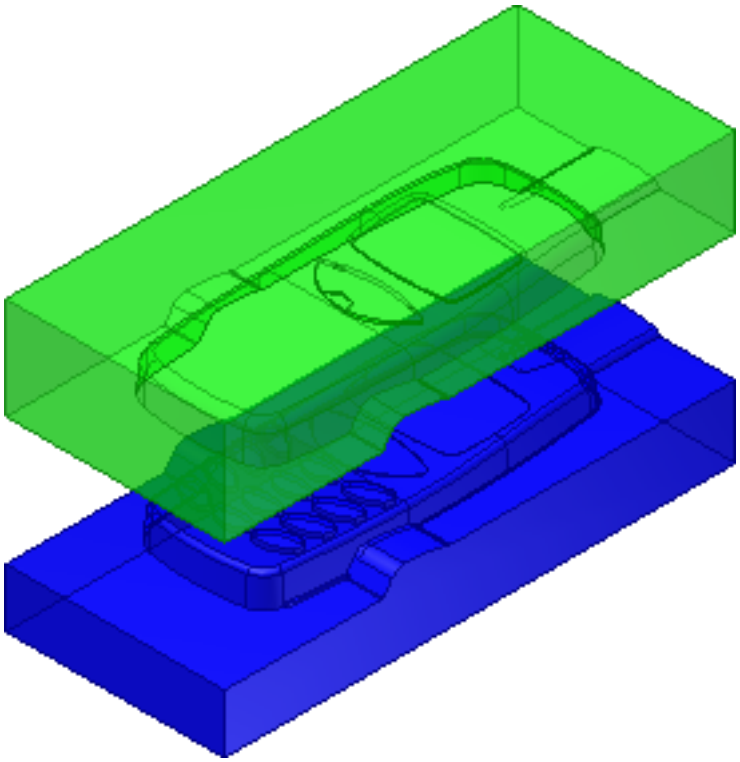
After the workpiece, patching surfaces, and runoff surface have been created, the Generate Core/Cavity command is used to split the workpiece into the core and cavity. Mold Design also provides a method of importing an existing core and cavity that was created by another design team or by using another CAD system.

1. On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Generate Core and Cavity** .
2. In the Generate Core and Cavity dialog box, select **Preview/Diagnose.** The bodies appear in the defined color.

   NOTE Notice the Opacity Settings controls for the Core Body and Cavity Body that allow you to change the transparency of each body. Notice the Repair Tolerance setting which can repair small gaps. We will not change either of these areas.

3. Select the Body Separation slider and move it to the right until it displays a value of 75.

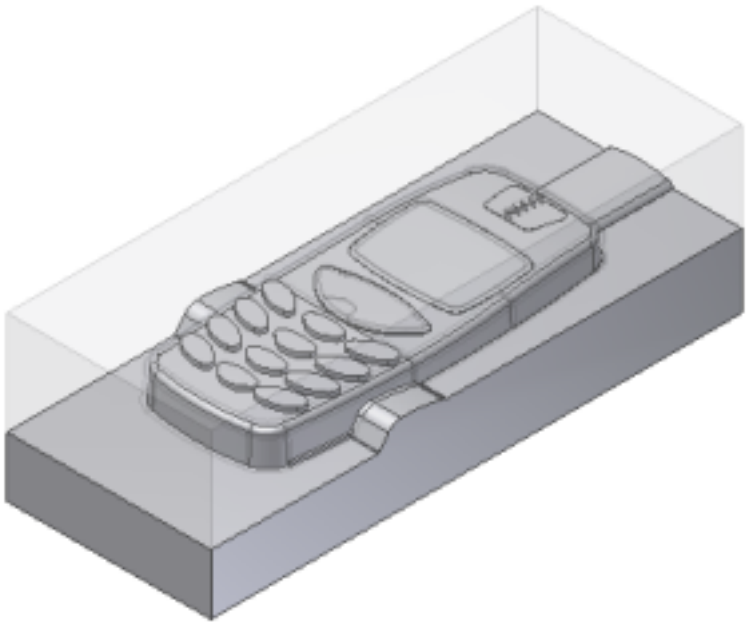
The core and cavity preview updates as you move the slider. The following image displays the preview of the separated bodies.



4. Click the Parting Diagnostics tab. In an actual design, the problems detected should be fixed. None of the problems listed cause the core and cavity generation to fail.
5. Click OK to generate the core and cavity.
6. Click OK to accept the default file names and location on the file naming dialog box. Notice that core files are appended with an _CR and the cavity files are appended with an _CV in the default naming scheme.
7. Select Finish Core/Cavity to return to the parent assembly.

The core and cavity are automatically generated, as shown in the following image.The system removes the driving moldable part and the workpiece from the display, and the core/cavity is now shown with the cavity displayed in a translucent style.

The browser displays the Core Cavity node. The About this tutorial (page 154) tutorial will further explain the mold assembly file structure.



We will not edit the core cavity in this exercise. To edit the core and cavity, right-click the Core Cavity node in the browser and select Edit Feature.

To delete the core and cavity, right-click the **Core Cavity** node in the Mold Design browser and then click **Delete**. In this tutorial, the core and cavity remains in the model and you continue working with it.

Previous (page 6) | Next (page 12)

Specifying the Gate Location

In this section, you manually create the gate location.

Gate locations are coordinates on the surface of the moldable part, which indicate where the physical gates will be placed. You can manually locate the gate position, or use the Gate Location analysis to recommend the best locations for the appropriate number of gates automatically.

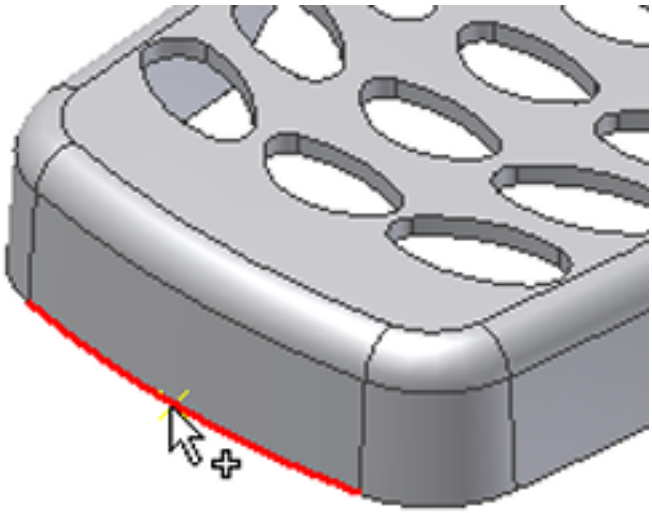
TIP To include a gate location in a pattern, check Copy to all pockets. Clear the check box to create a unique gate location.

NOTE You can access the Gate Location command in both the Mold Layout tab, and the Core Cavity tab.

1 On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Gate Location** . The core and cavity are removed from the display, and that the moldable part becomes visible to enable the specification of a gate location point. The **Gate Location** dialog box is also displayed.

2 Zoom in on the end of the model. Select a point near the mid-point of the lower edge, as shown in the following image. Because you selected an edge and not a vertex, a value indicating the edge length ratio appears in the dialog box.

NOTE If you select a vertex, the exact coordinates of the point appear. If you select on a moldable part face, U and V parameter values appear and can be modified.

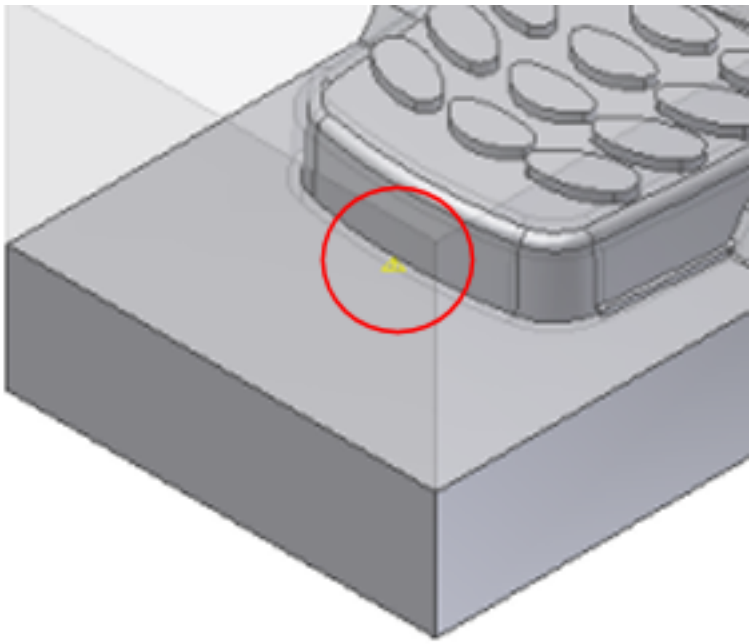


3 In the **Value** field of the **Gate Location** dialog, enter the value **0.5** and then click **Apply**. Additional gate point locations can be defined if a part requires multiple gates. This tutorial uses a single gate.

   The **Gate Location** dialog box provides a Suggest tab. An analysis runs on the moldable part and suggests locations for a user-specified number of gates. The About this tutorial (page 79) tutorial explains how to run a Gate Location analysis.

4 Click **OK** in the **File Naming** dialog box to accept the defaults and create the gate location.

5 Click **Done** on the **Gate Location** dialog box to finish the gate location process. Close the dialog box, and return to the display of the core and cavity. The point appears in the mold assembly, as shown in the following image.



Previous (page 10) | Next (page 14)

Sketching the Runner System

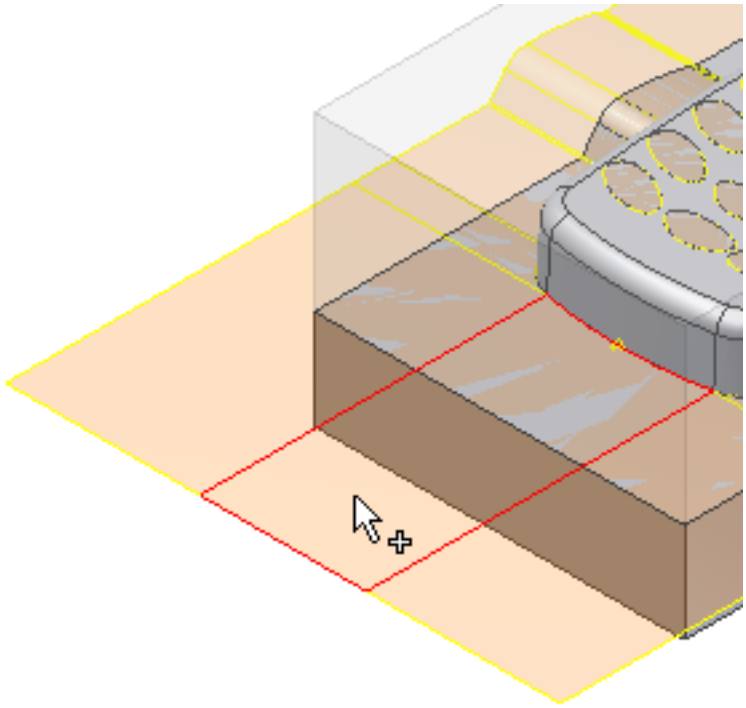
In this section, you manually create a runner sketch.

The runner sketch must be created before the features of the feeding system can be added. The feeding system, which consists of runners, gates, sprues, and cold wells, evenly distributes plastic melt from the machine nozzle to all the mold cavities.

The runner sketch can be sketched manually or added automatically. The Auto Runner Sketch command enables you to base the runner sketch on layouts that are commonly used in mold design. The About this tutorial (page 47) tutorial further explains the Auto Runner Sketch command.

The Manual Sketch command enables you to enter the sketch environment and draw a runner sketch.

1. On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Manual Sketch**  . The Manual Sketch command is located in the Auto Runner Sketch menu.
2. Select the planar runoff surface shown in the following image as the placement plane for the sketch.

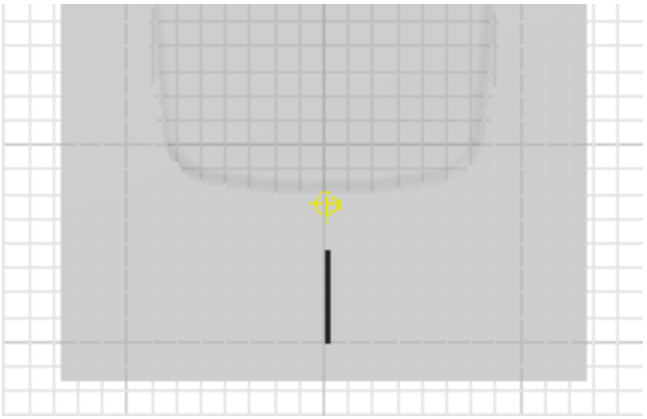


3 Make sure the **Type** is set to **Runner Sketch**, then click **OK** on the **Manual Sketch** dialog box to activate the sketch environment.

4 On the keyboard, press **F6** to orient the model in the Home View. In the upper-right corner of the graphics window, click **Top** on the ViewCube to orient the sketch.

5 Zoom in on the end of the model where you created a gate location in the previous section.

6 On the ribbon, click **Sketch tab** > **Draw panel** > **Line** . To represent the runner, sketch the line shown in the following image. The length of the line is not important, but the sketch of the line should not extend into the mold. Later in this tutorial, you create a gate to connect the runner to the mold.

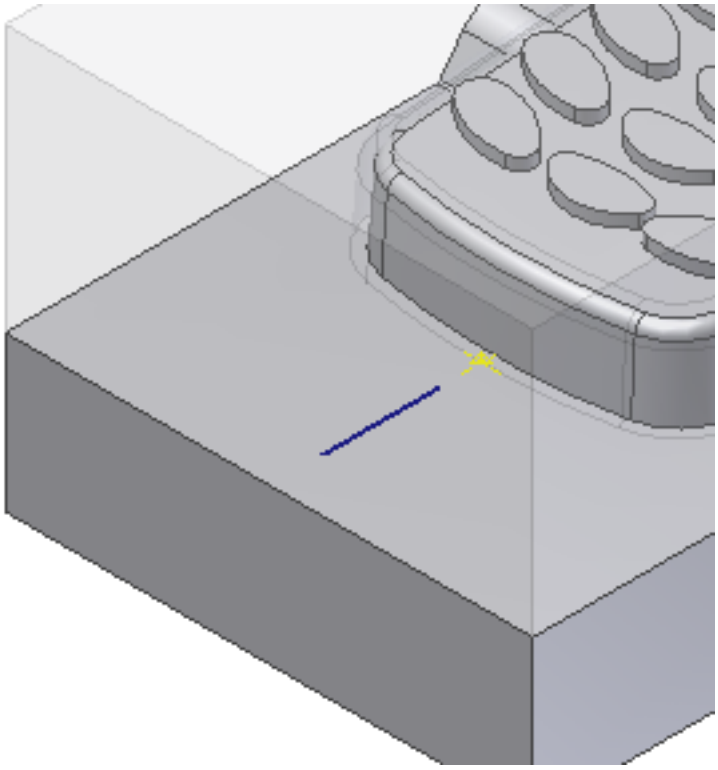


7 On the ribbon, click **Sketch tab** > **Draw panel** > **Project Geometry**  . Select the previously created gate location point to project that gate point into the current sketch.

8 Click **Sketch tab** > **Constrain panel** > **Coincident Constraint**  . Choose the projected point and the sketched line to ensure that they are aligned.

9 Click **Sketch tab** > **Exit panel** > **Finish Sketch**  to complete the sketch and return to the Mold Design environment.

10 On the keyboard, press **F6** to return to the Home View. The runner sketch and the gate point appear as shown in the following image.



Previous (page 12) | Next (page 18)

Creating a Runner

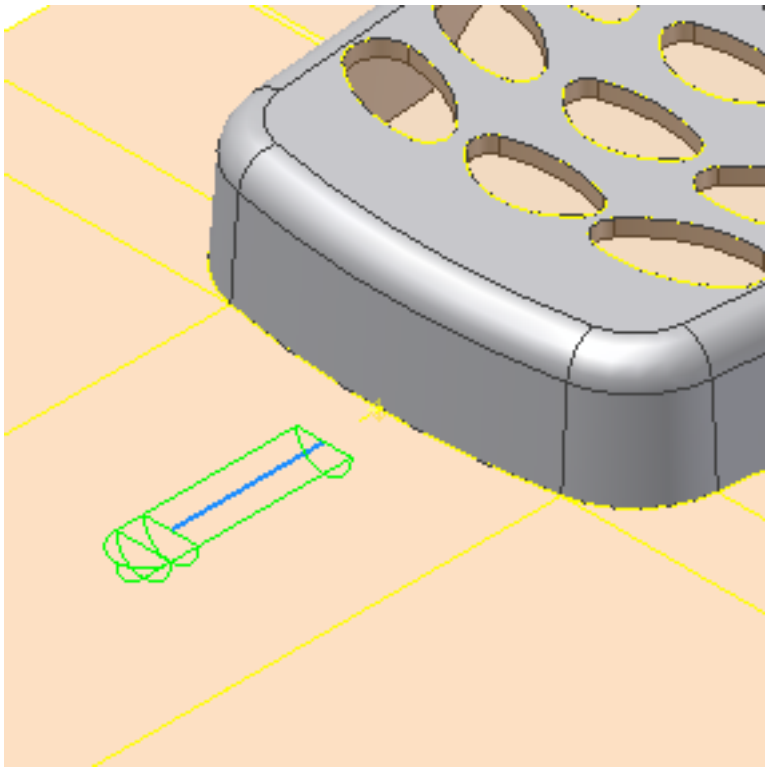
In this section, you use the runner sketch that you created in the previous section to create a runner.

The first step in creating a feeding system for the plastic melt is to create the runner system. Runners are channels for the plastic melt to flow from the sprue to the feed gates. The runner system consists of a main or primary runner, and possibly secondary or tertiary branch runners. A sprue feeds plastic melt to the runner system from the injection press.

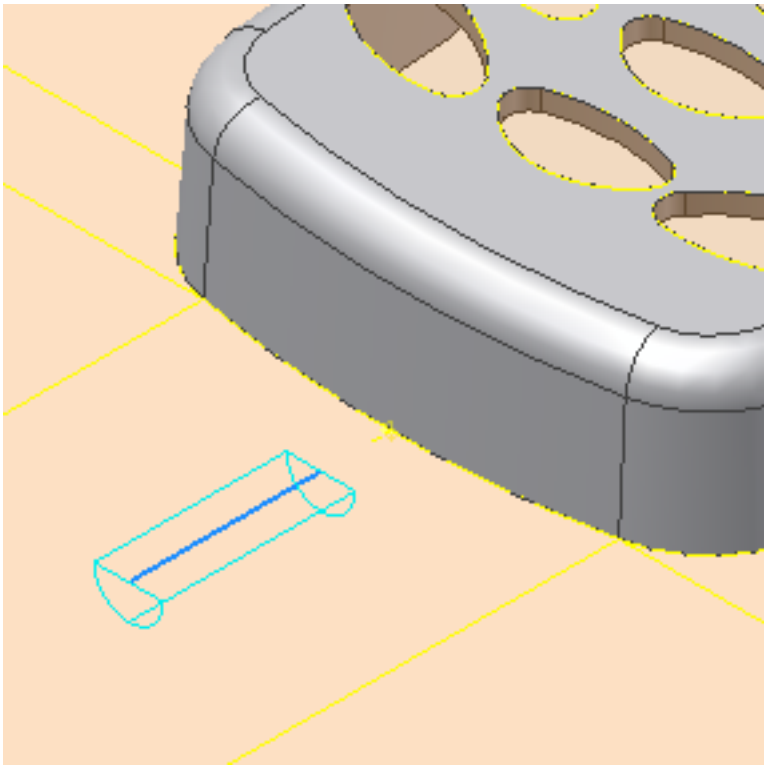
A secondary sprue, or sprue runner, can also be added if you are working in a three-plate mold. This tutorial uses a two-plate mold.

1. On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Runner** .
2. From the Section Type list, select **Semicircle**.
3. For the size of the semicircle, enter **3 mm**.
4. Select the sketched line of the runner that you created in the previous step as the path for the runner. The runner is previewed in the model, as shown in the following image.

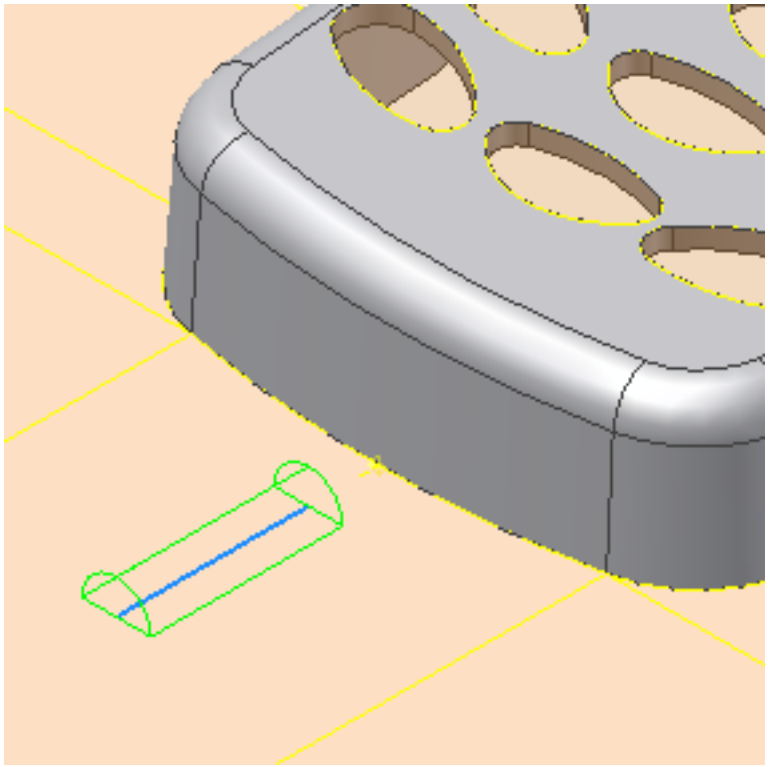
   NOTE The runner is previewed with a cold slug. The default Cold Slug Position is based on the position that you selected on the runner sketch when you specified the path.



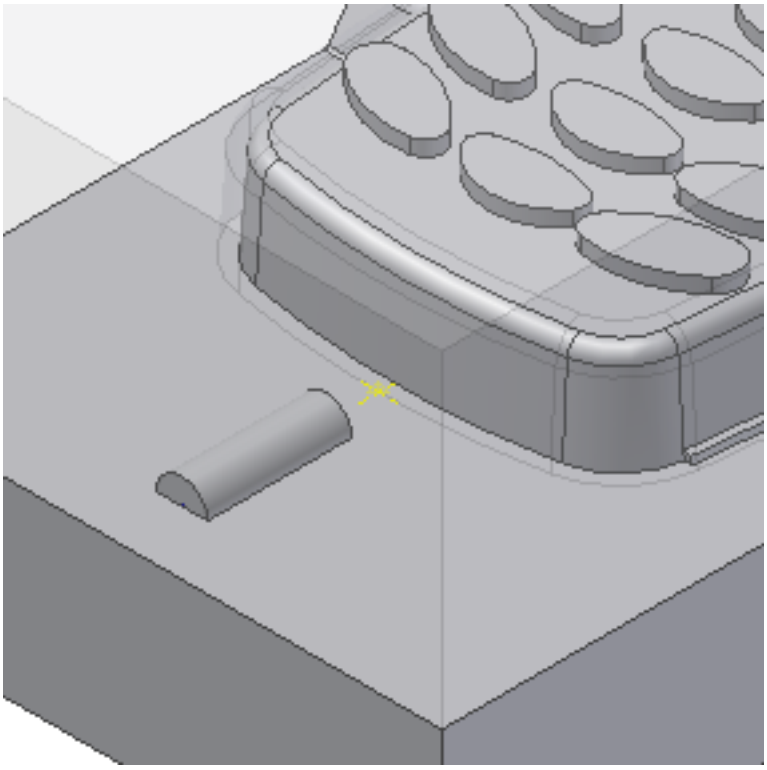
5 Click in the **Cold Slug Position** cell of the Runner Segment table, and select **None** from the list. The runner appears as shown in the following image.



6 When you create a semicircular runner, the runner can be previewed as part of the core or part of the cavity. To switch between both previews, select the **Core Side** and **Cavity Side** options in the Section Type area of the dialog. Switch the runner to **Cavity Side**. The preview updates as shown in the following image.



7 Click **OK** to close the **Create Runner** dialog box.



Previous (page 14) | Next (page 22)

Creating a Gate

In this section, you create a gate location.

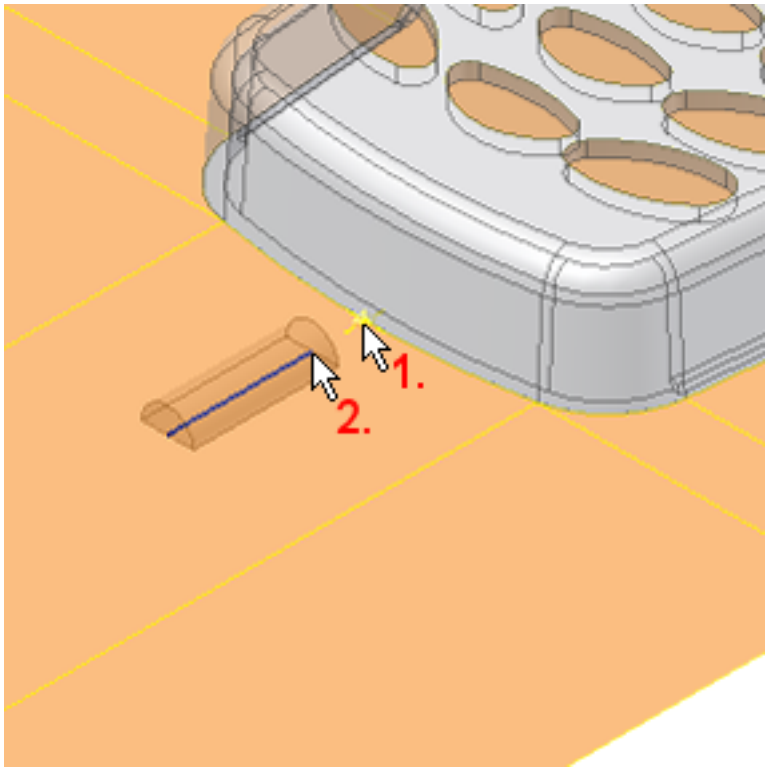
1. On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Gate**  .
2. Maintain the default type of gate, which is **Edge**.
3. Change the Placement option from One Point to **Two Points**.
4. Change the Placement option for the gate so that it is on the **Cavity Side**.

5 Change the values for the gate by entering new values for the following parameters:

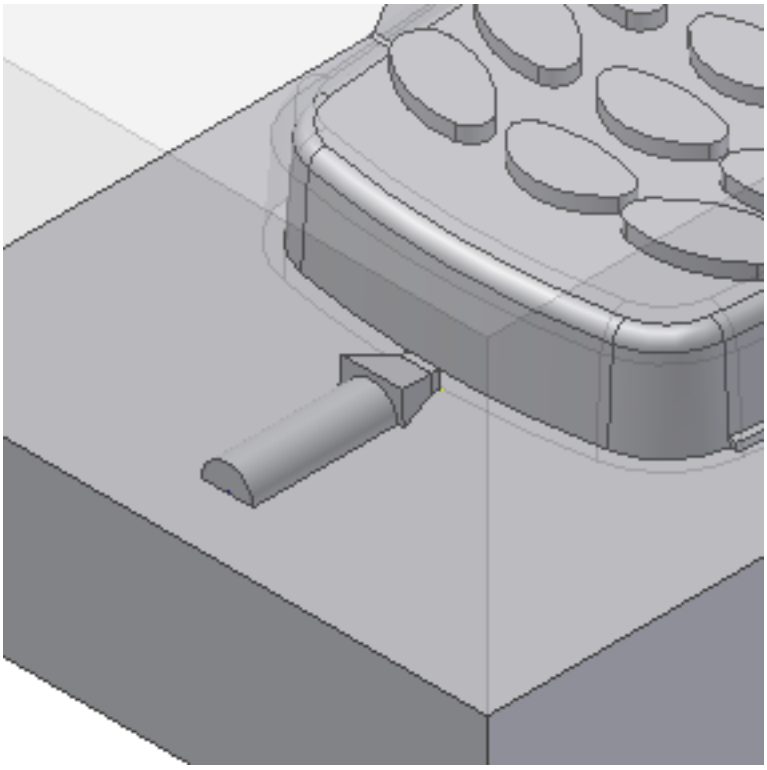
- W1 = **7 mm**
- H1 = **3.5 mm**
- W2 = **3.5 mm**
- H2 = **2 mm**

NOTE The available parameters vary, depending on the type of gate that is being created.

6 Activate the **Gate Location** selection tool in the Create Gate dialog box, and select the gate location point. The End point selection tool automatically becomes active. Select the end of the sketched line as the end point, as shown in the following image.



7 Click **OK** to close the **Create Gate** dialog box. The gate appears as shown in the following image.



Previous (page 18) | Next (page 24)

Creating the Mold Base

In this section, you create a mold base by using one of the standard mold bases supplied.

The mold base is an assembly of steel plates that contains the cores and cavities of a mold. You can place a standard mold base by selecting from a library of vendor mold bases, such as DME, HASCO, Futaba, and LKM. Or, you can customize any of the standard mold base types.

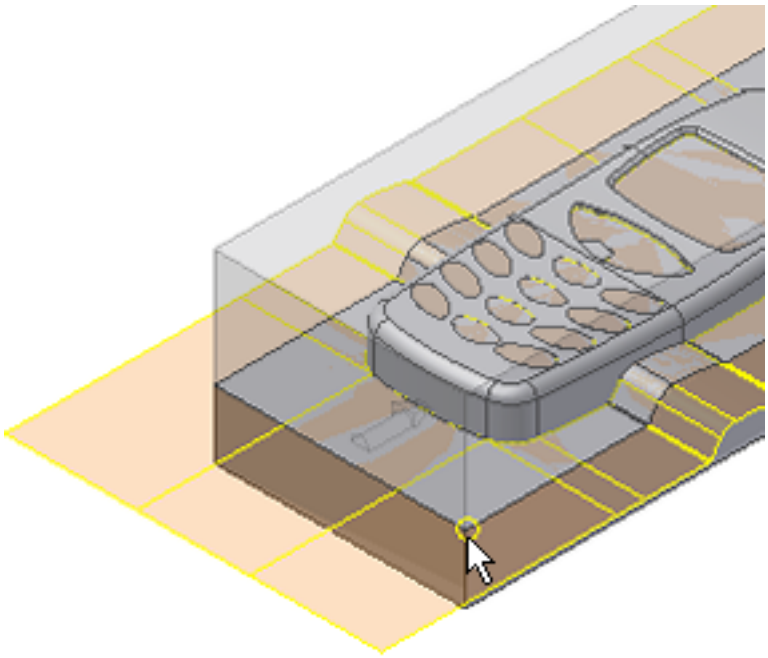
1 On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Mold Base** .

2 In the **Mold Base** dialog box, select the **FUTABA SA-S** Vendor and Type.

3 Select the **250 mm x 250 mm** mold base size.

NOTE The default size provides a recommendation based on the dimensions displayed under Layout Information.

4 Select the **Placement Ref Point** in the graphics window, as shown in the following image.



5 Expand the **Mold Base** dialog box by clicking the arrows at the top or bottom of the right-hand side of the dialog box.

6 In the Components list, click **SA-S-AP 250X250X60**.

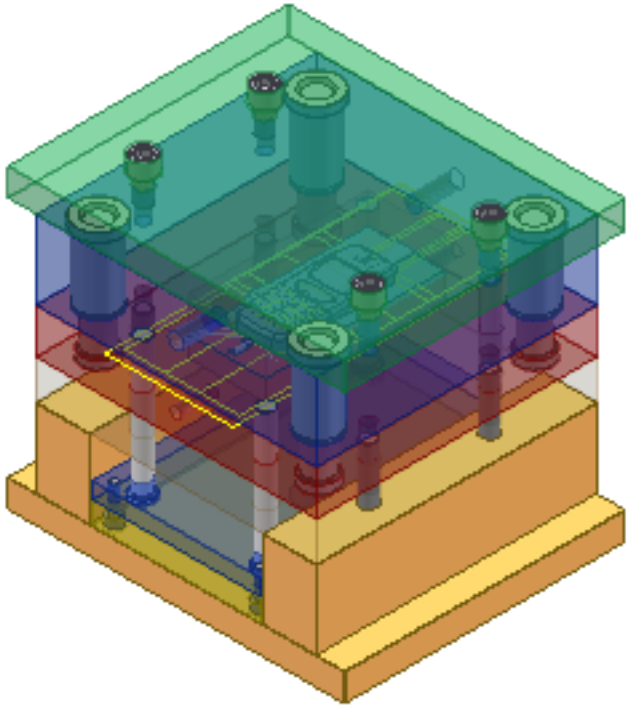
7 Click **Property Settings**  on the right of the selected component in the Component list .

8 From the Parameters list of the **SA-S-AP** dialog box, modify the **H_** value to **70 mm**.

9 Click **OK** on the **SA-S-AP** dialog box.

10 Click **OK** on the **Mold Base** dialog box.

11 To generate the mold base and close the dialog box, accept the defaults in the **File Naming** dialog box and click **OK**. The full mold base assembly appears as shown in the following image.



12 On the Quick Access toolbar, click **Save** . The mold assembly is comprised of many subassemblies and part files. The About this tutorial (page 154) tutorial explains the structure of the mold assembly.

13 Click **OK** on the **Save** dialog box to save the assembly and its dependents.

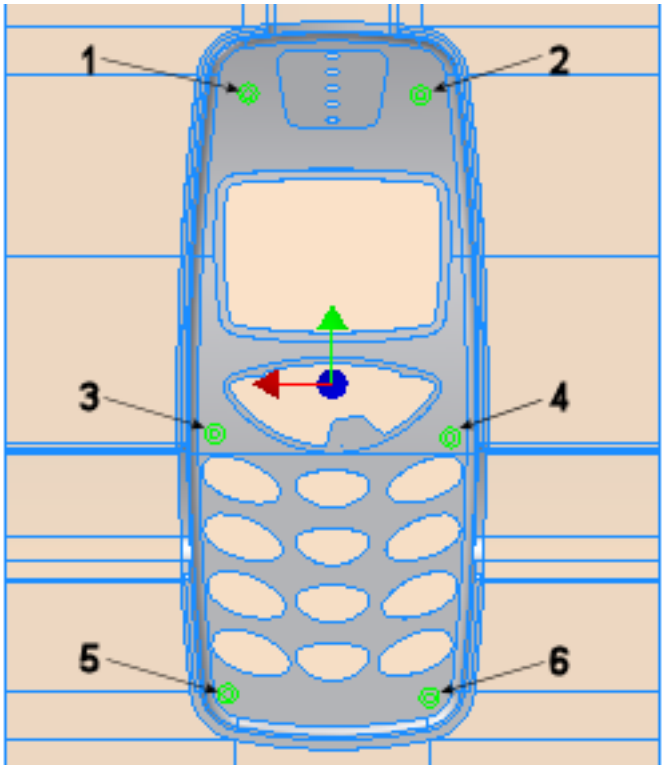
Previous (page 22) | Next (page 27)

Inserting Ejector Pins

In this section, you create the ejector pins to eject the plastic part. In a typical mold, the section attached to the fixed side of the press is referred to as the A side. The section attached to the moving side of the press is referred to as the B side.

Ejectors are pins or sleeves that push the plastic part out of the mold. Ejector pins are made of steel and are normally located on the B side of the mold. When the ejection system is activated, the pin pushes the plastic part or runner system out of the mold.

1. On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Ejector**  .
2. Modify the **d1** parameter value to **3 mm**.
3. Modify the **L** parameter value to **160 mm**.
4. Create six points in approximately the same locations as the points highlighted in the following image.



5 Select the arrows in the bottom right-hand corner of the dialog box to expand and view the list of ejectors. Edit the **X** and **Y** coordinates as shown in the following table.

Ejector	**X**	**Y**
EP1	14 mm	48 mm
EP2	-14 mm	48 mm
EP3	17 mm	-7 mm
EP4	-17 mm	-7 mm
EP5	15 mm	-48 mm

EP6	-15 mm	-48 mm

6 On the **Ejector: Mobile Top** dialog box, select the **Clearance** tab.

7 Change the clearance values for the ejector pins by entering the following values:

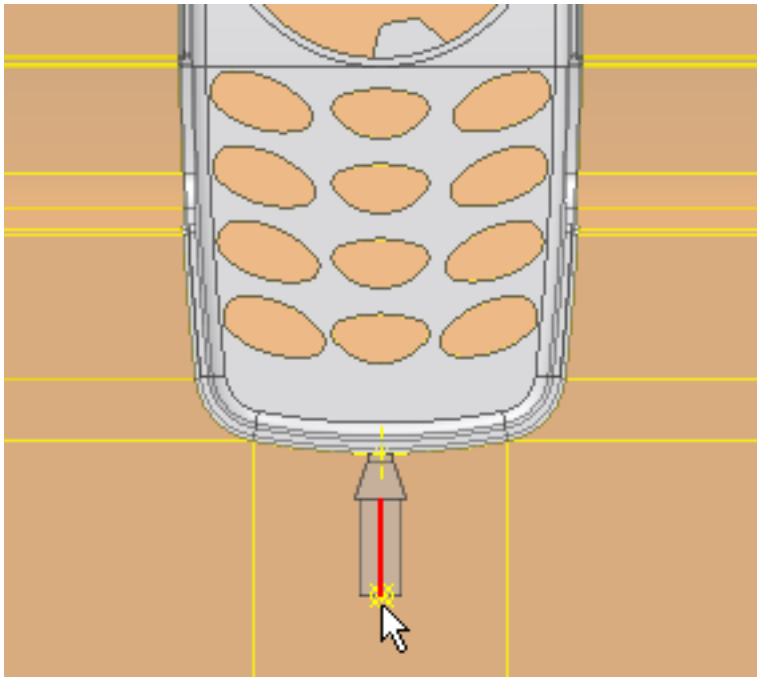
- CH = **7.0 mm**
- CEP = **3.6 mm**
- CB = **3.4 mm**

8 On the **Ejector: Mobile Top** dialog box, click **Apply.**

9 Click **OK** in the **File Naming** dialog box to accept the defaults and create the ejector pins.

10 From the **Ejector: Mobile Top** dialog box, select the **Design** tab and click **Eject Runner System**.

11 Select the end point of the runner sketch, as shown in the following image.



12 Click **OK** on the **Ejector** dialog box.

13 Click **OK** on the file naming dialog box to create the ejector pin and close the dialog box.

Previous (page 24) | Next (page 30)

Creating a Sprue Bushing and a Locating Ring

In this section, you add a sprue bushing and a locating ring to the mold design, and connect the sprue bushing to the runner.

The sprue bushing provides an opening through which plastic melt is injected into the mold. The plastic melt is directed by the sprue bushing through the runners, past the gate, and into the mold cavity. In some mold assemblies, the plastic melt flows directly into the cavity from the sprue bushing. The locating ring fits over the sprue bushing and aligns it with the injector nozzle.

1 On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Sprue Bushing**  .

2 From the **Sprue Bushing** dialog box, expand the **Type** list.

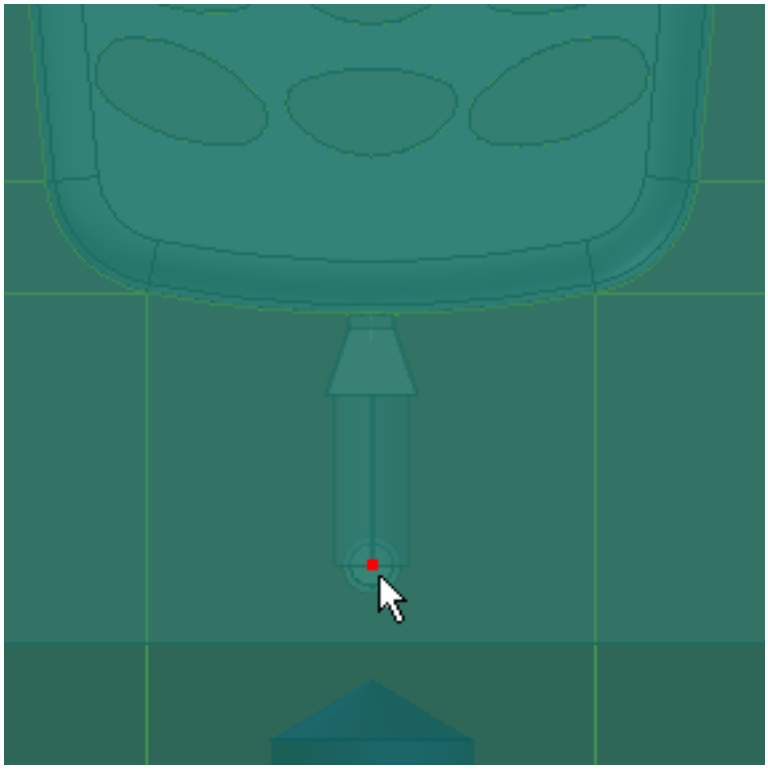
3 From the Vendor list, select **DME**, and then select **AGN** as the sprue bushing type.

4 Modify the following values:
- L = **76 mm**
- d1 = **3.5 mm**

5 In the Offset section of the **Sprue Bushing** dialog box, change the **Z** value to -20 mm.

6 In the upper-right corner of the graphics window, click **Top** on the ViewCube to orient the model.

7 Activate the **Point** selection tool on the Sprue Bushing dialog box, and select the end point of the runner sketch, as shown in the following image.



8 Click **OK** on the **Sprue Bushing** dialog box.

9 Click **OK** on the **File Naming** dialog box to accept the defaults and create the sprue bushing.

NOTE This location is for the sprue for a single cavity mold, but it illustrates how to create positions for the sprue bushing and the locating ring.

10 On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Locating Ring** .

11 From the **Locating Ring** dialog box, expand the **Type** list.

12 From the Vendor List, select **DME**.

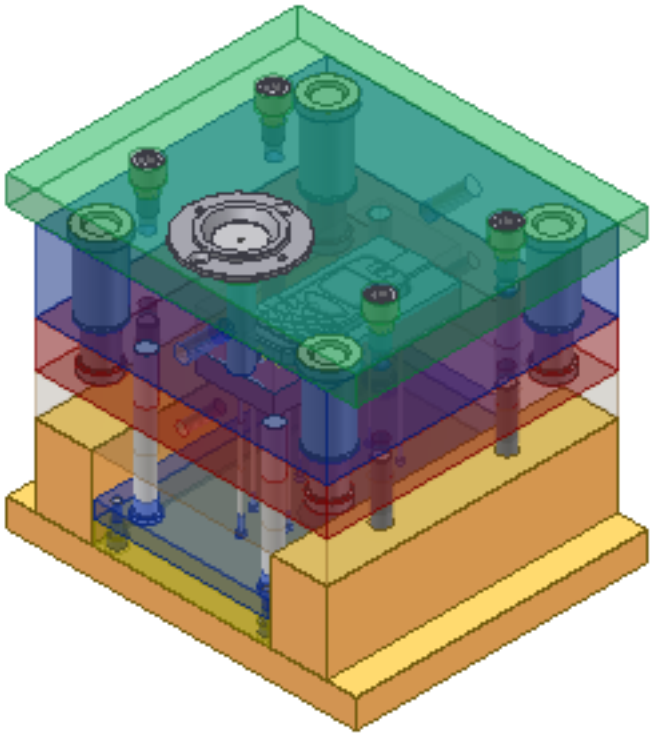
13 Set the Category to Locating Rings and then select **DHR 21 -Type A** as the locating ring type.

14 Modify the **Z** offset to **4 mm**.

15 Click **OK** on the **Locating Ring** dialog box.

16 Click **OK** on the **File Naming** dialog box to accept the defaults and create the locating ring.

17 On the keyboard, press **F6** to return to the Home View. The sprue bushing and locating ring are shown in the following image.



Previous (page 27) | Next (page 32)

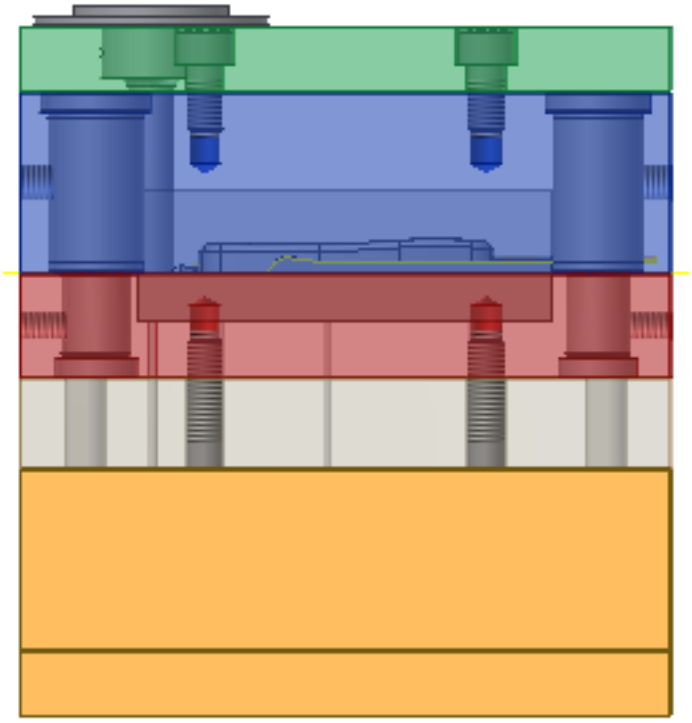
Creating Cooling Channels

In this section, you create a cooling channel for the mold design.

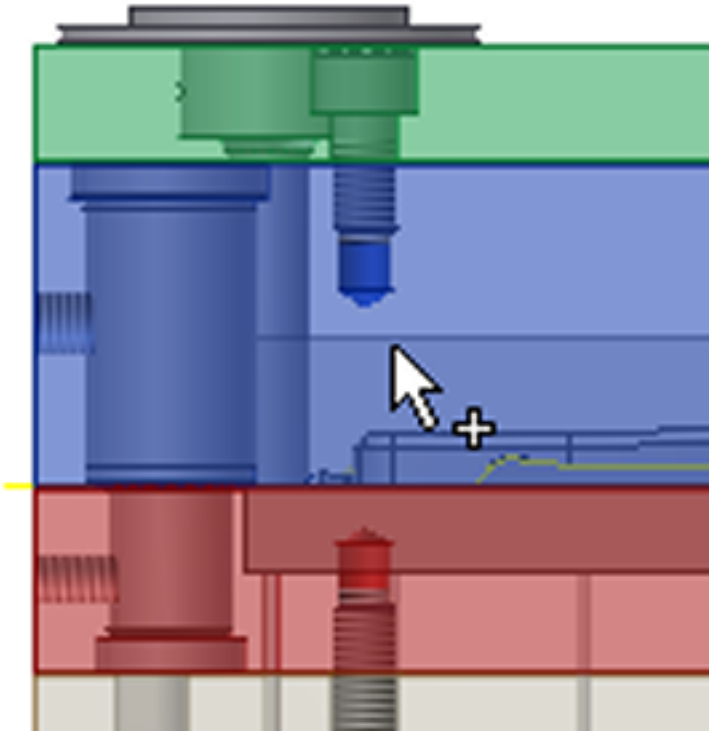
The cooling channel circulates a coolant through the mold to cool down the mold quickly and uniformly.

1 On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Cooling Channel** .

2 In the upper-right corner of the graphics window, click **Right** on the ViewCube to orient the model.



3 Activate the Face selection tool. Select the location shown in the following image.

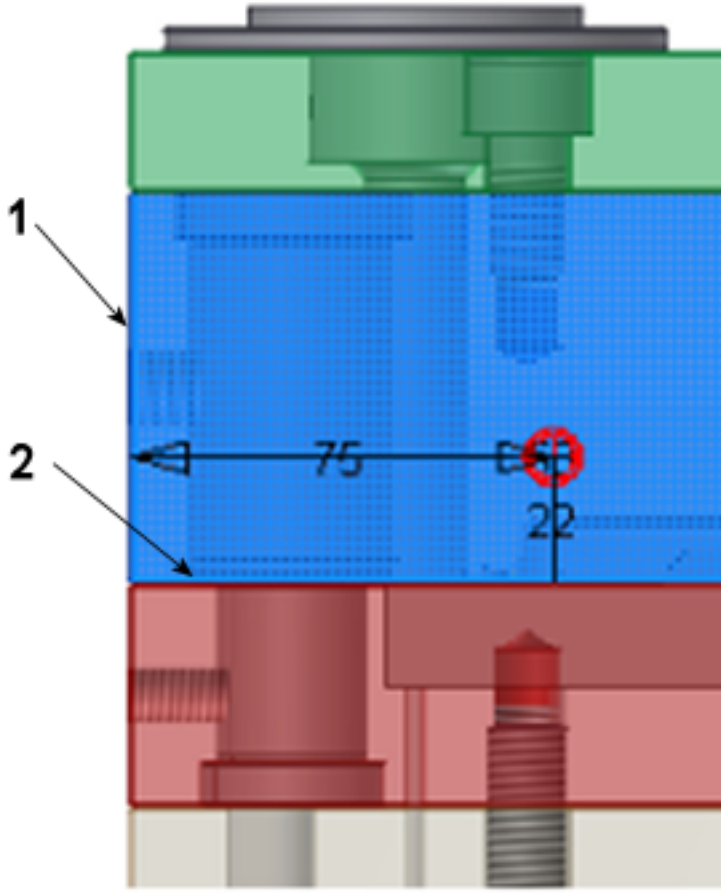


4 Select the edge identified as 1 (outside of chamfer) in the following image as the **Linear Edge 1** reference.

5 Select the edge identified as 2 (chamfer) in the following image as the **Linear Edge 2** reference.

6 Edit the X and Y axis locations as follows:

- Distance from Linear Edge 1 = 75 mm.
- Distance from Linear Edge 2 = 22 mm.

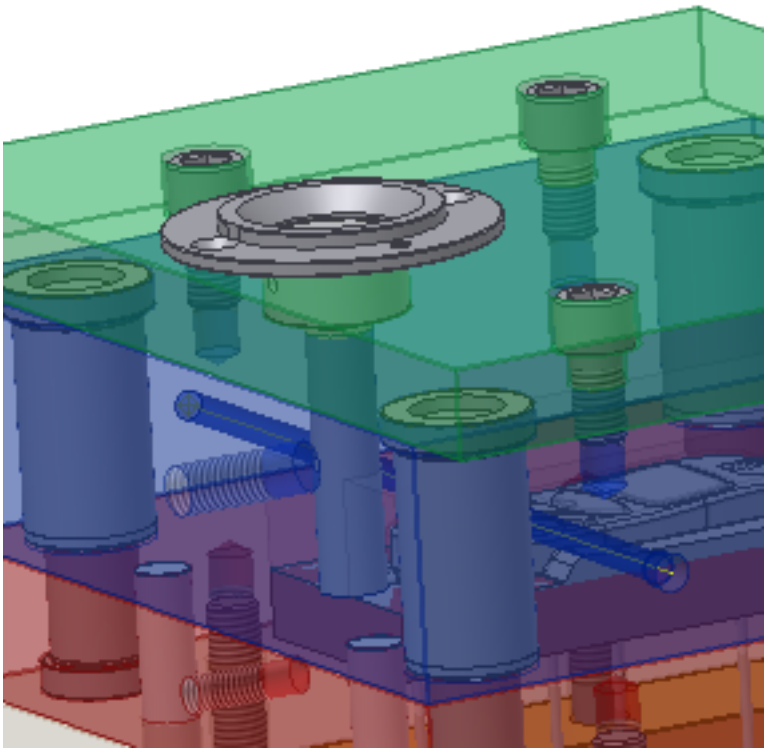


7 From the Extents List on the **Cooling Channel** dialog box, select **Through All**.

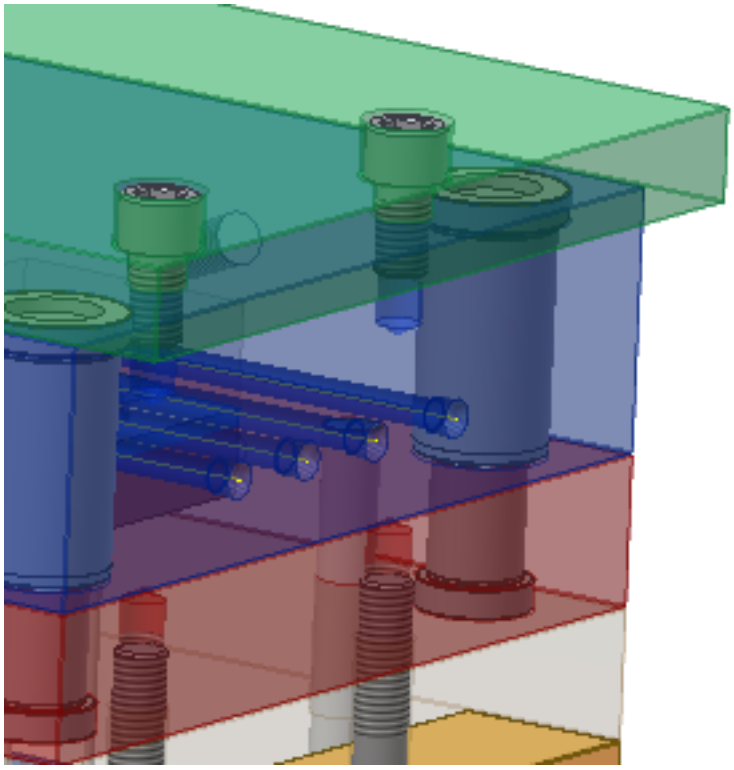
8 Click the **Symmetrical** check box.

9 Click **Apply**.

10 Click OK to accept the defaults on the file naming dialog box. The cooling channel appears as shown in the following image.



11 Repeat steps 3 through 10 to create three more cooling channels, as shown in the following image.



12 The distances from Linear Edge 1 and 2 for each of the cooling channels are listed in the following table. Make sure that you select the outside edges and not the chamfered edges.

Cooling Channel	Distance from Linear Edge 1	Distance from Linear Edge 2
Cooling Channel 2	105 mm	22 mm
Cooling Channel 3	135 mm	22 mm
Cooling Channel 4	170 mm	22 mm

13 Click Apply to create each cooling channel. After all four cooling channels have been created, click **Done** on the **Cooling Channel** dialog box.

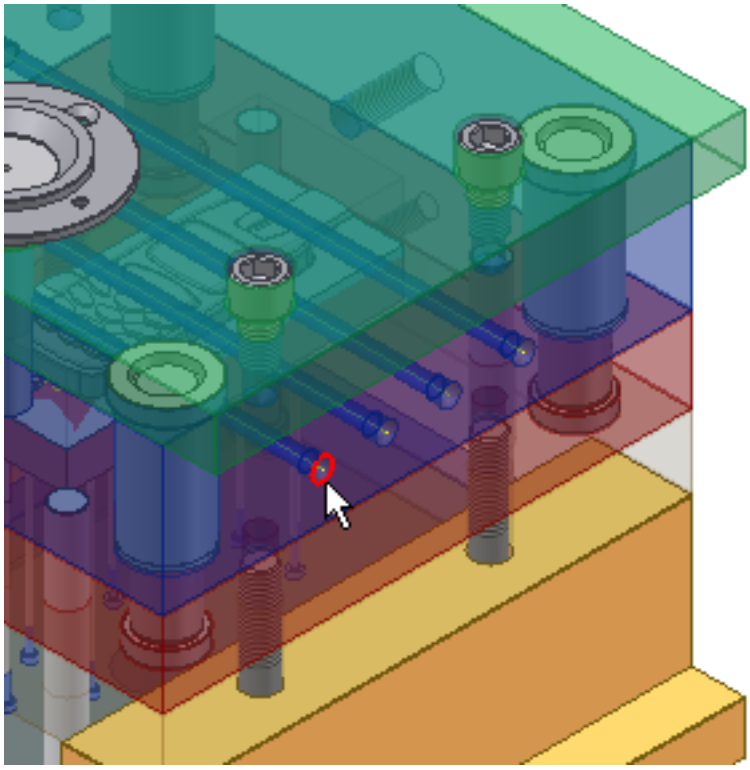
Previous (page 30) | Next (page 38)

Adding Cooling Components

In this section, you add connector plugs to the ends of the cooling channels.

Cooling components are used to conduct the coolant through the cooling channels. The following eight types of cooling components can be created: Baffle, Cascade Water Junction, Connector Plug, Heat Transfer Pipe, O-Ring, Pipe Plug, Sealing Plug, and Spiral Core.

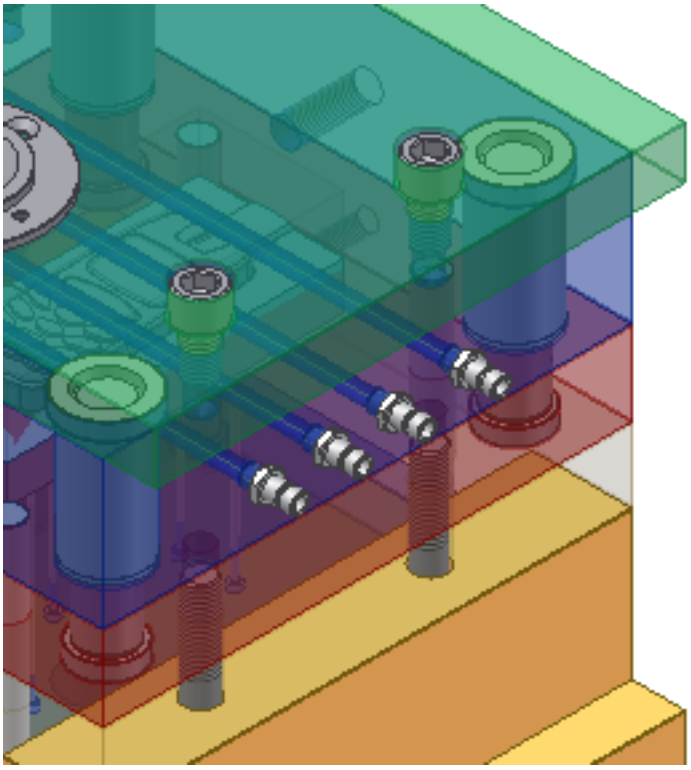
1. In the upper-right corner of the graphics window, click **Home** on the ViewCube to orient the model.
2. On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Cooling Components**  .
3. The **Cylindrical Edge** tool is active. In the graphics window, select the cylindrical edge, as shown in the following image.



4 Click **Apply** on the **Cooling Components** dialog box.

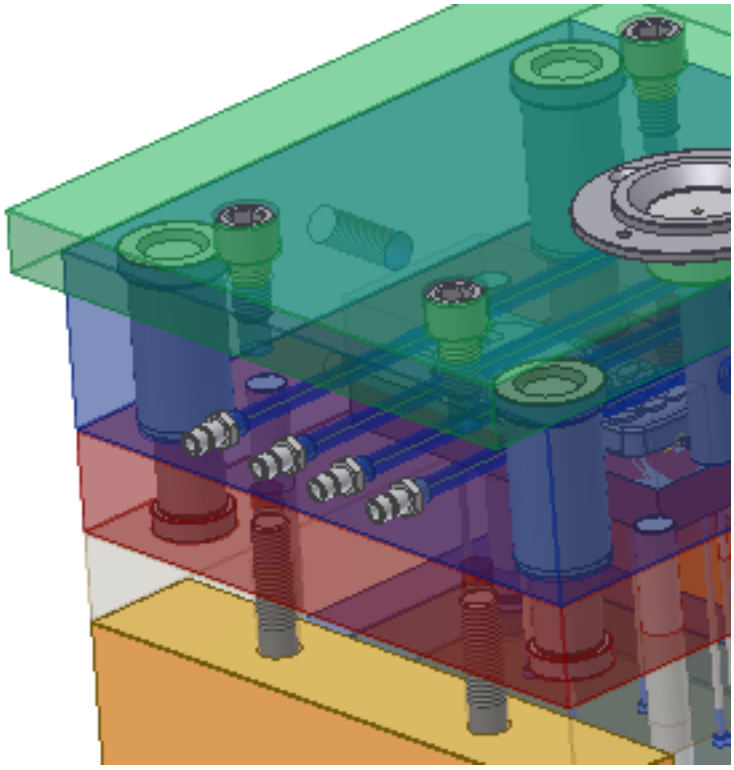
5 Click **OK** to accept the defaults in the **File Naming** dialog box.

6 Repeat steps 4 and 5 to add a cooling component to the end of each of the remaining three cooling channels, as shown in the following image.



7 Rotate the model so that the other sides of the cooling channels are visible.

8 Repeat steps 4 and 5 to add a cooling component to the end of each cooling channel, as shown in the following image.



9 After all eight cooling components have been added, click **Done** on the **Cooling Components** dialog box.

Previous (page 32) | Next (page 41)

Documenting the Mold Assembly

In this section, you use the 2D Drawing command to generate a drawing.

Inventor automatically generates a 2D drawing of the mold design, which includes basic views, dimensions, part lists, balloons, and hole table information.

1 On the Quick Access toolbar, click **Save** .

2 On the **Save** dialog box, click **OK** to save the assembly and its dependents.

3 On the ribbon, click **Mold Assembly tab** > **2-D Drawing panel** > **2-D Drawing**  .

The **2-D Drawing** dialog box lists all the available drawings that can be created for your mold design. You can isolate the drawings that you would like to create by using the Drawing List menu. The Drawing List groups can be modified from the **Drawing Filter** dialog.

1 From the **2-D Drawing** dialog box, click **Drawing filter**  .

2 From the **Drawing Filter** dialog box, you can add and remove groups, and customize the existing groups.

3 Click **Cancel** on the Drawing Filter dialog box.

Drawings can be generated from any of the files that make up the mold design. In this section, drawings of the overall mold design, the core, and the cavity is created.

1 From the **2-D Drawing** dialog, select the following:
- Mold Design 1.iam
- Mold Design1_Mobile Top_CR_1:1
- Mold Design1_Mobile Top_CV_1:1

2 Click inside the **Template** column for the **Mold Design1.iam** row.

3 From the **Select Template** dialog box, select the **Metric** tab. Notice that both idw and dwg templates are available for drawings.

4 Click **ISO.idw**, and click **OK**.

5 Ensure that the following drawings are also using the **ISO.idw** template:
- Mold Design1_Mobile Top_CR_1:1
- Mold Design1_Mobile Top_CV_1:1

6 Click inside the **Sheet Format** column for the **Mold Design1.iam** row, and select **Automatic Layout**.

7 From the **Automatic Drawing Setting** dialog, select the following options:
- Sheet Size = **A1**
- Scale = **Auto**

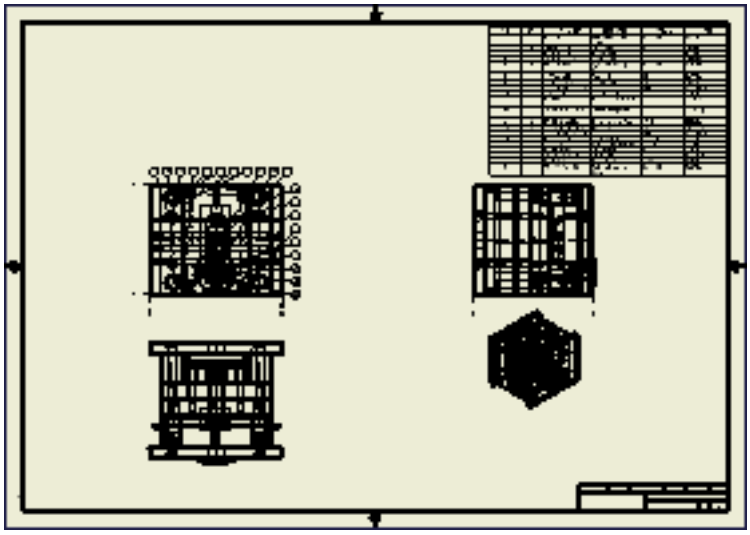
8 Click **OK**.

9 Click inside the **Sheet Format** column for the **Mold Design1_Mobile Top_CR_1:1** row, and select **A1 size, 6 views**.

10 Change the sheet format for **Mold Design1_Mobile Top_CV_1:1** to **A1 size, 6 views**.

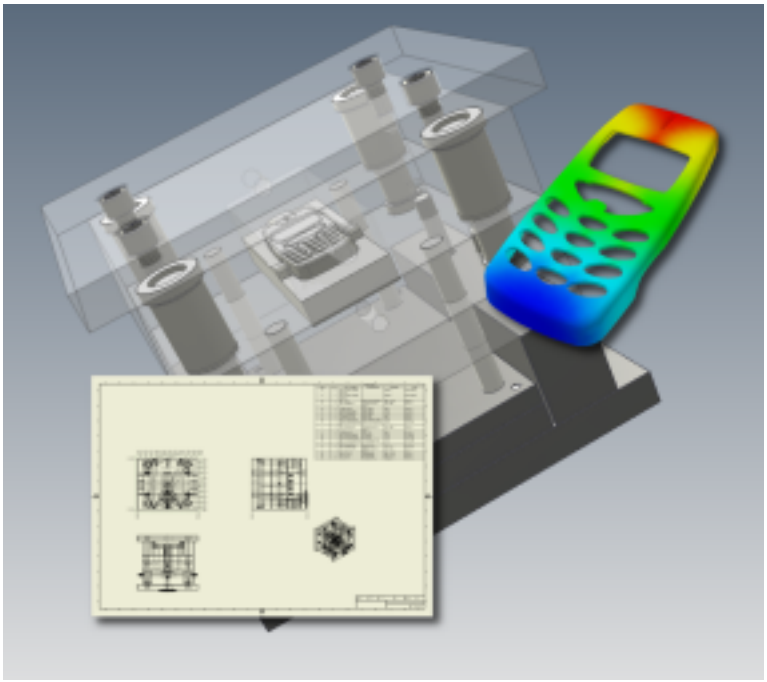
11 Click **OK** in the **2-D Drawing** dialog box.

12 Click **OK** in the **File Naming** dialog box to accept the defaults and generate the drawings.



Previous (page 38) | Next (page 44)

Summary



In this tutorial, you opened an Inventor part file in the mold application and used it to step through the process of mold base design.

You learned how to:

- Generate the core and cavity.
- Create the feeding system, including runners, gates, sprues, sprue bushings, and locating rings.
- Select a mold base.
- Create ejector pins.
- Create the cooling system, including cooling channels and cooling components.
- Generate a drawing from your mold design.

What next?

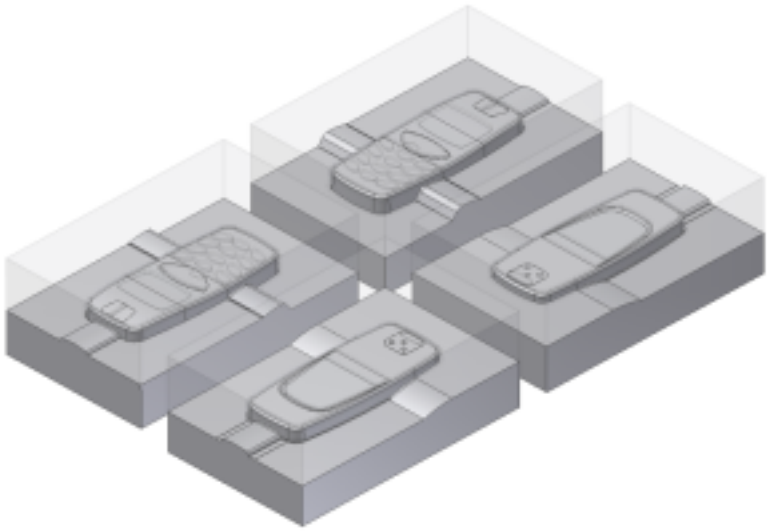
Try the tutorial on Family Mold Design (page 47).

Previous (page 41)

Family Mold Design

2

About this tutorial



Create a mold assembly with multiple moldable parts.

Skill level	**Level 3**
Time required	60 Minutes
Prerequisites	■ Know how to set the active project and navigate the model space with the vari-

	ous view commands. See the Inventor Help section *Getting Started* for further information. ■ Completion of the About this tutorial (page 1) Tutorial.
Tutorial files used	Exp-Mobile Bottom.ipt, Exp-Mobile Top.ipt, Mold.ipj

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Navigation

Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Tutorial objectives

In this tutorial, you discover how to place multiple components in the Mold Design environment. After placing multiple components, you will use patterning functionality to pattern each of the components that were placed in the mold assembly. The tutorial progresses through the standard mold workflow. This tutorial is intended to provide an overview of the features and functions of the Inventor Mold Design application. The example uses a simplified part and is intended to depict concepts and procedures - it is not intended as a primer in good mold design. After completing this tutorial, you will be able to:

- Use Inventor Mold Design to create a family mold consisting of multiple moldable parts.
- Adjust the orientation and position of components placed in the Mold Design environment.
- Assign the material and its shrinkage value.
- Generate the core and cavity for the moldable parts.
- Use patterning functionality to generate multiple components quickly in the mold assembly.

- Define the runner system, the gates, and the mold base for the mold assembly.

Next (page 49)

Getting Started and Placing Moldable Parts

In this topic, you place two different components into the mold environment.

1. Set the project to **Mold.ipj**.
2. If you did not complete the General Mold Workflow tutorial, ensure that the Mold library is attached to this project file. In the **Projects** dialog box, click **Configure Content Center Libraries**  . In the **Configure Libraries** dialog box, scroll to the Inventor Mold libraries and ensure that the **In Use** check box is enabled. Click **OK**. If this library is not enabled, you cannot add the mold base later in the tutorial.
3. Click Inventor  to start a new mold assembly.
4. Select **New** to open the **New File** dialog box.
5. From the **Metric** tab of the **New File** dialog, click **Mold Design (mm).iam** and then click **OK**.
6. In the **Create Mold Design** dialog, click **OK** to accept the defaults. The Mold Design environment is now available.
7. In the Mold Design browser, right-click the mold icon and choose **Prompt for file names** in the context menu. You can control the names and locations of system generated files.

   NOTE The enabled or disabled state of the file name setting is persistent between sessions. If the setting is not checked, the File Naming dialog box does not appear.

8. On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Plastic Part**  .
9. In the **Plastic Part** dialog, select **Exp-Mobile Top.ipt** in the Mold folder and click **Open**.

10 Left click in the graphics window to place the part with the default alignment option.

11 Click **OK** on the **File Naming** dialog box to accept the defaults. The Inventor part file is placed in the Mold Design environment. This model contains imported geometry of the top section of a mobile phone.

12 Click **Plastic Part**  again.

13 Select **Exp-Mobile Bottom.ipt** and click **Open**.

14 Click in the graphics window to place the bottom part as shown in the following image.

15 Click **OK** on the **File Naming** dialog box to accept the defaults. This model contains the exported geometry of the bottom section of a mobile phone.

You have assembled two plastic parts in the mold assembly. Additional parts can be added, if necessary. In this tutorial, you only add two components.

Previous (page 47) | Next (page 50)

Adjusting Component Orientation and Position

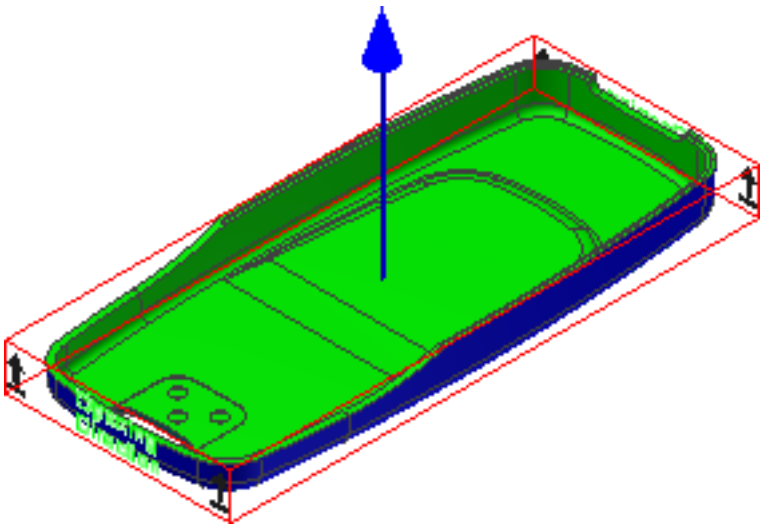
In this setion, you change the orientation of the Exp-Mobile Bottom part so it is consistent with the Exp-Mobile Top part.

In the General Mold Workflow tutorial you did not modify the orientation or position of the part. This mold assembly requires you to make some modifications. You use the **Adjust Orientation** command to modify the direction of a plastic part. The default direction of the mold opening does not align with the positive Z axis. You also ensure that the position of components is consistent. The **Adjust Position** command enables you to specify the position of plastic parts when there are multiple parts in a family mold.

1 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Adjust Orientation** .

2 Ensure that the **Plastic Part** selector option is active .

3 Select the **Exp-Mobile Bottom.ipt** part in the graphics window. The model displays the pull direction of the part, as shown in the following image. The orientation is not correct.



4 Click **Align with Axis** .

5 Click **Flip moldable part** .

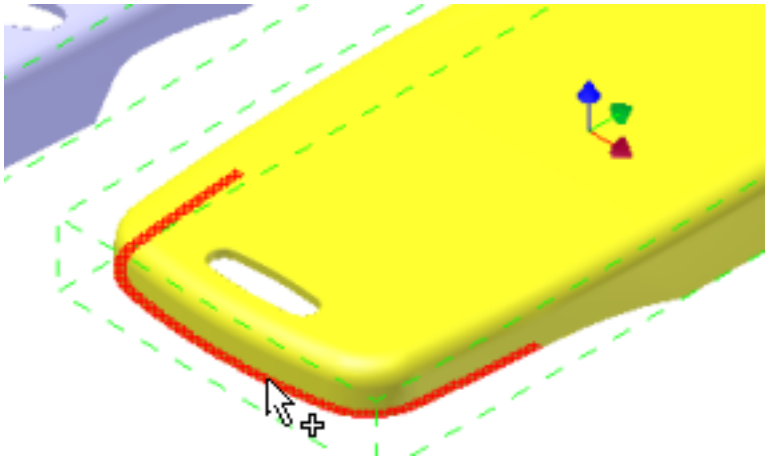
6 Click **Done**. The models appear as shown in the following image. The orientation is now correct.

7 Reorient the models to the Front view by using the ViewCube. Your model varies depending on how you placed the Exp-Mobile Bottom.ipt file. However, you must adjust the positioning of the two components relative to one another for the family mold, regardless of placement.

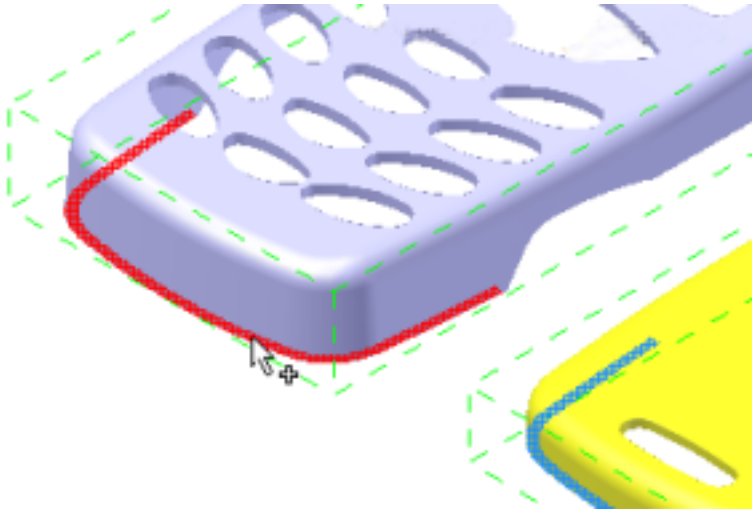
8 Right-click in the graphics window and click **Previous View** to return to the previous 3D view of the mold assembly.

9 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Adjust Position** . The Adjust Position command is in drop-down next to Adjust Orientation.

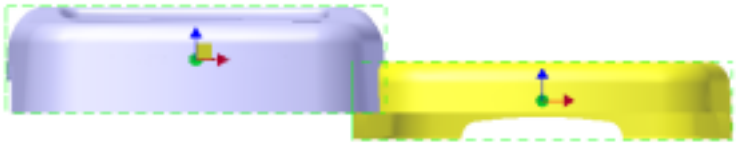
10 Ensure that the **Plastic Part** option is selected . Select the **Exp-Mobile Bottom.ipt** part, which turns yellow, and then select the surface highlighted in red in the following image.



11 Ensure that the **Reference** option is selected . Select the **Exp-Mobile Top.ipt** part, which retains its original color, and then select the surface highlighted in red in the following image.



12 Reorient the models to the Front view by using the ViewCube. The models are like the following image.



13 Adjust the X and Y offsets as follows:
- X Offset = **120**
- Y Offset = **0**

14 Click **OK**. Right-click in the graphics window and click **Previous View** to return to the previous 3D view of the mold assembly. The mold assembly appears as shown in the following image.

Previous (page 49) | Next (page 54)

Defining Material and Shrinkage

In this section, you define the material and the shrinkage.

1 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Select Material** .

2 From the list of Manufacturers, select **SABIC Innovative Plastics US, LLC**.

3 From the list of Trade names, select **Cycoloy C2950 HF-71676**.

4 Click **OK**.

NOTE The material is also listed in the Commonly used materials panel if you completed the General Mold Workflow tutorial.

5 In the Mold Design browser, double-click the **Exp-Mobile Top** node to activate it.

6 On the ribbon, click **Core/Cavity tab** > **Plastic Part panel** > **Part Shrinkage**  .

7 In the **Part Shrinkage** dialog box, notice that the shrinkage range is between .21% and .72%. The system has assumed an isotropic value of .49%. Maintain the default isotropic shrinkage value and click **OK**. There is no need to review the shrinkage value of the Exp-Mobile Bottom model because the default .49% value is appropriate.

8 Click Cancel to close the dialog box.

9 On the **Exit** panel, click **Finish Core/Cavity**  .

Previous (page 50) | Next (page 55)

Generating the Core and Cavity

In this section, you define the workpieces, patching and runoff surfaces, and core and cavity for the Exp-Mobile Top and Exp-Mobile Bottom components.

Each of these commands must be completed on both the Exp-Mobile Top and Exp-Mobile Bottom components. When a command is completed on one of these models, its other patterned instance automatically updates.

Defining the Exp-Mobile Top component

1 In the Mold Design browser, double-click the **Exp-Mobile Top** node to activate it.

2 On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Define Workpiece Setting**  .

3 Maintain the default Rectangular Workpiece Type and change the following:
   - X_total = **100 mm**
   - Z_total = **60 mm**
4 Click **OK**.
5 Click **OK** on the **File Naming** dialog box to accept the defaults. The workpiece appears as shown in the following image.

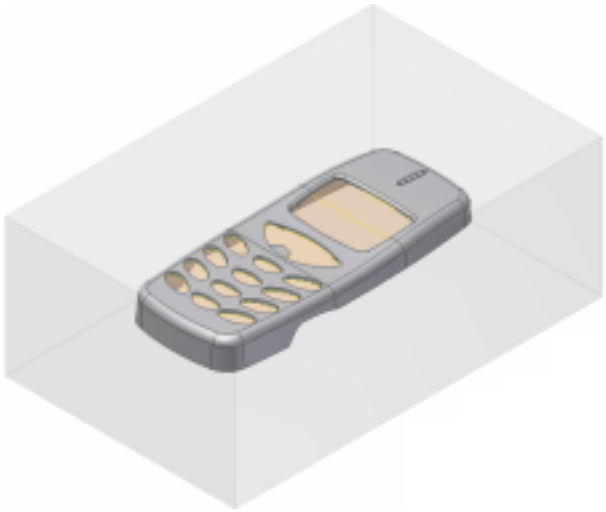
6 Click **Create Patching Surface** .

7 Click Auto Detect on the dialog box. The system automatically detects 19 patches and lists them by name in the **Create Patching Surface** dialog box. Some of the openings may not be successfully patched until you perform the next step.
8 To achieve higher quality surfaces in the final product, click **All Lower** to switch the patching surfaces to the lower edges of the model .

NOTE Different models have been used in the General Mold Workflow tutorial and this tutorial, so the patching surface procedures vary. It is important to remember that the Create Patching Surface command is a starting point for patch surface creation. Reviewing the surfaces that are created and using manual patching techniques where necessary is recommended.

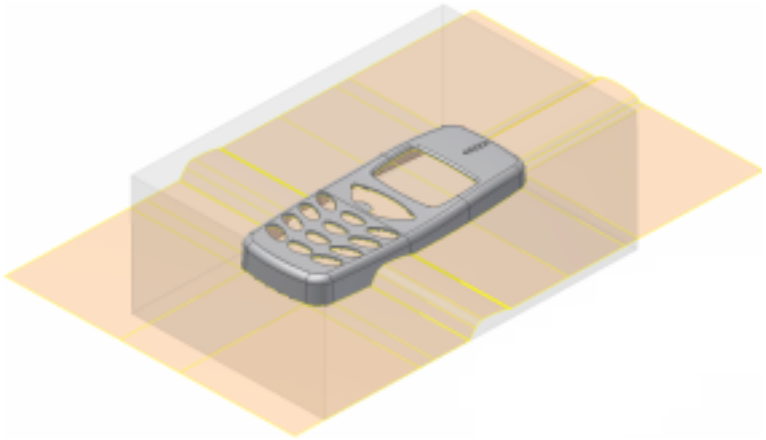
Refer to the tutorial section*Defining the parting surface* in the *General Mold Workflow* tutorial to see an animation that shows creating patching surfaces manually.

9 Click **OK**. The surfaces are created as shown in the following image.



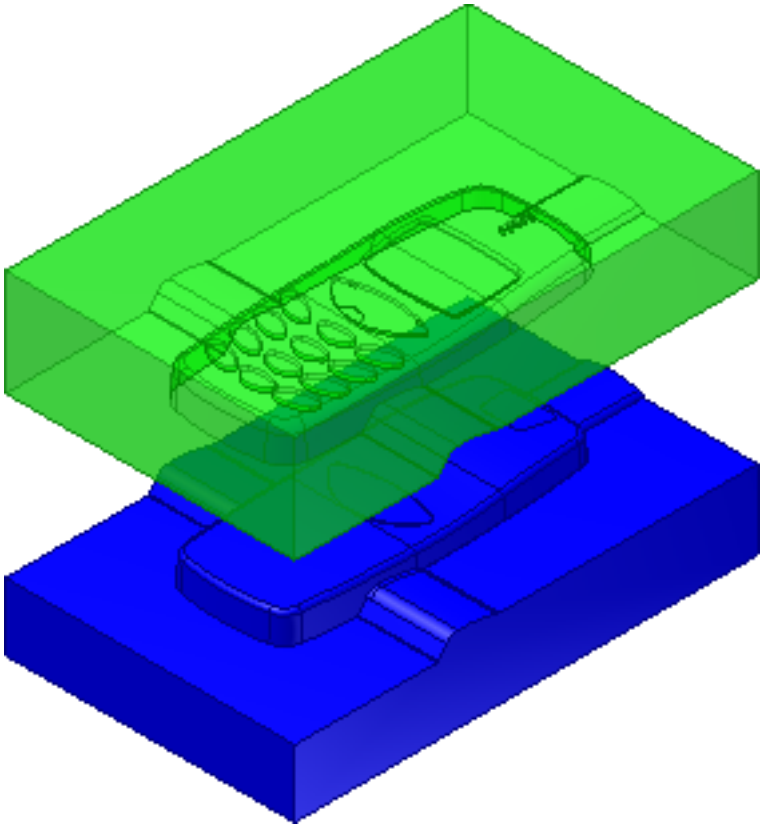
10 Click **Create Runoff Surface** .

11 Click Auto Detect on the dialog box and then click **OK** to accept the default runoff surface. The surface is shown in the following image.



12 Click **Generate Core and Cavity** .

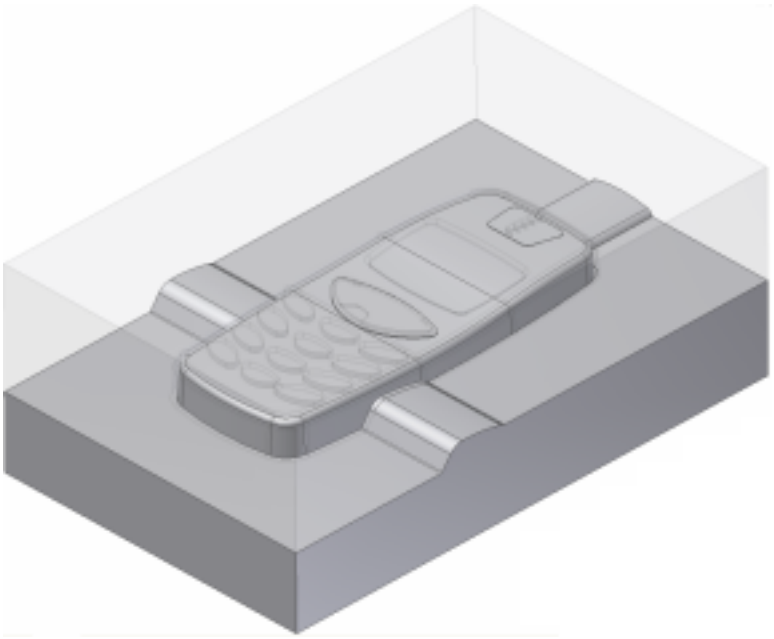
13 Click **Preview/Diagnose**. Change the **Body Separation** value to 100. The preview changes to the specified value as shown in the following image.



14 Click the **Parting Diagnostics** tab. No significant problems are detected. Click **OK** to accept.

15 Click **OK** on the **File Naming** dialog box to accept the defaults.

16 The core and cavity is automatically generated, as shown in the following image.



Defining the Exp-Mobile Bottom component

1. In the **Mold Design** browser, double-click the **Exp-Mobile Bottom** node to activate it.
2. On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Define Workpiece Setting** .
3. Maintain the default Rectangular Workpiece Type and change the following:
   - X_total = **100 mm**
   - Z_total = **60 mm**
4. Click **OK**.
5. Click **OK** on the **File Naming** dialog box to accept the defaults.
6. Click **Create Patching Surface** .
7. Click Auto Detect on the dialog box. The system automatically detects one patch.

8 Click **OK**.

9 Click **Create Runoff Surface** .

10 Click Auto Detect on the dialog box and then click **OK** to accept the default runoff surface.

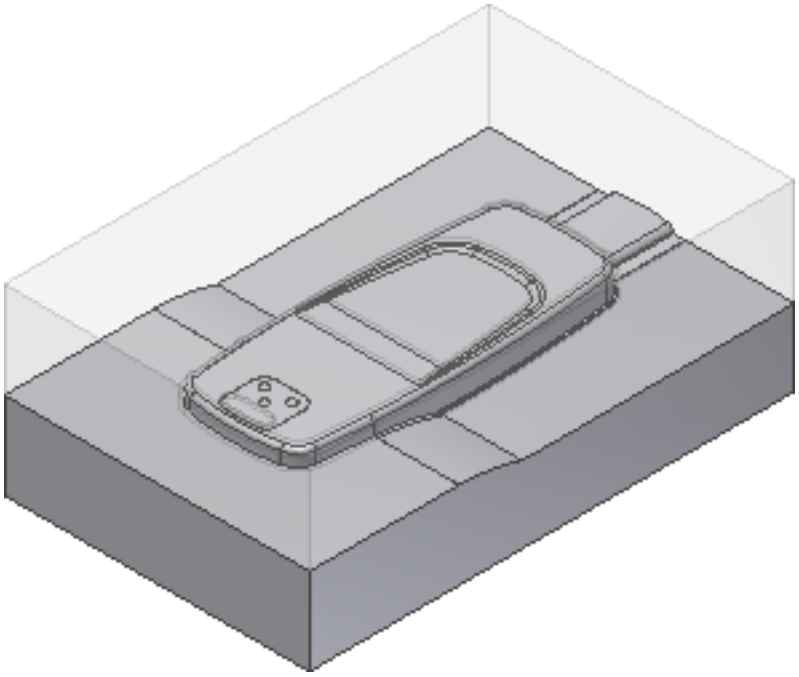
11 Click **Generate Core and Cavity** .

12 Click **Preview/Diagnose** to preview the core and cavity.

13 Click the **Parting Diagnostics** tab to check for errors.

14 Click **OK** on the **Generate Core and Cavity** dialog box.

15 Click **OK** on the **File Naming** dialog box to accept the defaults. The core and cavity are automatically generated, as shown in the following image.



16 On the **Exit** panel, click **Finish Core/Cavity** .

Previous (page 54) | Next (page 62)

Patterning Mold Assembly Components

In this topic, you use the Pattern command to design a multi-cavity mold in which the plastic parts are laid out in a rectangular pattern.

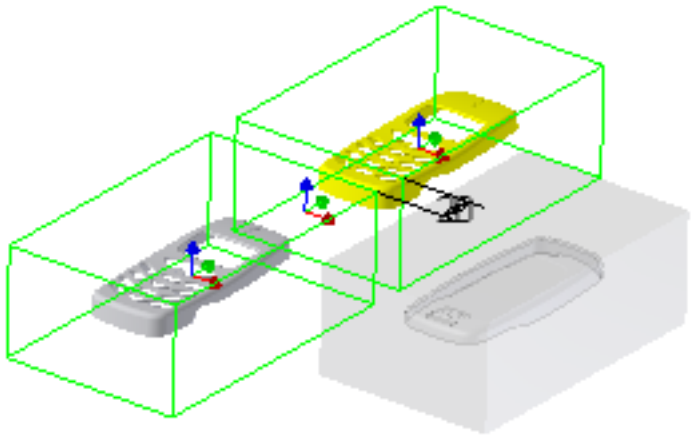
1 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Pattern** .

2 Ensure that the **Plastic Part** selector option is active.

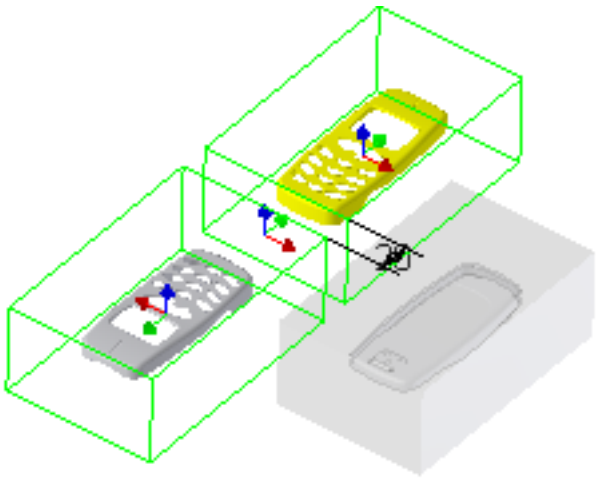
3 Select the **Exp-Mobile Top.ipt** part in the graphics window. Remain in the **Rectangular** tab to generate a rectangular pattern of the components.

4 In the Y Direction, enter the following:
- Number of instances = **2 ul**
- Offset = **180 mm**

The mold assembly appears, as shown in the following image.



5 Click **X Balance** . The orientation of the mold components changes, as shown in the following image.



6 Select the **Variable** tab and then select the **Inherit from the last pattern** option to maintain the rectangular pattern that was previously defined. Two elements representing the two instances are listed.

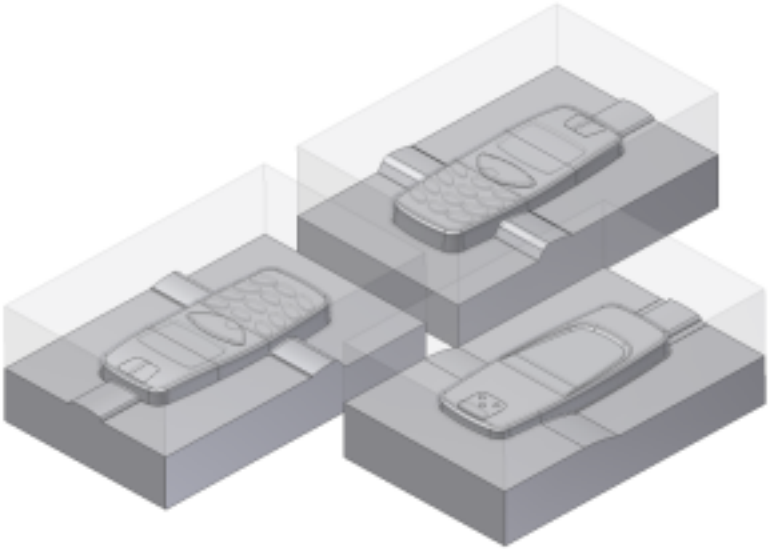
7 Modify the offset values to reposition the mold assembly coordinate to the center of the layout. In the X Offset, enter the following:

- Element 1 = **-60 mm**
- Element 2 = **-60 mm**

8 Expand the dialog box to expose the More options. Notice that both components are designated as unique files. This means that each component can have unique features instead of being identical copies.

9 Click **OK.**

10 Click **OK** on the **File Naming** dialog box to accept the defaults and complete the pattern.



11 Click **Pattern** .

12 Select the **Exp-Mobile Bottom.ipt** part in the graphics window. Remain in the **Rectangular** tab.

13 In the Y Direction, enter the following:

- Number of instances = **2 ul**
- Offset = **180 mm**

14 To change the orientation of the mold components, click **X Balance** .

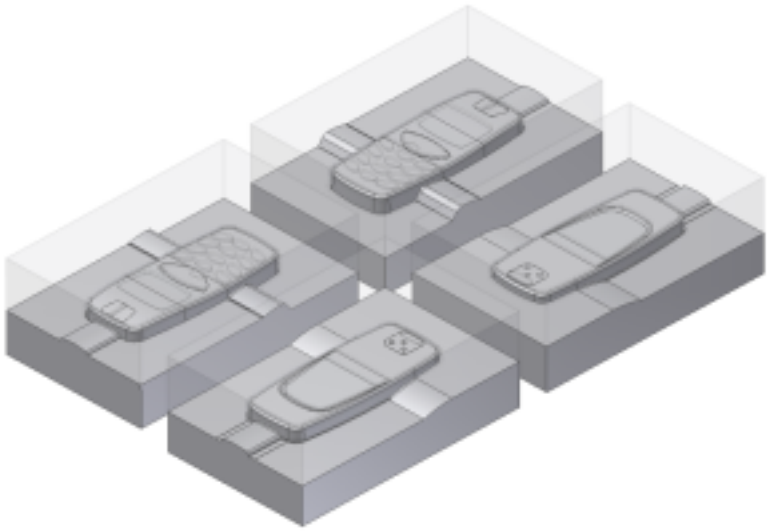
15 Select the **Variable** tab and then select the **Inherit from the last pattern** option to maintain the rectangular pattern that was previously defined. Two elements representing the two instances are listed. Notice that these files are also designated as unique files in the More section of the dialog box.

16 Modify the offset values to reposition the mold assembly coordinate to the center of the layout. In the X Offset, enter the following:

- Element 1 = **-60 mm**
- Element 2 = **-60 mm**

17 Click **OK**.

18 Click **OK** on the **File Naming** dialog box to accept the defaults and complete the pattern. The mold assembly appears as shown in the following image.



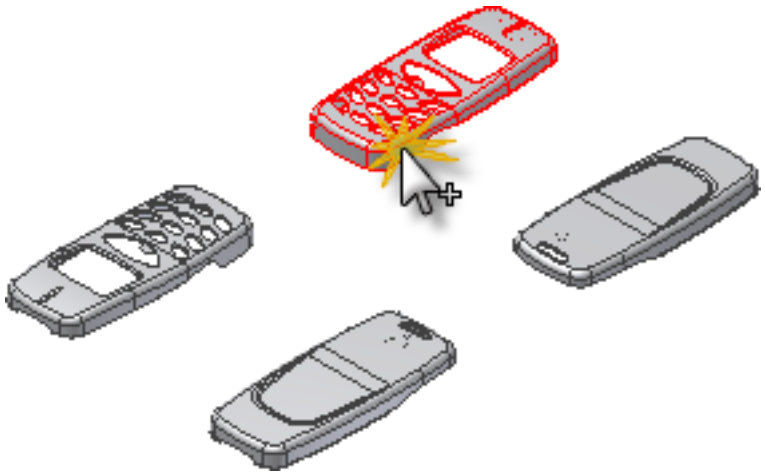
Previous (page 55) | Next (page 65)

Defining Gate Locations

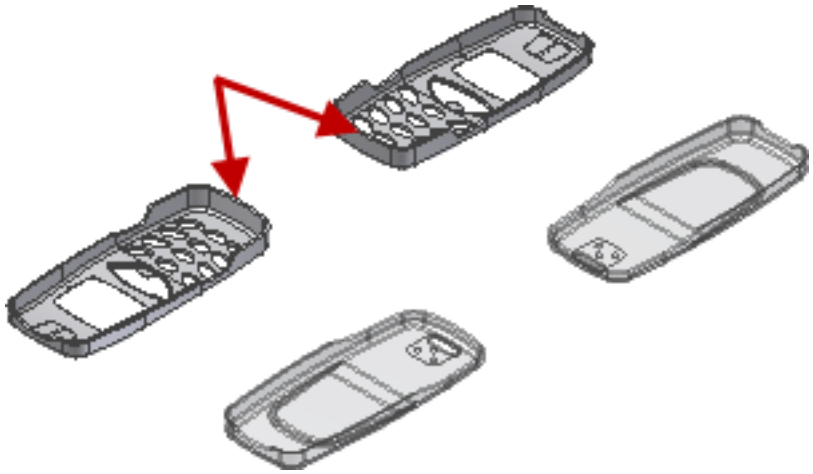
In this topic, you define the gate locations for the Exp-Mobile Top and Exp-Mobile Bottom components.

1 On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Gate Location**  .

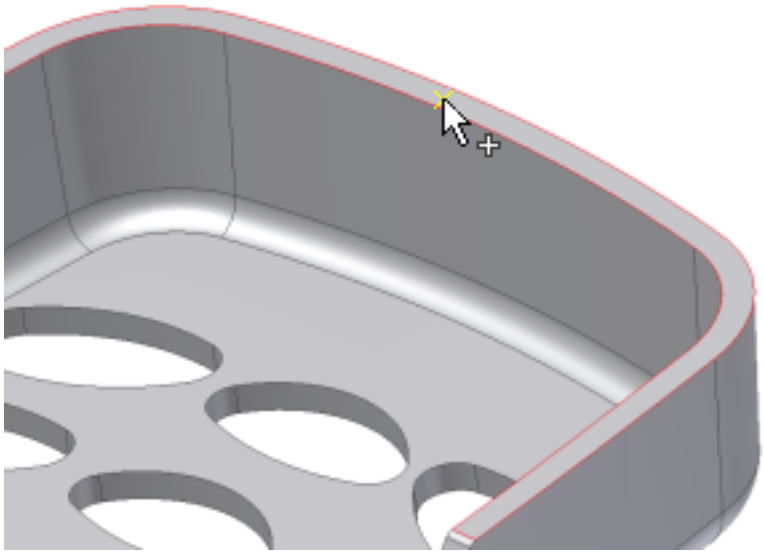
2 Select the model shown in the following image to satisfy the **Plastic Part** selection in the **Gate Location** dialog box.



3 The two instances of the Exp-Mobile Top model are available for specifying the Gate Location.Rotate the model to match the following image. We create a gate location in each of the areas noted.



4 Zoom in to display the end of the Exp-Mobile Top model shown in the following image. Select a point near the center of the thin planar surface to satisfy the **Location** selection.



Selecting on this surface constrains the gate location to this surface. The gate location on the surface is initially defined as the point that you selected. Exact U and V values can be assigned in the **Gate Location** dialog box to change this position.

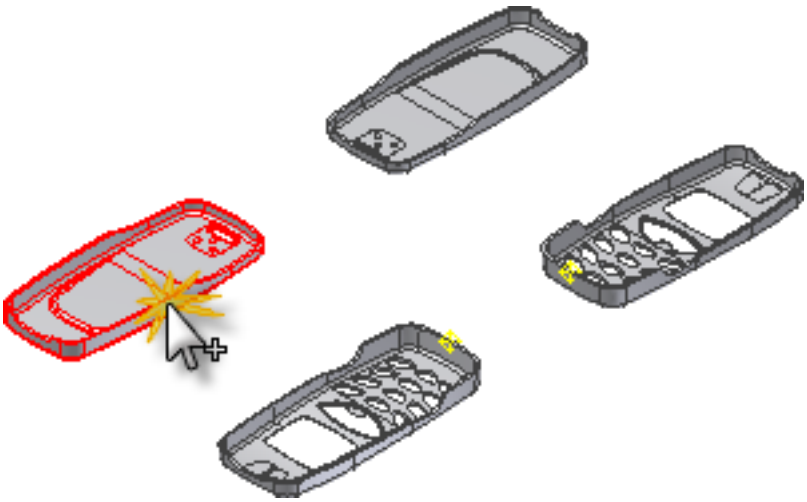
5 In the **Value** field of the **Gate Location** dialog box, enter the following:
   - Position U = **0.5**
   - Position V = **0.09**

6 Make sure **Copy to all pockets** is checked. This copies the gate location to all patterned elements. Click **Apply** to create the gate location on both instances.

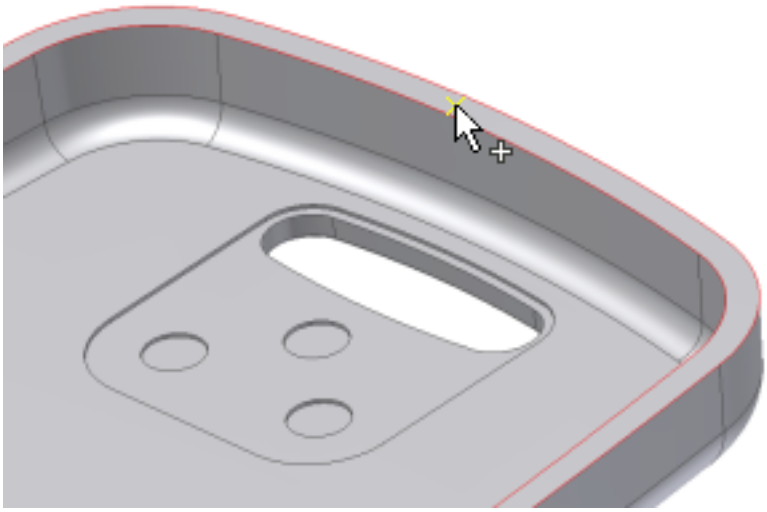
7 Click **OK** on the **File Naming** dialog box to accept the defaults.

8 Click the **Plastic Part** Part selector.

9 Select the component Exp-Mobile Bottom as shown in the following image.



10 Zoom to the area of the Exp-Mobile Bottom model shown in the following image. Select a point near the middle of the lower surface to satisfy the **Location**.



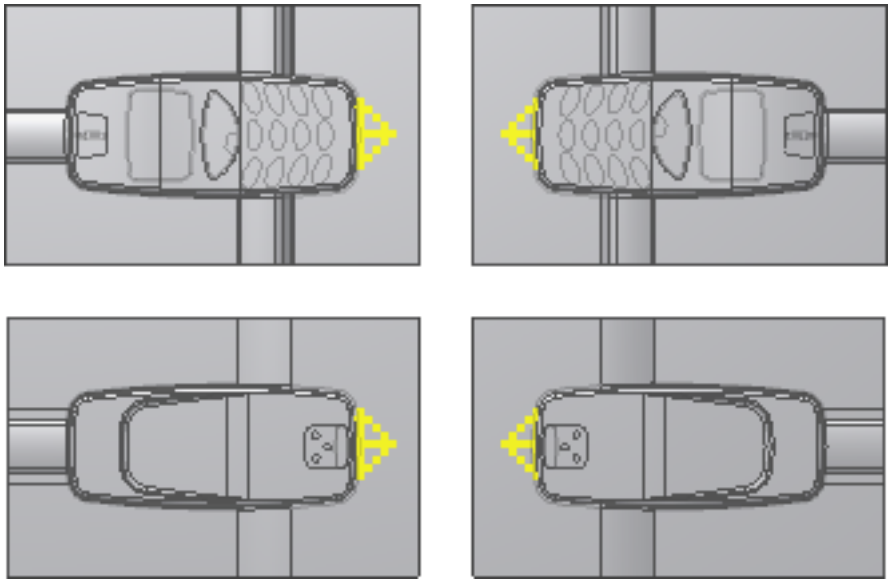
11 In the **Value** field of the **Gate Location** dialog, enter the following:

- Position U = **0.5**
- Position V = **0.04**

12 Make sure **Copy to all pockets** is checked. This copies the gate location to all patterned elements. Click **Apply** to create the gate location on both instances.

13 Click **Done** to finish the command.

14 The points appear in the mold assembly on all models and the Gate Location is present in the browser under each parent part.



NOTE The **Gate Location** dialog box provides a **Suggest** tab that runs a simulation on the moldable part. It also provides locations for a user-specified number of gates. The About this tutorial (page 79) tutorial explains how to run a Gate Location simulation.

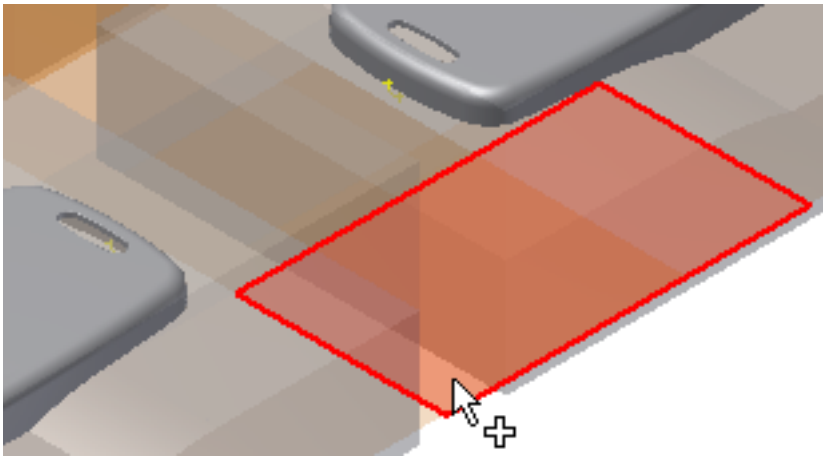
Previous (page 62) | Next (page 69)

Defining the Runner System

In this topic, you create the runner sketch by using a predefined runner balance and pattern type.

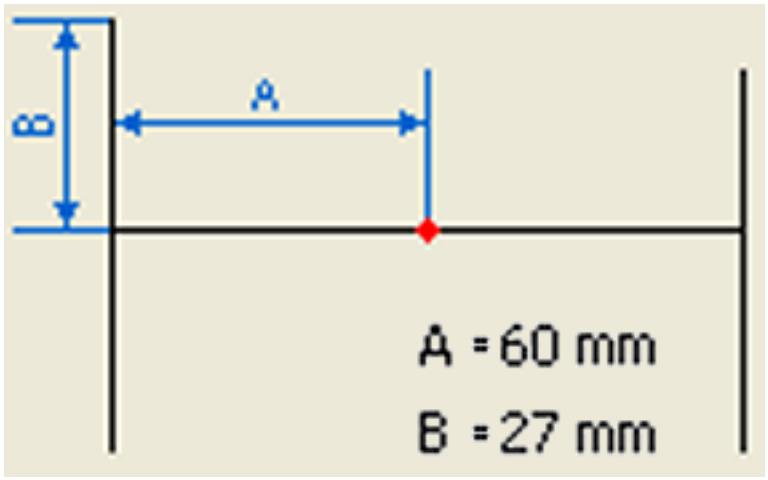
After the runner sketch has been created, you will add the runner to the mold assembly.

1. On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Auto Runner Sketch** .
2. Select the surface that is highlighted in the following image. This surface was generated as a runoff surface.



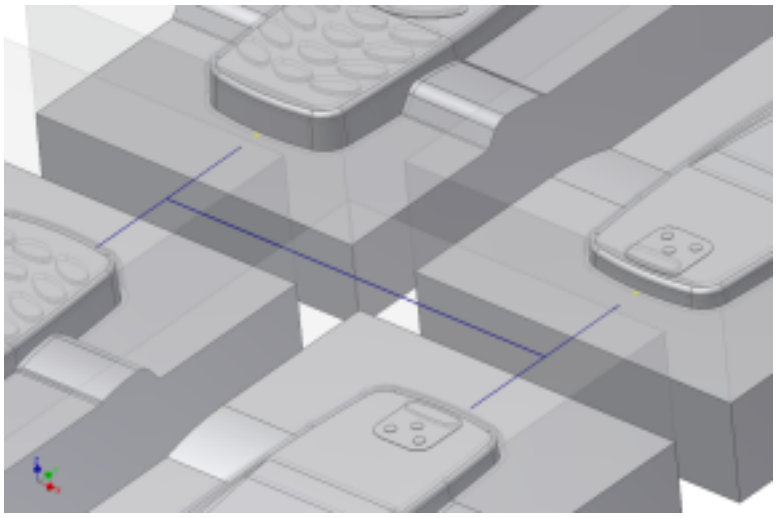
3. From the Balance menu, select **H Balance**.
4. From the Pattern menu, select **Pattern 1**.
5. In the Parameters section, enter the following:
   - A = **60 mm**
   - B = **27 mm**

   The A and B values define the runner as shown in the following image.



6 In the Option section of the dialog box, clear the **Activate Sketch Edit** option. In this exercise, you are not required to edit the sketch.

7 In the **Auto Runner Sketch** dialog box, click **OK**. The runner sketch appears as shown in the following image.



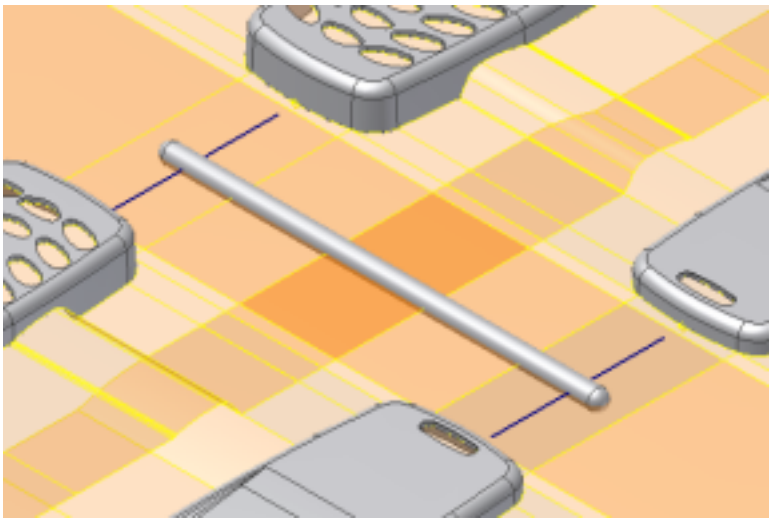
8 Click **Runner** .

9 From the Section Type menu, select **Circle**.

10 For the size of the circular runner, enter the following:
   - Diameter = **6 mm**

11 Select the two lengths of the runner sketch that make up the 120 mm run near the ends that are closest to the two subrunners. Selecting in these locations ensures that the cold slugs are located in the correct positions.

   NOTE If the cold slugs do not appear in the correct locations or fail, change the start and end position options. Click in the Cold Slug Position cell and reversing the start and end options.



12 Click **Apply.**

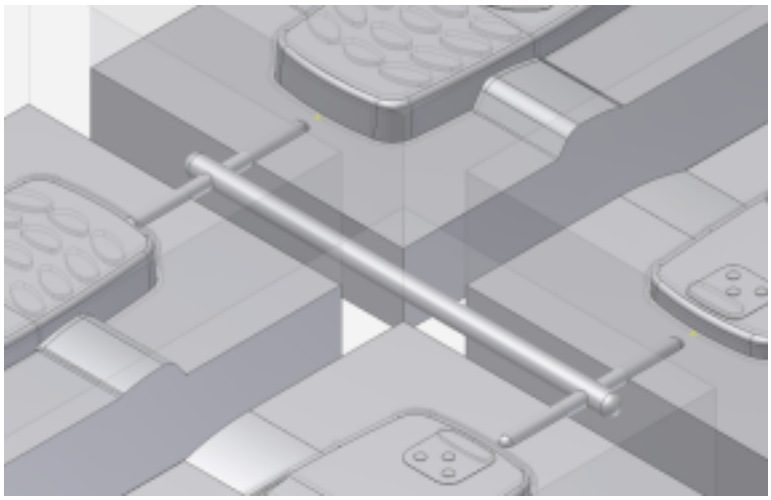
   The runner appears in the model, as shown in the previous image.

   Now that the main runner has been defined, you can define subrunners. The subrunners have a smaller diameter. For the size of the circular runner, enter the following:
   - Diameter = **4 mm**

13 On one of the four subrunner sketch lines, select on the line at the end closest to the moldable part. Select points on the three remaining subrunner sketch lines in similar locations. It is important that you select near the moldable part so that the start and end locations for the cold slug position are defined properly.

14 Click in the Cold Slug Length cells of the **Create Runner** dialog box and enter the following:
   - Segment 1 = **0 mm**
   - Segment 2 = **0 mm**
   - Segment 3 = **0 mm**
   - Segment 4 = **0 mm**

15 Click **OK** to complete the runner. The model appears as shown in the following image.

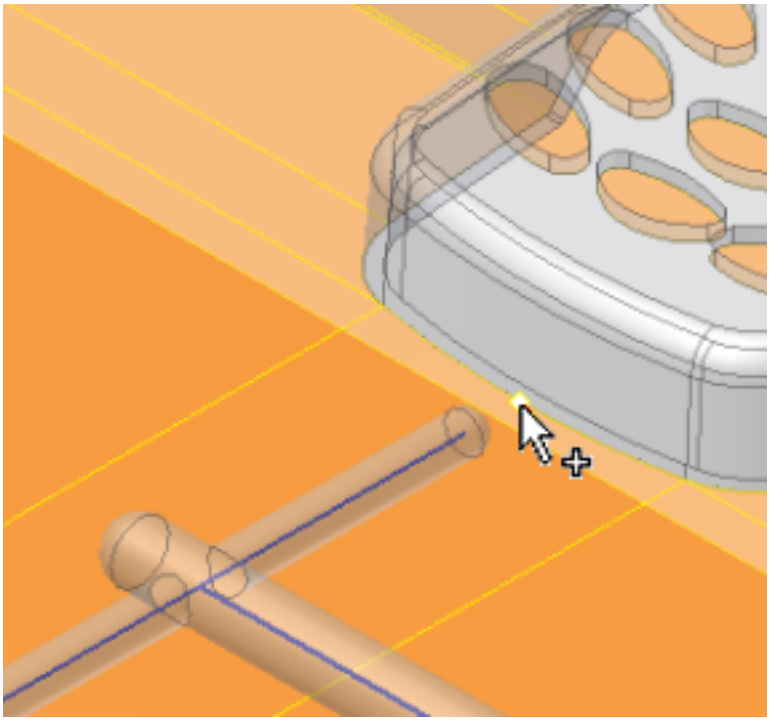


Previous (page 65) | Next (page 73)

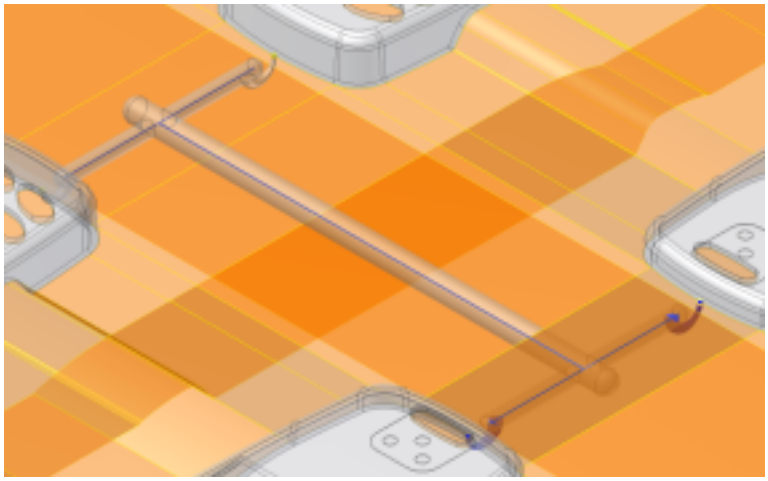
Adding Gates

In this topic, you add a gate.

1 On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Gate** .
2 In the Type menu, select **Tunnel** as the type of gate.
3 Maintain the placement option as **One Point**.
4 Select the gate location shown in the following image.



5 In the Placement section, enter the following:
   - Rotation = **270 deg**
6 Maintain the gate on the down side.
7 Enable the **Copy to all pockets** option to duplicate the gate on the patterned element.
8 Click **Apply**.
9 Select the gate location on the Mobile-Bottom part.
10 The previous settings are maintained in the **Create Gate** dialog box. The gates are previewed as shown in the following image.



11 Click **OK**.

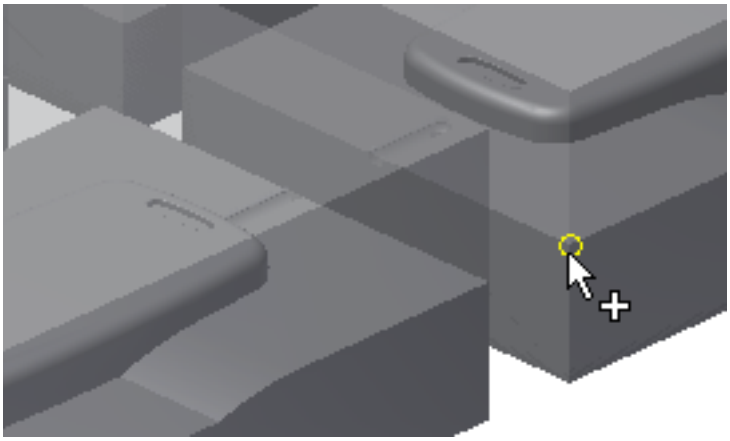
Previous (page 69) | Next (page 75)

Creating the Mold Base

In this topic, you create a mold base by using one of the standard mold bases that is supplied.

1 On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Mold Base** .

2 Select the reference point shown in the following image.

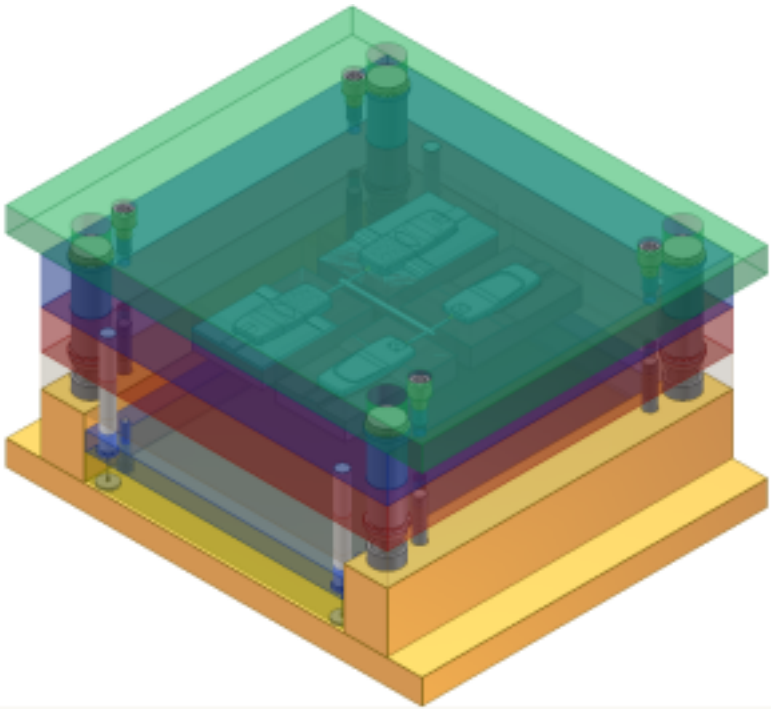


3 In the **Mold Base** dialog box, select the **DME** Vendor and the **E** Type.

4 Select the **496 mm x 496 mm** mold base size.

5 Click **OK** to generate the mold base.

6 Click OK on the **File Naming** dialog box to accept the defaults. The mold base appears as shown in the following image.



7 The runner and gates must be updated. On the Quick Access toolbar, click **Mold Update** to update the mold assembly.

8 **Save** the assembly.

Previous (page 73) | Next (page 77)

Summary

In this tutorial, you discovered how to use family mold functionality to create a mold design for multiple plastic parts in one mold base. You patterned the components that are placed into the mold assembly.

You placed two separate components and created a patterned family by using the rectangular pattern type. Circular patterns can also be generated.

You learned how to:

- Use Inventor Mold Design to create a family mold consisting of multiple moldable parts.
- Adjust the orientation and position of components placed in the mold environment.
- Assign material and shrinkage values.
- Generate the core and cavity for moldable parts.
- Use patterning functionality.
- Create the runner system, the gates, and the mold base for the mold assembly.

What next?

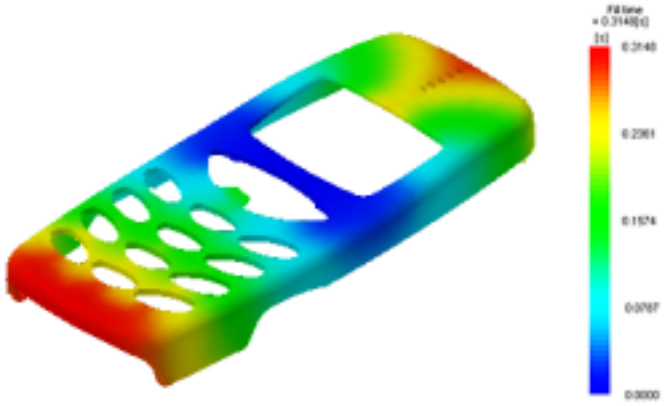
You can now add the mold base and place an ejector, sprue bushing, and locating ring.

Previous (page 75)

Core and Cavity Considerations

3

About this tutorial



Create and analyze a single part core and cavity.

Skill level	**Level 3**
Time required	75 minutes

NOTE This time varies depending on the computing power of the system running the mold analyses.

Prerequisites	■ Know how to set the active project and navigate the model space with the various view tools. See the Inventor Help topic *Getting Started* for further information. ■ Completion of the About this tutorial (page 1) Tutorial.
Tutorial file used	MobileTopComplete.ipt, Mold.ipj

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Navigation

Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Tutorial objectives

In this tutorial, you create a mold assembly of a single mold model. Unlike the other tutorials in the mold suite, you set up Inventor Mold Design to analyze the model and make recommendations for the design. After completing this tutorial, you will be able to do the following:

- Perform a simulation to suggest gate locations in the model.
- Manually modify suggested gate locations.
- Run a simulation to provide suggested process settings for the model.
- Run a Part Fill Analysis and review the graphical results.
- Use the Examine Results command to review specific areas of the model.
- Use the Part Shrinkage command to predict the amount of material shrinkage.
- Use the Parting Diagnostics command to evaluate the parting surfaces on the core and cavity.

Next (page 81)

Getting Started

In this section, you create a mold design assembly and add the plastic part to mold. You then apply a material to the model.

For this tutorial, the orientation and position of the model do not need to be modified.

1. Click **Inventor**  and select **Manage** > **Projects**.
2. Set the project to **Mold.ipj**.
3. Ensure that the Mold libraries are attached to this project file. In the **Projects** dialog box, click **Configure Content Center Libraries**  .
4. In the **Configure Libraries** dialog box, scroll to the Inventor Mold libraries and ensure that the **In Use** check box is enabled. If the Mold Metric library is not enabled, you cannot add the mold base later in the tutorial.
5. Click **OK** on the **Configure Libraries** dialog box.
6. Click **Done** on the **Projects** dialog box.
7. Click **Inventor**  .
8. Select **New** to open the **New File** dialog box and start a new mold assembly.
9. From the **Metric** tab of the **New File** dialog box, click **Mold Design (mm).iam**, and then click **OK**.
10. In the **Create Mold Design** dialog box, click **OK** to accept the defaults. The Mold Design environment is now available.

    NOTE It is important to choose a descriptive name that easily identifies the mold file. In this tutorial, the default file name is used.

11. On the browser, right-click the mold icon and make sure **Prompt for file names** is enabled.

12 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Plastic Part** .

13 In the **Plastic Part** dialog box, select the **MobileTopComplete.ipt** part file, and click **Open**. This model contains the top section of a mobile phone.

14 Left-click in the graphics window to place the part with the default alignment option.

15 Click **OK** to accept the defaults on the **File Naming** dialog box.

16 On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Select Material** .

17 Assign the following material to the model.
- Manufacturer: **Generic Shrinkage Characterised Material**
- Trade name: **Generic PC+ABS (Cycoloy)**

18 Click **OK** to close the dialog box.

Previous (page 79) | Next (page 82)

Defining the Gate Location

In this section, you define the location of the gate.

Gate locations are coordinates on the surface of a moldable part, which indicate where the physical gates will be placed. Up to ten gate locations can exist on

a part, and they can be established manually or the Inventor Mold Design application can suggest them. You run a Gate Location simulation, which is used to recommend injection locations for the part, based on minimizing the flow resistance.

1. On the ribbon, click **Mold Layout tab** > **Runners and Channels panel** > **Gate Location**  .
2. In the **Gate Location** dialog box, select the **Suggest** tab.
3. In the **Number of Gate Locations [1:10]** field, maintain the default **1 ul** value to suggest a single gate.
4. Click **Start** on the **Gate Location** dialog box.
5. Read the **Analysis running** dialog box and click **OK**. You can stop the simulation at any time by clicking **Stop** on the **Gate Location** dialog box.
6. While the simulation is still running, on the ribbon, click **Mold Layout tab** > **Tools panel** > **Job Manager**  .

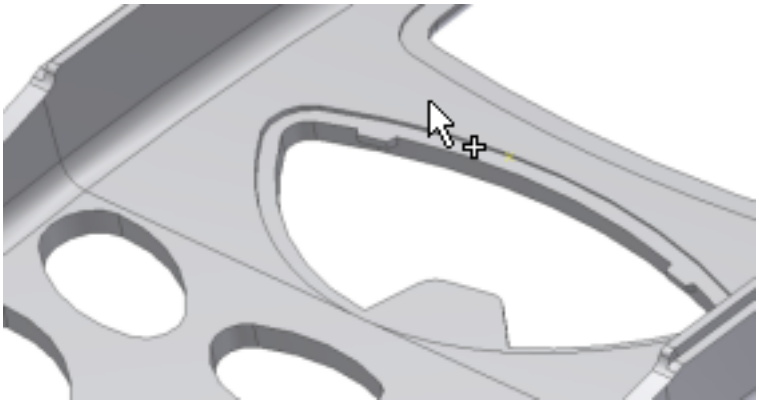
   NOTE When you first open it, the **Job Manager** dialog box indicates that no simulations are running. It takes time for the list to update and the Gate Location simulation to appear.

7. The **Job Manager** dialog box enables you to monitor the progress and stop it, if necessary. You can close the **Job Manager** dialog box without stopping the simulation, but for now, keep this dialog box open until the simulation has finished.
8. The **Summary** dialog box appears when the simulation has finished. Select the **Gate Location** tab, which contains the recommended location of the gate.
9. Click **OK** on the **Summary** dialog box.
10. Click **OK** on the File Naming dialog box to accept the defaults.
11. Inventor Mold Design generates the Gate Location, which is based on the simulation results, and closes the dialog box.
12. On the Mold Design browser, expand the **MobileTopComplete** > **Gate Locations** nodes. Right-click **Gate Location 1** and click **Edit Feature**.
13. Rotate the model and zoom into the point. Notice the location of the system-generated gate location.

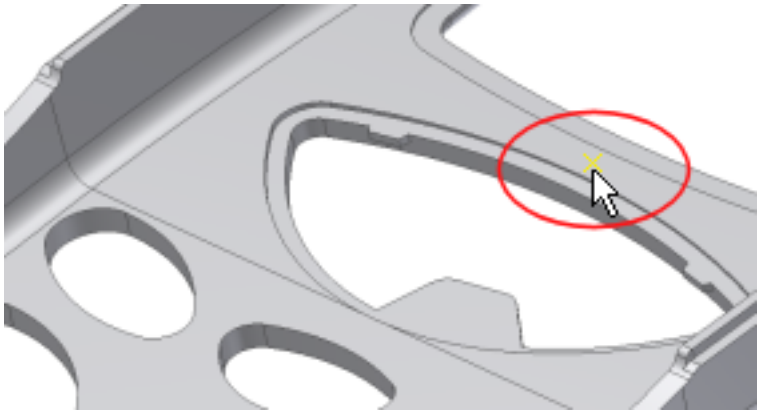
The simulation result is only a suggestion for the optimal region. The final gate location is a compromise based on part use, appearance, and tooling restrictions. In this situation, the suggested gate location is on a thin feature. It is not ideal because it makes it difficult to pack out the part. You now modify the reference surface and values to reposition the gate point. The new gate point is close to the suggested location, and is on the nominal wall of the part and on the underside of the model. Placing the gate location on the top of the model would be unsuitable because it could leave a witness mark.

NOTE An important part of the injection molding process is called "packing" or the "packing phase". Packing refers to applying pressure to force additional material into the part. Packing is done to reduce the amount of volumetric shrinkage in the part. If thermoplastic materials are allowed to shrink without packing, the polymer can shrink 10% to 25% non-uniformly. Gating the part in a thin area causes the thin area to freeze before the thicker areas are packed out enough. It causes the thicker area to have higher shrinkage. Higher shrinkage can cause surface defects and increased warpage.

1. Click **Location** on the **Gate Location** dialog box, and select the surface shown in the following image.



2. Change the gate location by modifying the following parameters:
   - U = **0.5**
   - V = **0.525**
3. Click **OK** on the **Gate Location** dialog box. The gate location updates, as shown in the following image.



When using the simulation tools in Inventor Mold Design, an AIT directory is created in Windows Explorer. This directory is located in the mold design directory in which you are working, and contains the simulation data and results files. The AIT directory is generated when running simulations. These files are considered as part of your design, and will be properly handled by system utilities like Pack & Go and Vault.

Previous (page 81) | Next (page 85)

Defining Part Process Settings

In this section, you define the part process settings.

The Part Process Settings command determines the optimum mold temperature, melt temperature, and injection time based on the selected material and part geometry. Optimum processing conditions help to ensure that the part fills without defects. Before running the Part Process Settings command, a material must be defined and at least one gate location must exist.

1. On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Core/Cavity** 
2. On the ribbon, click **Core/Cavity tab** > **Plastic Part panel** > **Part Process Settings**  .

3 From the **Part process settings** dialog box, maintain the default settings and select the **Suggest** tab.

By default, the simulation is run with values for material properties, maximum injection pressure limit, machine injection time, and machine clamp open time. They are derived from the Autodesk Moldflow material database. Alternatively, you can override these values and choose your own.

1 Specify **Low gloss** as the required surface finish.
2 Click **Start** on the **Part process settings** dialog box.
3 Click **OK** on the **Analysis running** dialog box.
4 Notice the results for the Mold temperature, Melt temperature, and Injection time. The results obtained from this simulation will be used to perform the part fill simulation in the next topic.
5 Click **OK** on the **Summary** dialog box to close the dialog box.
6 On the ribbon, click **Core/Cavity tab** > **Plastic Part panel** > **Part Process Settings**  .
7 Notice the values on the **Set** tab have been updated to the values calculated by the simulation.
8 Click **Cancel** on the **Part process settings** dialog box to close the dialog box without making any further changes.

Previous (page 82) | Next (page 86)

Performing a Part Fill Analysis

In this section, you perform a part fill simulation, which predicts the flow of plastic melt inside the mold during the filling process.

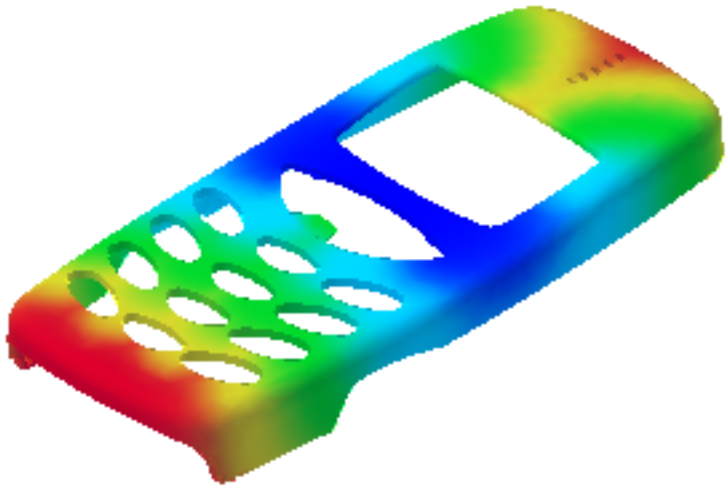
To do it, the flow front is calculated as it grows through the part incrementally from the injection location. The simulation runs until the velocity pressure switch-over point has been reached. There are two types of fill simulation: a Part Fill analysis evaluates individual parts, and a Mold Fill analysis evaluates the entire mold design, including the feed system. Before running a Part Fill

analysis, the material and the part process settings must be defined, and at least one gate location must exist.

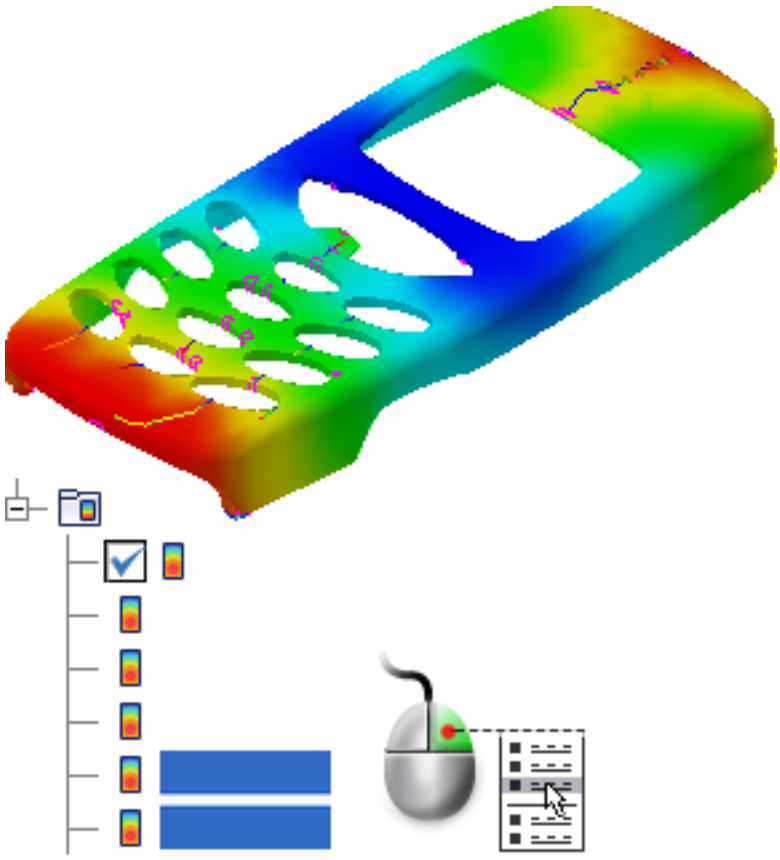
1. On the ribbon, click **Core/Cavity tab** > **Plastic Part panel** > **Part Fill Analysis**  .
2. Click **Start** on the **Part Fill Analysis** dialog box.
3. Click **OK** on the **Analysis running** dialog box.
4. Review the **Summary** dialog box.

The top of the Summary dialog box indicates that the part can be filled easily but part quality may be unacceptable. This information indicates to review the Quality prediction plot, determine the magnitude of the quality problems, and check whether the locations of these problems are critical to part performance. Additional fill analyses can be displayed to determine whether changes in variables, such as the process settings, gate location, and material, improve the results.

1. Click **Cancel** to close the **Summary** dialog box.
2. Right-click in the model window and click **Home View** to return the model to its default orientation.
3. On the Mold Design browser, expand the **MobileTopComplete** > **Results** > **Fill** nodes. Six results were generated: Fill time, Plastic flow, Confidence of fill, Quality prediction, Air traps, and Weld lines.
4. Double-click the **Fill time** node. The model displays as shown in the following image.



5 Right-click the **Air traps** and **Weld lines** nodes in the browser and enable **Overlay** in the context menu. The fill time display now includes the air traps and weld lines as shown in the following image.



6 You can enable each of these nodes individually or together to help visualize the model. Right-click the **Air traps** and **Weld lines** nodes in the browser and clear the **Overlay** selection.

7 Use the color contour legend to review the color display on the model, which shows the fill time for each area.

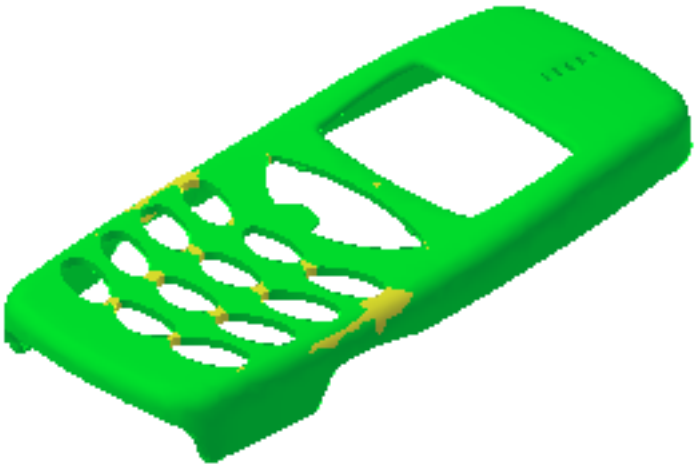
The Fill time result shows the position of the flow front at regular intervals as the cavity fills. The color contour represents the parts of the mold that are being filled simultaneously. At the start of the injection, the flow front is represented in dark blue. The last places to fill are represented in red. An absence of color indicates sections that did not fill, that is, the part has a short shot.

The spacing between the contour lines, or width of the color bands, is an indication of the flow velocity. Because the time duration between each line or band is the same, the further apart the lines or bands are, the faster the flow front is moving. A part with a good fill time result has a balanced flow pattern. The following also applies:

- All the flow paths finish at the same time. The flow fronts should reach the extremities of the model simultaneously and each flow path should end with red contours. If one flow path finishes before the others, it may indicate overpacking.
- The contours are evenly spaced. The spacing indicates the speed at which the plastic melt is flowing. Widely spaced contours indicate rapid flow. Narrow contours indicate the part is filling slowly. Closely spaced contours indicate hesitation.

In this tutorial, the central location of the gate provides a nearly balanced fill pattern. The flow front reaches the top of the part slightly before the bottom. The flow velocity is high near the gate, and relatively high on the bottom of the part (below the keypad). These regions have high velocities because the flow front is not large compared to the rest of the part. Areas of high velocity tend to have higher shear stress.

1. In the Mold Design browser, double-click the **Quality prediction** node. The model displays as shown in the following image.



The quality prediction result estimates the expected quality of the appearance of the part, and its mechanical properties. The quality predictions that are displayed are based on the following sliding scale:

- Green indicates high quality.
- Yellow indicates that there may be quality issues.
- Red indicates that there are quality issues.

Most of the display for this model is green, indicating high quality. The model is yellow in some areas, indicating there may be some quality issues that need further investigation. None of the areas that indicate quality issues are large.

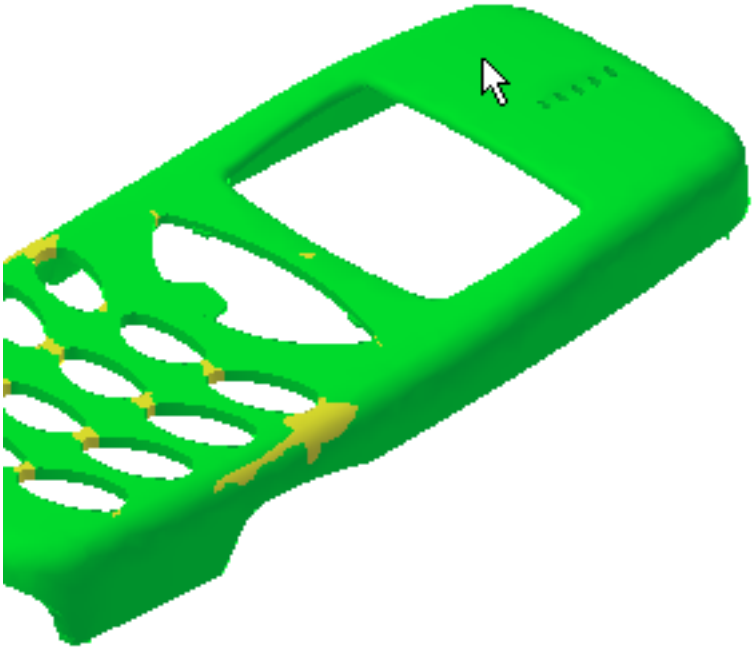
Previous (page 85) | Next (page 91)

Examining the Results

In this section, you examine the Fill results that you generated in the last section.

The Examine Results command enables you to query the value of a result plot at a specific point and, where appropriate, determine the underlying, contributory results. Using this command, you can obtain advice on how to interpret the results that are being displayed. You also get advice about how to fix any molding problems that are revealed in a specific location.

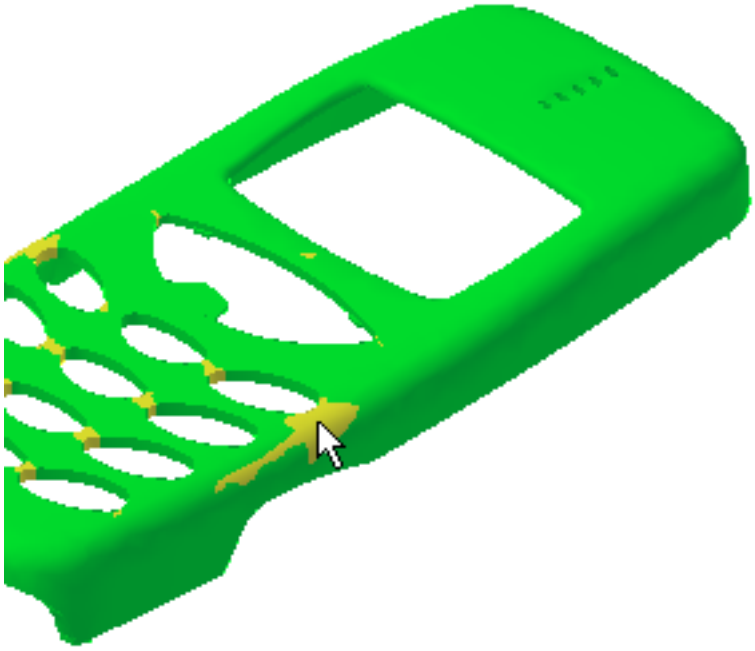
1. On the ribbon, click **Core/Cavity tab** > **Tools panel** > **Examine Results**  .
2. From the graphics window, select the green area shown in the following image.



3 The results for this section of the model update and are displayed in the Results adviser dialog box. Notice that the Confidence of fill in this area is high. This means that the selected area has a high probability of filling with the plastic melt.

4 Select the **Quality prediction** tab and review the results. Notice that the Quality prediction is also high.

5 Select a yellow area in the graphics window, as shown in the following image.



6 Review the results for this section of the part. The Confidence of fill is high but the Quality prediction is medium; and a note in the Quality prediction tab indicates that the shear stress exceeds the recommended limit.

7 Select **Click here to view additional help**.

The Mold Design Help window opens to the Medium Quality Prediction page. Clicking this link from inside the Results adviser dialog box automatically opens the help documentation to the type of prediction you obtained, and explains the result. Suggestions on how to resolve the situation are also provided. In this case, the message informs you that the shear stress exceeds the recommended limit. Clicking the link for that issue provides further explanation of the result, the possible causes of this result, and suggestions on how to resolve the problem.

1 Close the Mold Design Help window.

2 Click **Done** on the **Results adviser** dialog box.

3 On the Mold Design browser, double-click the **Quality prediction** node to clear the result display from the model. Compress the display of the Results subnodes.

Previous (page 86) | Next (page 94)

Defining Part Shrinkage

In this section, you use the Part Shrinkage command to estimate the percentage of shrinkage that occur based on the mold dimensions. Shrinkage is inherent in the injection molding process and occurs as the plastic melt in the part cools.

If necessary, adjustments can then be made to the mold dimensions until the part dimensions are within design specifications. Before running a shrinkage simulation, a material must be defined, and at least one gate location must exist.

1 On the ribbon, click **Core/Cavity tab** > **Plastic Part panel** > **Part Shrinkage**  .

2 From the **Part Shrinkage** dialog box, select the **Suggest** tab and click **Start**.

3 Click **OK** on the **Analysis running** dialog box.

4 Review and take note of the results. The **Results** dialog box provides the following information:

- Nominal Dimensions: The original dimensions of the product.
- Shrinkage Allowance: Set to 0% by default, but you can define it.
- Mold Cavity Dimensions: Calculated by adding shrinkage allowance to nominal part dimensions.
- Estimated Shrinkage: The result of a shrinkage simulation based on mold cavity dimensions.
- Estimated Dimensions: Calculated by subtracting the estimated shrinkage from the mold cavity dimensions.

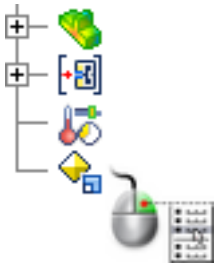
If the Estimated dimensions are close in value to the Nominal dimensions, the mold cavity dimensions do not need to be modified. If the Estimated and Nominal dimensions are not close in value, the estimated shrinkage values can be used to repopulate the shrinkage allowance. Or, the values can be manually adjusted before repeating the

simulation. In this tutorial, the computed values are not acceptable. The simulation must be rerun, using the estimated shrinkage values as the shrinkage allowance.

5 Click **OK** on the **Summary** dialog box.

6 Click **Yes** in the **Edit shrinkage** dialog box to replace the current values. A Shrinkage node is created in the Mold Design browser.

7 On the Mold Design browser, right-click the **Shrinkage** node shown in the following image and click **Edit Feature**.



8 Notice the Shrink Percentage values are equal to the values obtained from the shrinkage simulation.

The range of shrinkage values is only an indication of typical values in the database and is not necessarily an acceptable value. Because they are based on processing conditions and part geometry, the simulation values could be slightly above or below these values.

1 From the **Part Shrinkage** dialog box, select the **Suggest** tab, and click **Start** to rerun the simulation using the new shrinkage allowance values.

2 Click **Yes** on the Analysis results exist dialog box.

3 Click **OK** on the **Analysis running** dialog box.

4 Review the results of the simulation. Notice that this time the Nominal Part Dimensions and the Estimated Dimensions are close. The mold cavity dimensions are now acceptable.

5 Click **OK** on the **Summary** dialog box.

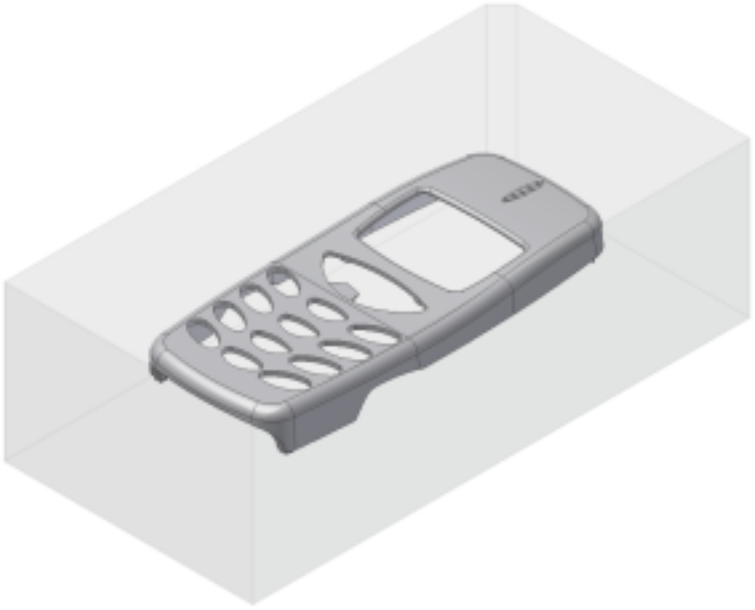
6 Click **Yes** on the **Part Shrinkage** dialog box to replace the part shrinkage values.

Previous (page 91) | Next (page 96)

Defining the Workpiece

In this section, you define the workpiece that to use to create the core and cavity for the mold.

1. On the ribbon, click **Core/Cavity tab** > **Plastic Part panel** > **Define Workpiece**  .
2. Maintain **Rectangular** as the Workpiece Type and enter the following Workpiece Dimensions:
   - X_total: **80 mm**
   - Y_total: **160 mm**
   - Z_total: **50 mm**
3. Click **More**  to expand the **Define Workpiece Setting** dialog box . Notice that you can design your rectangular workpiece with a Foolproof feature that enables you to add chamfers and fillets.
4. Click **Chamfer**  on the **Define Workpiece Setting** dialog box, and select the check box next to the top left-hand corner of the workpiece. Enter a value of **5 mm**.
5. Click **OK** on the **Define Workpiece Setting** dialog box.
6. Click **OK** on the File Naming dialog box to accept the defaults.
7. The model appears as shown in the following image.

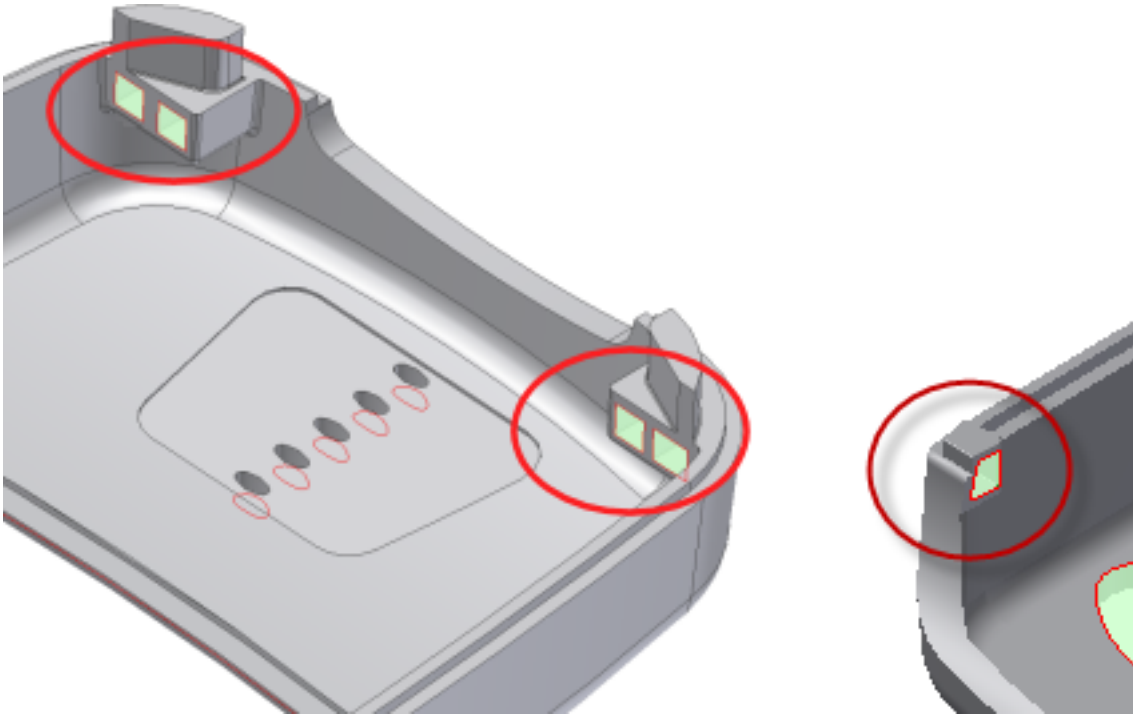


Previous (page 94) | Next (page 97)

Defining the Patching and Runoff Surface

In this section, you define the patching and runoff surfaces for the mold assembly.

1. On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Create Patching Surface**  .
2. Click Auto Detect to start the calculation. Orient the model and then locate the circled patches, on each end of the model as shown in the following image. These patches must be created by using inserts so these patching surfaces must be deleted.



It is important to keep in mind that the Inventor Mold Design patching algorithm is a starting point for patching surfaces. Review the patching surfaces that the system detected. Add or delete patches depending on your requirements and what was automatically created for you.

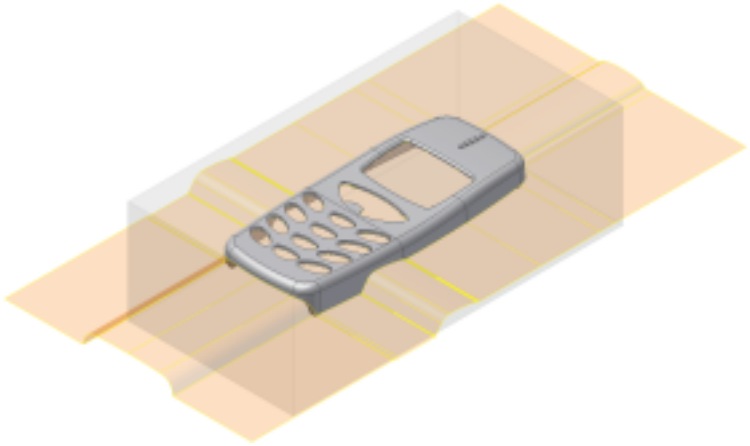
1. Determine the name of each patch to delete by placing your mouse pointer on each of the patches in the graphics window. Note the names that appear.
2. In the Create Patching Surface dialog box, scroll through the list of patches and locate any one of the patches identified in the previous step. Select this patch and in the graphics window verify that the required patch is highlighted. Press **Delete** on your keyboard to remove the patch.
3. Delete the other surface patches shown in the image. Ensure there are no patches on any of the four square cutouts, and that you have not deleted any of the other surface patches in the model.

4 To achieve higher-quality surfaces in the final product, click **All Lower** in the **Create Patching Surface** dialog box to switch the patching surfaces to the lower edges of the model.

5 Click **OK** on the **Create Patching Surface** dialog box to generate the patching surfaces.

6 On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Create Runoff Surface** .

7 Click Auto Detect to start the calculation. Click **OK** on the **Create Runoff Surface** dialog box to generate the surface shown in the following image.



Previous (page 96) | Next (page 99)

Running the Parting Diagnostic and Generating the Core and Cavity

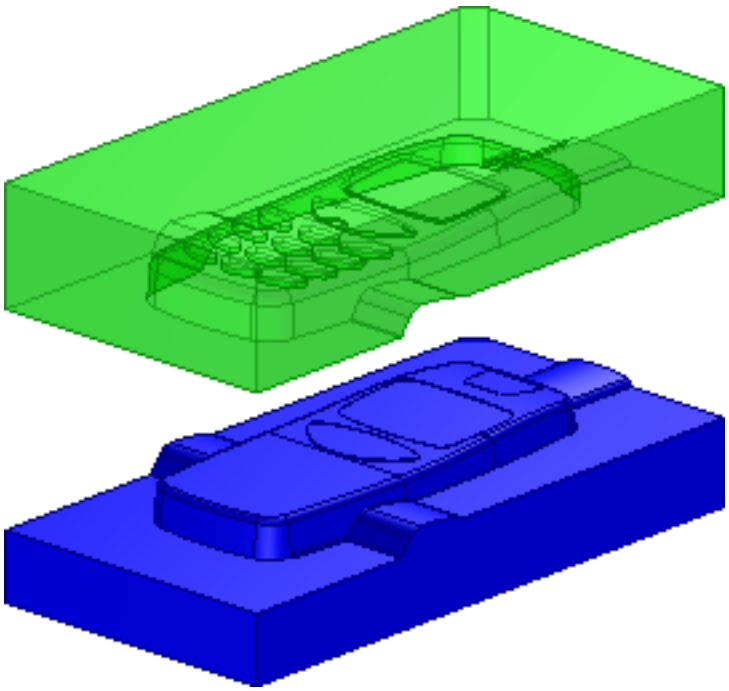
In this section, you use Parting Diagnostics before creating the core and cavity to verify that the parting surface has been completed successfully. You then generate the core and cavity for the model.

If there is a problem with the parting surface, such as a leak due to a hole or breach, the problems list. You can highlight the potential problem areas in the graphics window by selecting them in the list.

1. On the ribbon, click **Core/Cavity tab** > **Parting Design panel** > **Generate Core and Cavity**  .

On the dialog box, choose **Preview/Diagnose** to start the parting analysis. Click the **Parting Diagnostics** tab. The **Problems list** in the dialog box indicates that there are some undercut faces in the model. It indicates to use Inserts for these faces to ensure that the product can be ejected smoothly. You learn how to create Inserts in the Mold Structural Assembly tutorial. For now, continue with this topic.

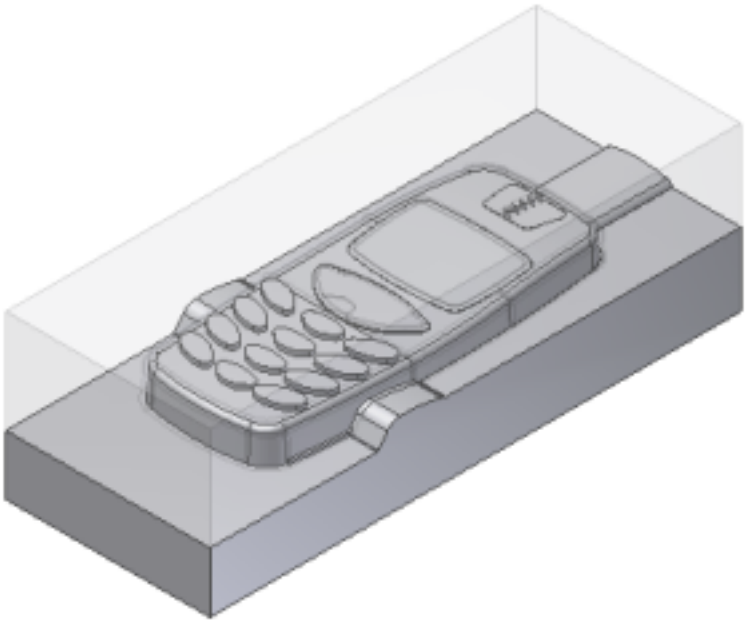
1. Click the **Preview** tab.
2. Rotate the model. Notice that the core and cavity bodies are blue and green.
   - Green indicates the cavity body which has a positive draft angle in the pull direction.
   - Blue indicates the core body which has a negative draft angle in the pull direction.
3. Drag the **Body Separation** slider all the way to the right. The core and cavity bodies separate, as shown in the following image. The Surface Separation slider shows you what the core and cavity bodies will look like. It helps you detect potential errors in the parting design before the actual parts are generated. In this design, the core and cavity bodies are satisfactory.



4 Click **OK** to generate the core and cavity.

5 Click **OK** to accept the defaults on the **File Naming** dialog box.

6 The system removes the driving moldable part from display and the core and cavity set are shown. The core body is opaque and the cavity body is translucent.



7 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 97) | Next (page 102)

Summary

In this tutorial, you learned about the simulations that can be performed in Inventor Mold Design. These simulations are designed to provide you with suggestions on how you can design your mold assembly. It is recommended that you adapt these suggestions to suit the specific usage, appearance, and tooling restrictions of the plastic part.

You learned how to:

- Perform a simulation to suggest gate locations in the model.
- Manually modify suggested gate locations.
- Run a simulation to provide suggested process settings for the model.
- Run a Part Fill Analysis and review the graphical results.
- Use the Examine Results command to review specific areas of the model.

- Use the Part Shrinkage command to predict the amount of material shrinkage.
- Use the Parting Diagnostics command to evaluate the parting surfaces on the core and cavity.

What next?

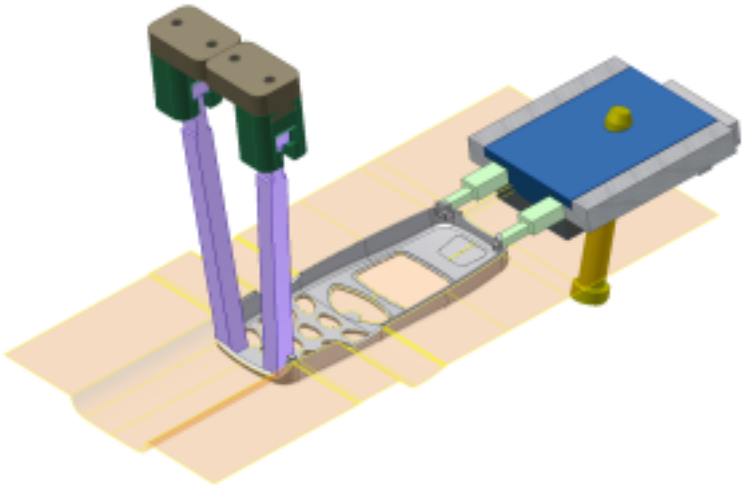
Learn how to incorporate inserts for sliders and lifters in the Mold Structural Assembly tutorial. You can then create gates, runners, and cooling channel components, and place the mold base.

Previous (page 99)

Mold Structural Assembly

4

About this tutorial



Add mold features to an existing assembly.

Skill level	**Level 3**
Time required	60 minutes
Prerequisites	■ Know how to set the active project and navigate the model space with the vari-

	ous view tools. See the Inventor Help section *Getting Started* for further information. ■ Completion of the About this tutorial (page 1) Tutorial. ■ Completion of the *About this tutorial (page 79)* Tutorial.
Tutorial file used	Mold Structural Assembly directory

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Navigation

Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Tutorial objectives

In this tutorial, you open an existing assembly and add additional mold features to complete the design. You incorporate inserts, lifter and slider assemblies, and lock sets into the mold design. After completing this tutorial, you will be able to:

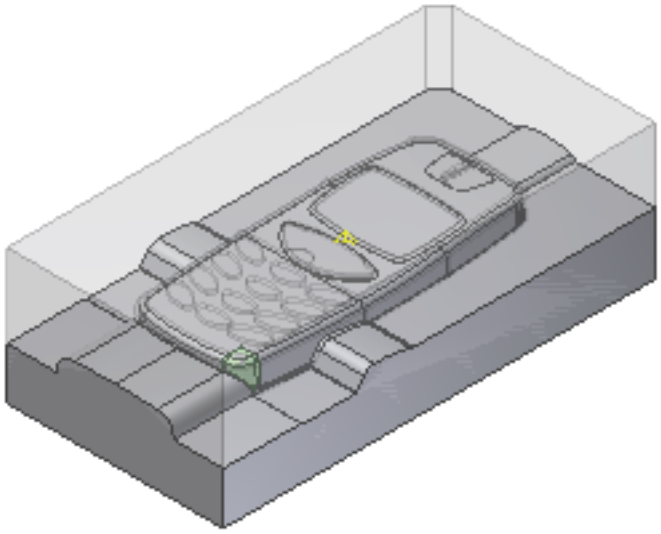
- Create and add existing inserts to a mold assembly.
- Create heels associated with existing inserts.
- Create lifter and slider assemblies.
- Perform Boolean operations on lifters and sliders.
- Place lock sets in a mold assembly.
- Generate the workpiece pocket.

Next (page 107)

Getting Started

In this section, you open an existing mold assembly. You add mold features to this mold assembly throughout the tutorial.

1. Click **Inventor**  to launch Inventor.
2. Close all open windows in Inventor .
3. On the ribbon, click **Get Started tab** > **Launch panel** > **Projects** .
4. Click **Browse.**
5. Set your active project to *MoldStructuralAssembly.ipj*
6. In the **Projects** dialog box, click **Configure Content Center Libraries** .
7. In the **Configure Libraries** dialog box, scroll to the Inventor Mold Metric library and ensure that the **In Use** check box is enabled.
8. Click **OK** on the **Configure Libraries** dialog box.
9. Click **Done** on the **Projects** dialog box.
10. On the ribbon, click **Get Started tab** > **Launch panel** > **Open** .
11. Navigate to and select **\Mold Top\Mold Top.iam**.
12. Click **Open** to open the mold assembly. The mold assembly appears as shown in the following image.



13 Right-click Mold Design in the browser and make sure **Prompt for File Names** is enabled.

Previous (page 105) | Next (page 108)

Creating an Insert for a Slider Assembly

In the next three sections, you create inserts with heels that make up a slider assembly. The inserts are used to create an undercut on the part.

Inserts are individual pieces of metal in a mold base that are used to create a portion of the part. In this case, the insert is part of a slider. Inserts are added to an assembly by creating an insert or placing an existing insert.

You can create an insert by using a template, a shape, or a sketch:

- With the template method, an insert is created from one of the following predefined types: rectangular, square, circle, key1-flat, key2-flat, slot, or rounded rectangle. In this tutorial, you learn how to create an insert by using the template method.
- With the shape method, the profile of the insert is based on selected faces of the model. Only faces on the product, core, cavity, inserts, and core pins can be selected.

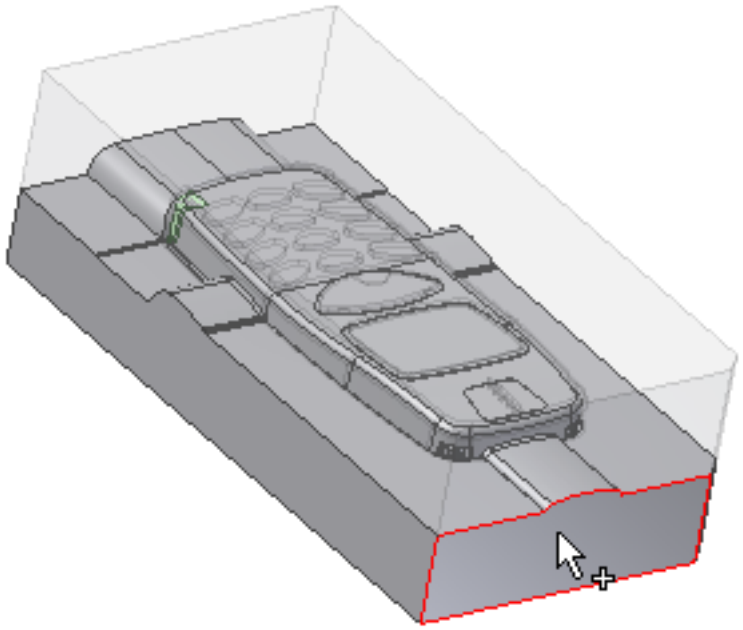
- With the sketch method, an existing sketch can be used as the profile for the insert. The sketch is created by using the Manual Sketch command, and the sketch must already exist in the model before this method can be used.

Click the following link to see how an insert is created by using the From Sketch method:

Creating an insert using the From Sketch method

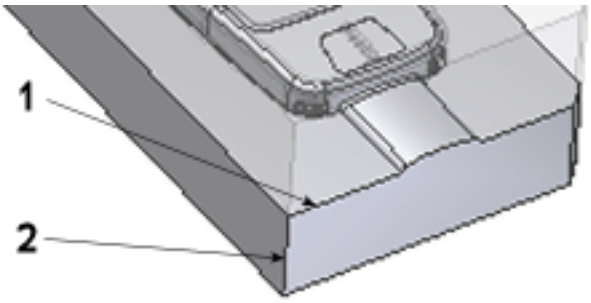
Click the following image to play an animation that shows creating an insert using the From Sketch method.

1. On the ribbon, click **Mold Layout tab** > **Mold Layout panel** > **Core/Cavity** .
2. From the **Core/Cavity** tab, click **Insert panel** > **Create Insert** .
3. From the **Create Insert** dialog box, select **Template** in the Profile type menu. Maintain **Rectangle** as the Profile shape.
4. Leave the Placement type as **Linear**. With a linear placement, you are required to select a plane and two linear references. The plane defines the surface on which the rectangular profile will be created. The references are selected edges that are used to locate the profile. To locate the profile correctly, Reference 1 and Reference 2 should be perpendicular to each other.
5. The **Plane** selection tool automatically activates. Reorient the model and select the surface shown in the following image.



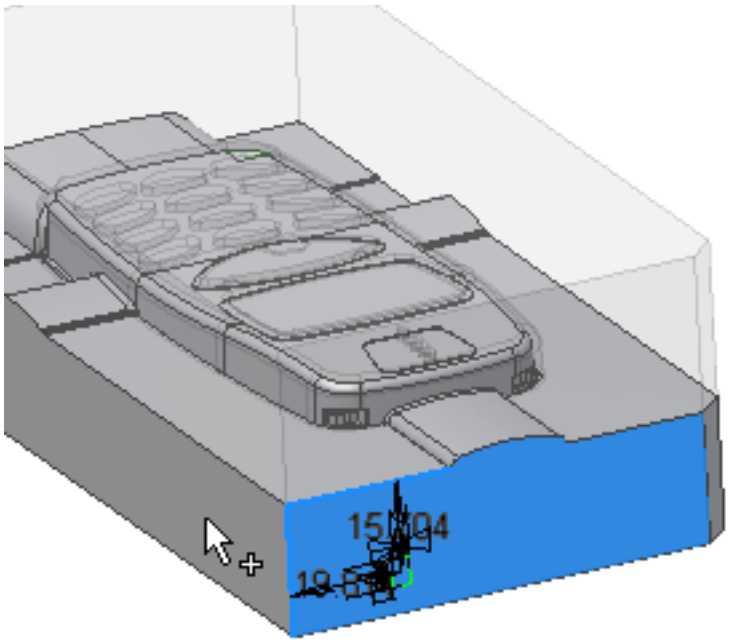
6 The system prompts you to select **Reference 1**. Select the edge labeled 1 in the following image.

7 The system prompts you to select **Reference 2**. Select the edge labeled 2 in the following image.



8 On the **Create Insert** dialog box, ensure that the **Select component**  tool is selected.

9 The **Reference**  selection tool should be active . Select the core in the graphics window as the termination reference, as shown in the following image.

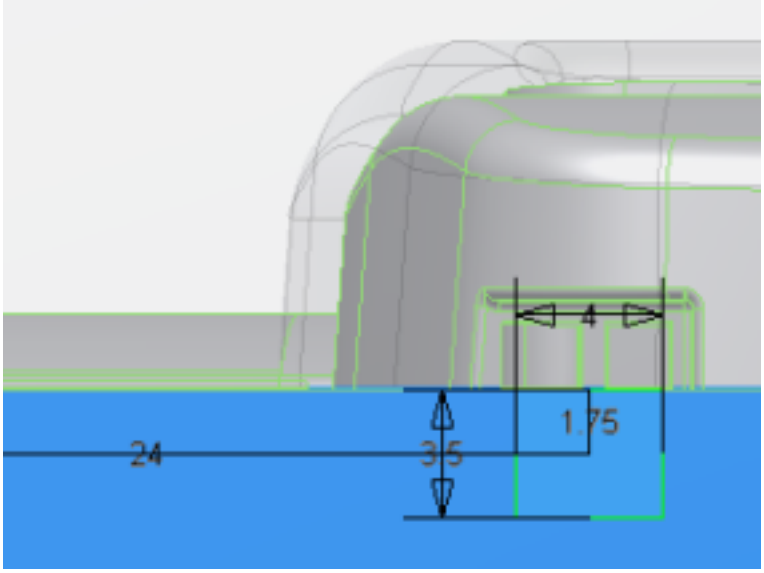


10 In the upper-right corner of the graphics window, click **Top** on the ViewCube to orient the model.

11 Modify the following dimensions:

- Distance from Reference 1 = **1.75 mm**
- Distance from Reference 2 = **24 mm**
- Width of rectangle = **4 mm**
- Height of rectangle = **3.5 mm**

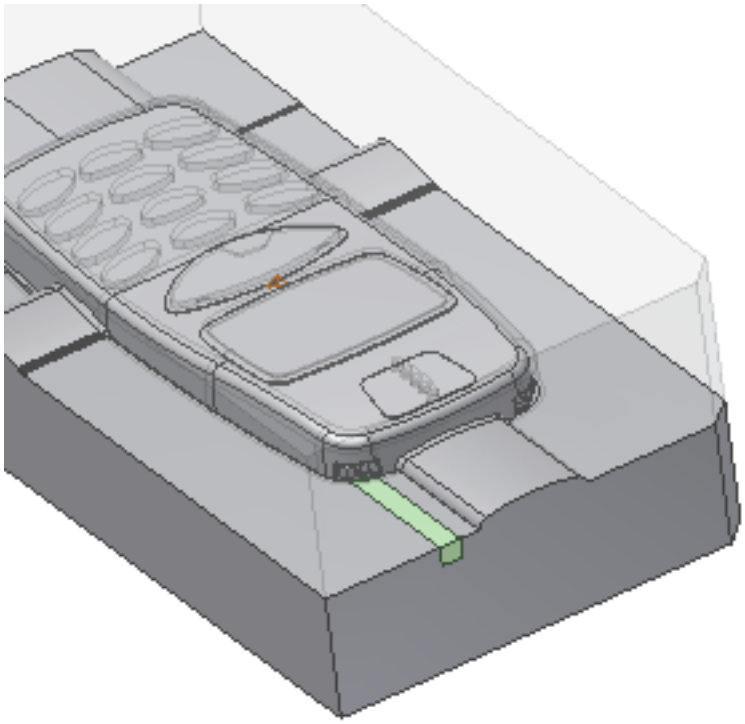
The sketch appears as shown in the following image.



12 Click **OK** on the **Create Insert** dialog box.

13 Click **OK** on the **File Naming** dialog box to accept the defaults and generate the insert.

14 Reorient the model, as shown in the following image.

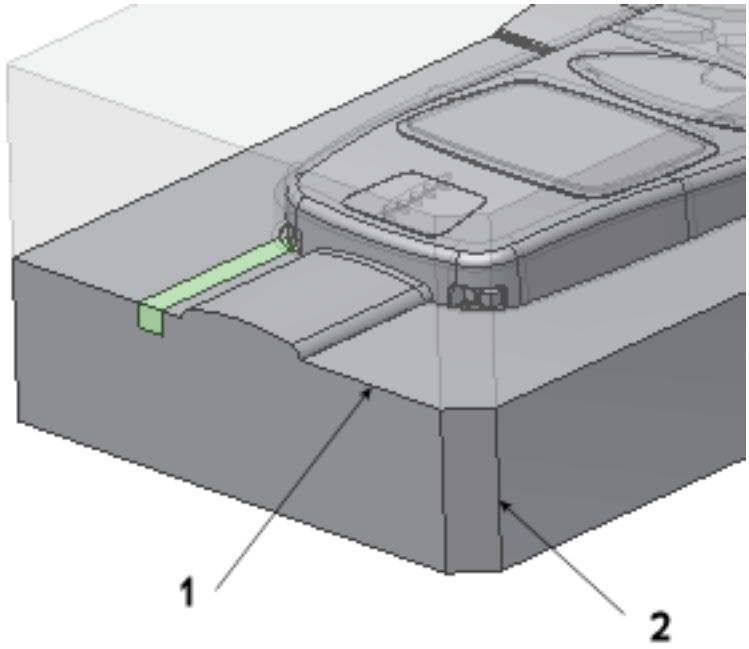


Previous (page 107) | Next (page 113)

Creating Another Insert for a Slider Assembly

In this section, you continue to create the inserts for the slider assembly.

1 On the ribbon, click **Core/Cavity tab** > **Insert panel** > **Create Insert**  . You use the Rectangle Template profile to create another insert. The plane is the same plane that you used to create the first insert, and the references are shown in the following image.



2 In the **Create Insert** dialog box, ensure that the **Select component**  tool is selected.

3 Select the core in the graphics window as the termination reference.

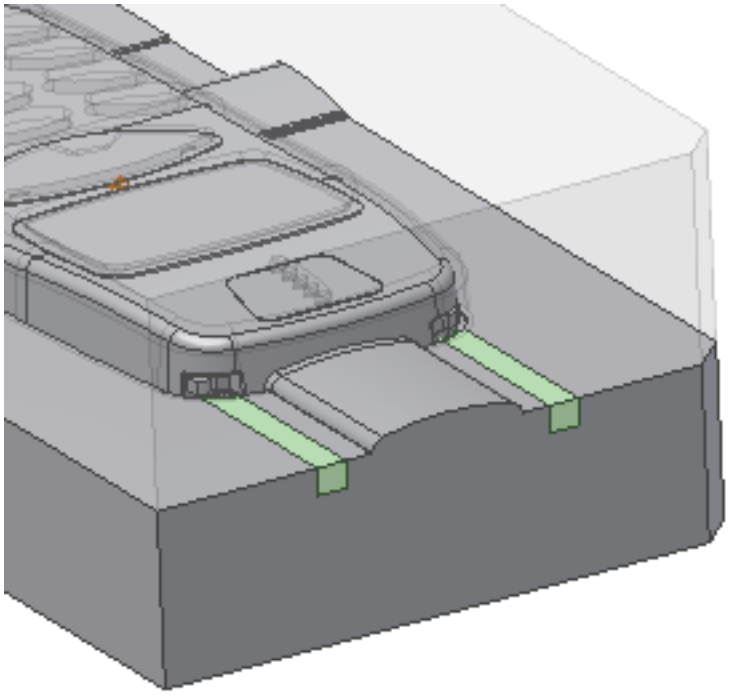
4 In the upper-right corner of the graphics window, click **Top** on the ViewCube to orient the model.

5 Modify the following dimensions:
- Distance from Reference 1 = **1.75 mm**
- Distance from Reference 2 = **24 mm**
- Width of rectangle = **4 mm**
- Height of rectangle = **3.5 mm**

6 Click **OK** on the **Create Insert** dialog box.

7 Click **OK** on the **File Naming** dialog box to accept the defaults and generate the insert.

8 Reorient the model as shown in the following image. The two inserts that have been created by using the template method are displayed.



9 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 108) | Next (page 115)

Creating Heels for Slider Inserts

In this section, you create heels as features on the inserts. Heels are created on the inserts to help lock an insert into the core or cavity.

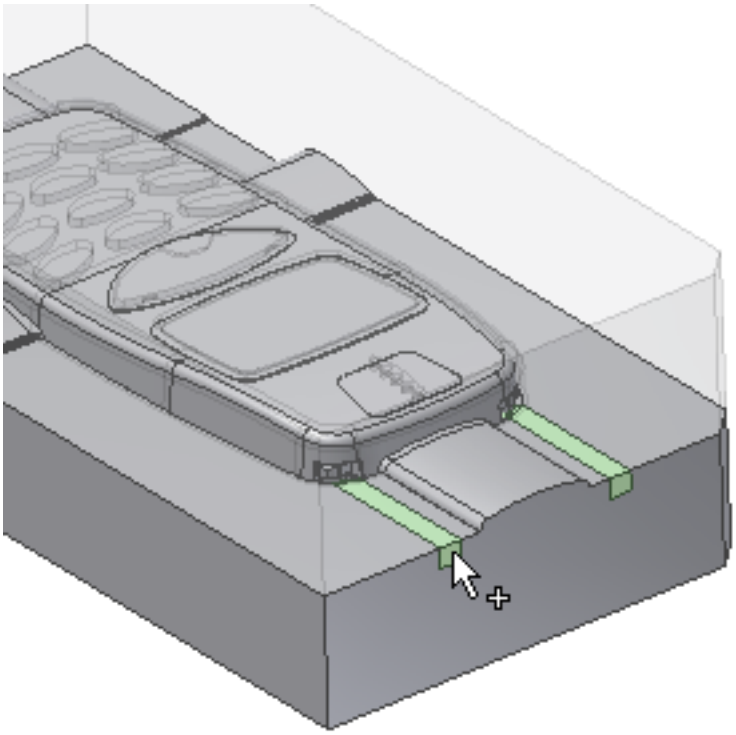
- With the template method, the heel profile is based on the following predefined types: rectangular, square, circle, key 1-flat, key 2-flats, slot, or rounded rectangle.

- With the thicken method, the heel is defined by selecting a planar face on the insert as the start plane on which to extrude the heel. An offset value for the profile of the heel is then defined.

1. On the ribbon, click **Core/Cavity tab** > **Insert panel** > **Create Heel**

   .
2. Select **Template** from the Profile type menu. Select **Rectangle** from the Profile shape menu.
3. Orient the model as shown in the following image, and select the first insert highlighted by the arrow as shown in the following image.



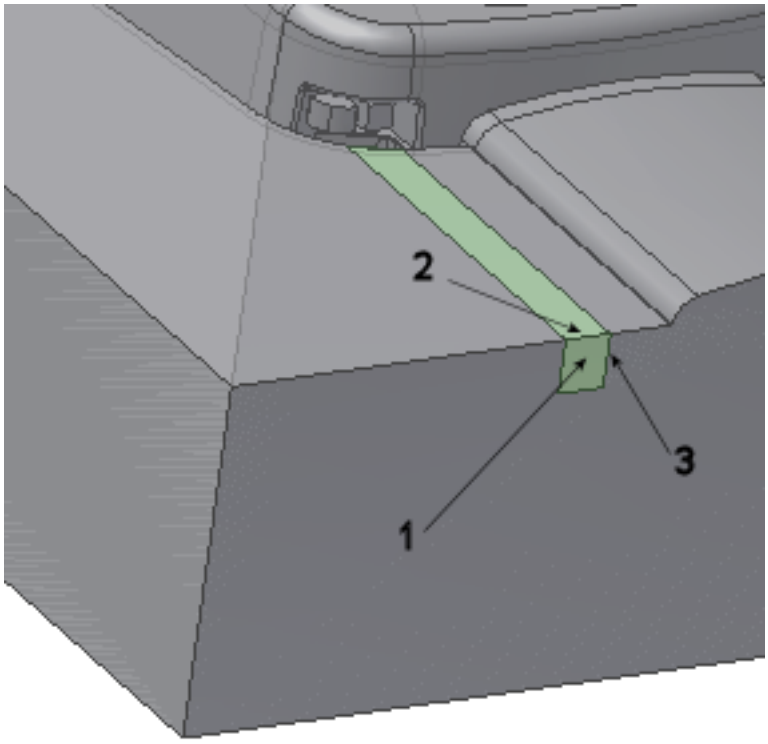
4. Leave the Placement type as **Linear**. With a linear placement, you are required to select a plane and two linear references. The plane defines the surface on the selected insert in which the rectangular profile will be created. The references are edges on the selected insert that are used

to locate the profile. To locate the profile correctly, Reference 1 and Reference 2 should be perpendicular to each other.

5 The **Face** selection tool should be active . Select the surface of the insert labeled **1** in the following image.

6 As **Reference 1**, select the edge of the insert labeled **2** in the following image.

7 As **Reference 2**, select the edge of the insert labeled **3** in the following image.

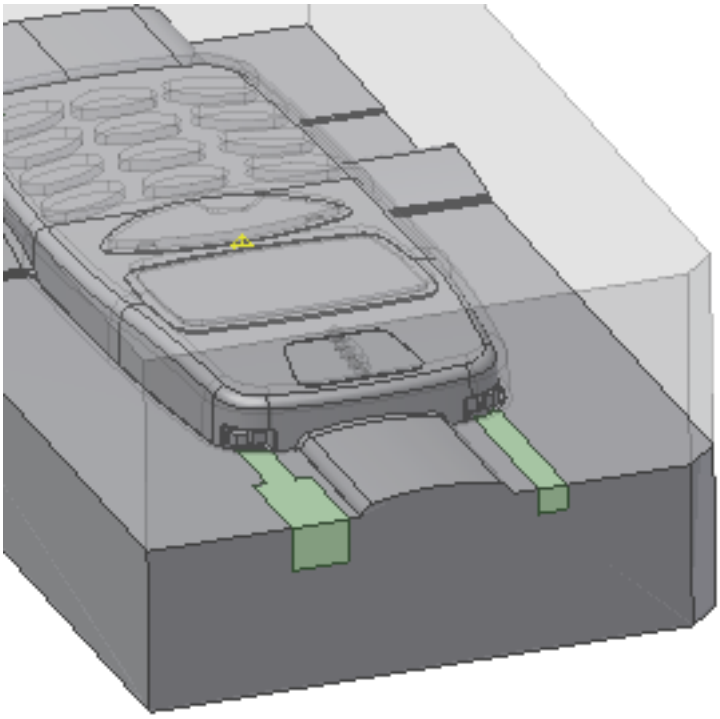


8 Modify the following dimensions:

- Overall width of heel = **8 mm**
- Overall height of heel = **6 mm**
- Distance from Reference 1 to the center = **3 mm**
- Distance from Reference 2 to the center = **2 mm**

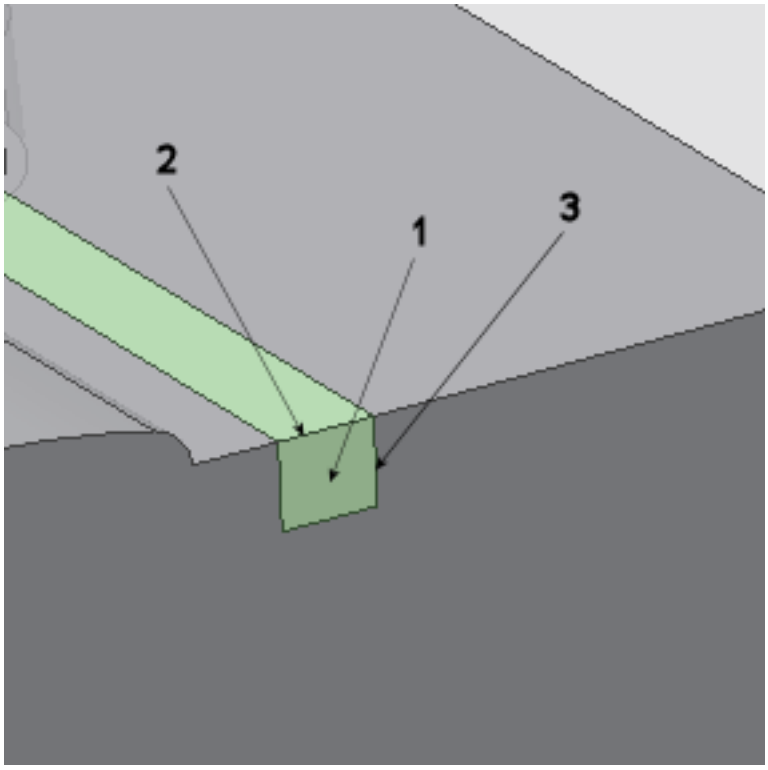
9 On the **Create Heel** dialog box, enter a Distance of **15 mm**.

10 Click **Apply** on the **Create Heel** dialog box to generate the heel. The model appears as shown in the following image.



11 Create a second heel on the other insert using the profile type and shape that was used to create the first heel. Use the references shown in the following image to select the surface and edges:

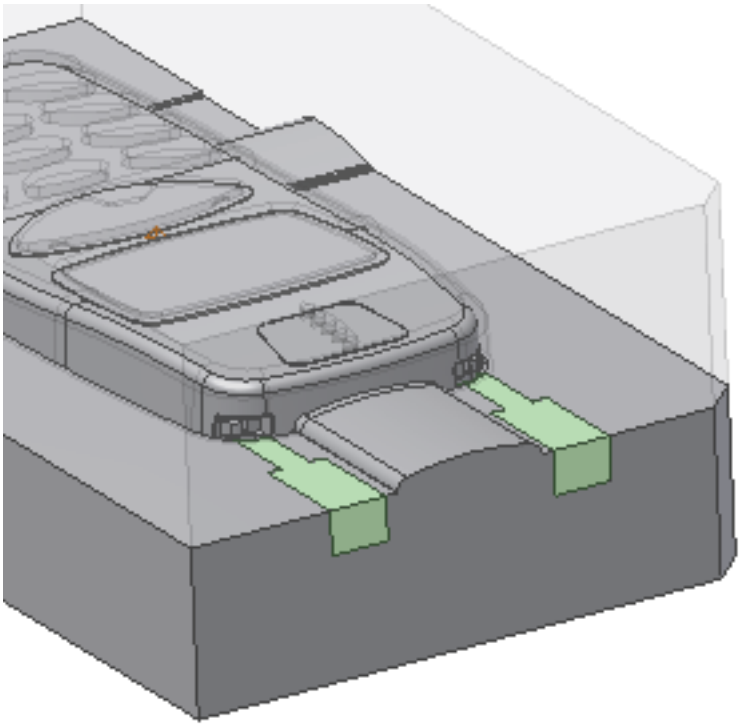
- Select the surface labeled **1** as the Face.
- Select the edge labeled **2** as Reference 1.
- Select the edge labeled **3** as Reference 2.



12 Modify the following dimensions:
- Overall width of heel = **8 mm**
- Overall height of heel = **6 mm**
- Distance from Reference 1 to the center = **3 mm**
- Distance from Reference 2 to the center = **2 mm**

13 On the **Create Heel** dialog box, enter a distance of **15 mm**.

14 Click **OK** on the **Create Heel** dialog box to generate the heel. The model appears as shown in the following image.



15 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 113) | Next (page 120)

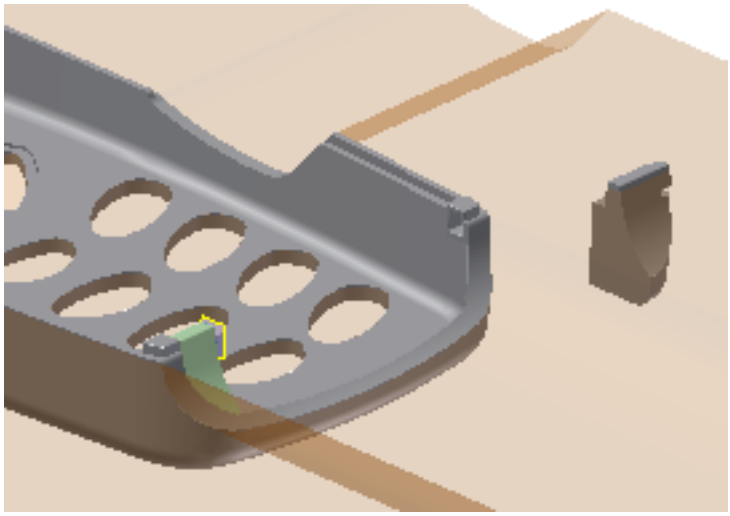
Placing an Insert

In this section, you place an existing insert into the mold assembly by using the **Place Insert** command.

1 On the Quick Access toolbar, click **Design View** > **Assembly Orientation**. The Quick Access toolbar is located along the top of the Inventor window. The assembly is automatically reoriented to this view.

NOTE If you reorient the model, the Assembly Orientation view automatically updates with your display changes, such as rotation, zoom, and component visibility. To return to the previous view, press **F5** on your keyboard. Remember that a limited number of previous views are stored.

1. On the ribbon, click **Core/Cavity tab** > **Insert panel** > **Place Insert**  .
2. From the **Place Insert** dialog box, select **Insert_M3128.ipt** and click **Open**.
3. Left-click your mouse to place the insert.
4. Click **OK** on the **File Naming** dialog box to accept the defaults and generate the insert.
5. Position the insert as shown in the following image.



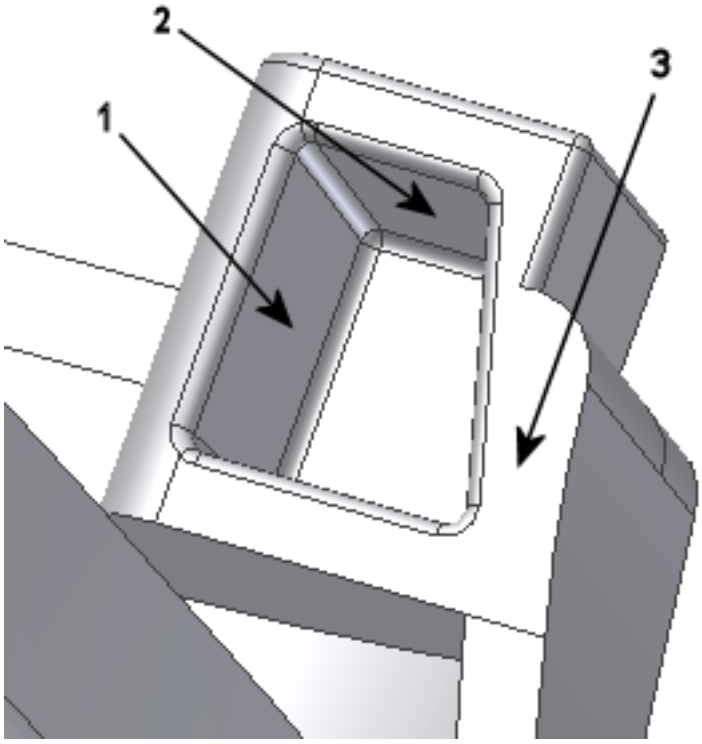
6. On the Model browser, ensure that the visibility of the following components is turned off:
   - Mold Top_Top_WP:1
   - Mold Top_Top_CR:1
   - Mold Top_Top_CV:1
7. Ensure the visibility of Mold Top_Top_MP:1 is turned on.

Previous (page 115) | Next (page 122)

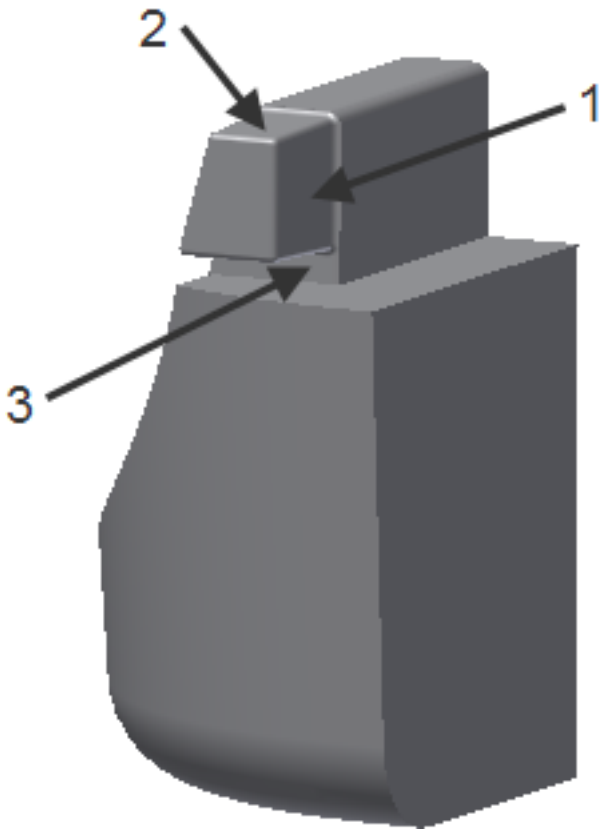
Adding Constraints to the Insert

In this section, you use standard assembly constraints to constrain the insert. The location of the placement will be a mirror image of the insert that was created for you.

1. In the Model browser, expand the Mold Top_Top_MP:1 node to review the constraints as they are added.
2. On the ribbon, click **Assemble tab** > **Position panel** > **Constrain**  .
3. On the **Place Constraint** dialog box, ensure the Mate type is selected  .
4. Select the surface labeled **1** in the following image as the **First Selection**  reference. This surface is on the moldable part component.



5 Select the surface labeled **1** in the following image as the **Second Selection**  reference. This surface is on the placed insert component.



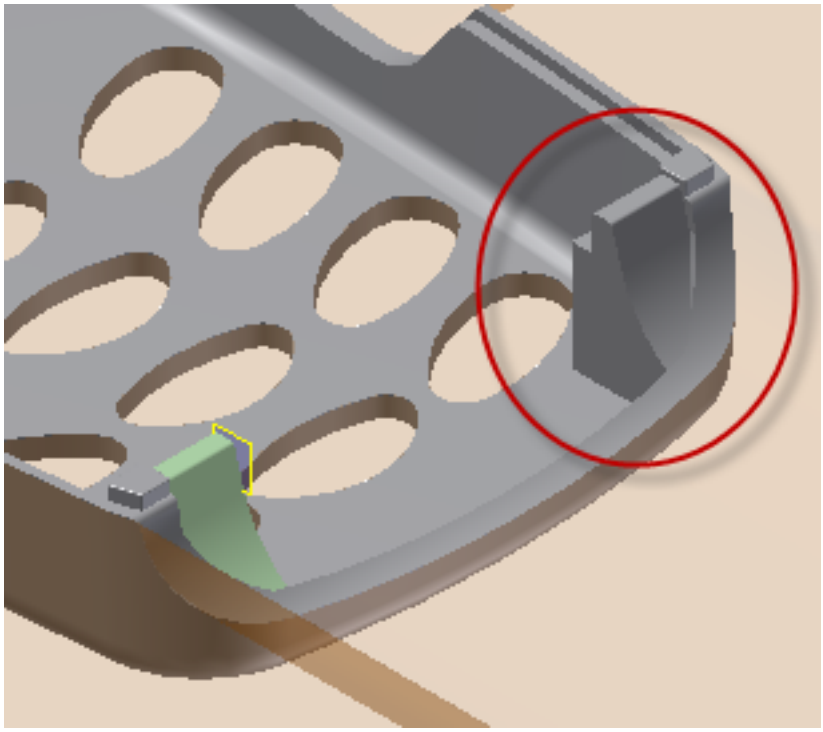
6 Click **Apply**.

NOTE Consider using the  command on the Position panel to drag the component away from its currently constrained location. You can select the reference surfaces more easily.

---

1 Continue to constrain the insert by using mate constraints:
   - Mate the surfaces labeled **2** in the two previous images.
   - Mate the surfaces labeled **3** in the two previous images.

   The fully constrained insert appears as shown in the following image.



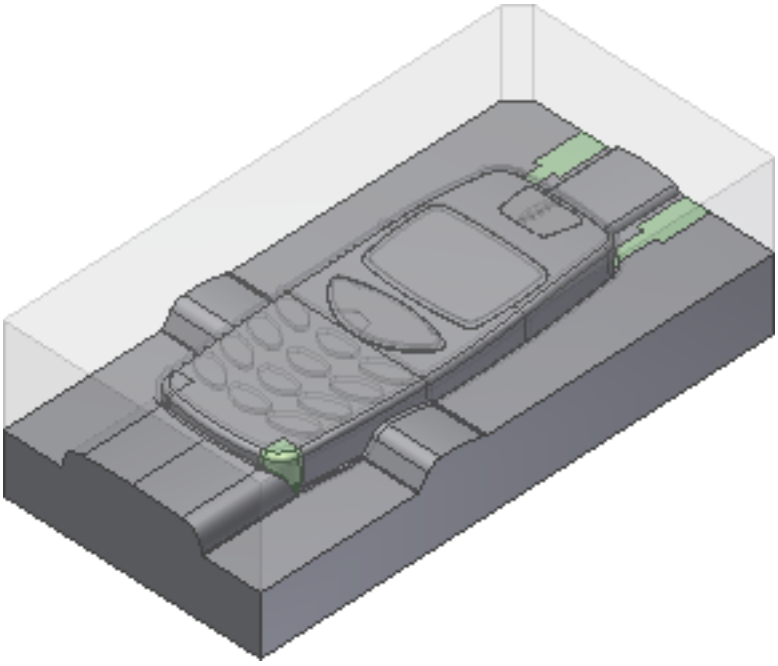
2 When the component is fully constrained, click **Cancel** on the **Place Constraint** dialog box to close the dialog box.

3 To exit the Assembly environment, click **Assemble tab** > **Return panel** > **Return** .

4 On the ribbon, click **Core/Cavity tab** > **Exit panel** > **Finish Core/Cavity** .

5 On the Mold Design browser, expand the **Representations** > **View** nodes, and double-click the **Default** view.

6 On the keyboard, press **F6** to return to the Home View. The model appears as shown in the following image.



7 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 120) | Next (page 126)

Removing the Insert Volume from the Core

In this section, you use the Mold Boolean command to trim the placed insert from the core.

The insert does not automatically trim the core when the **Place Insert** command is used to place an insert in the assembly.

1 From the Mold Design browser, expand the **Representations** > **View: Default** nodes. Right-click **View: Default** and click **New.**

2 From the Mold Design browser, expand the **Core Cavity** node.

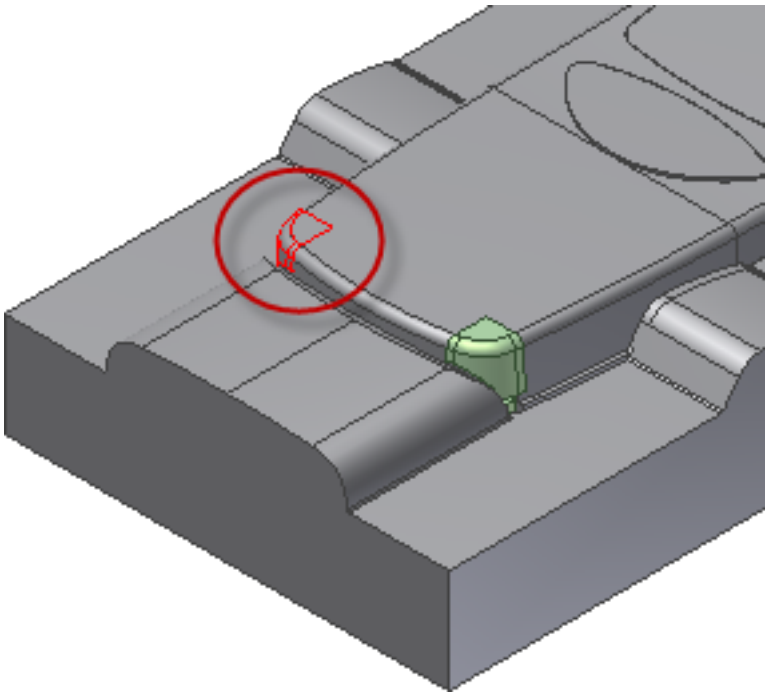
3 Right-click on **Cavity** and clear the **Visibility** option.

4 Right-click on **Workpiece** in the Mold Design browser and ensure the **Visibility** option is clear.

5 On the ribbon, click **Mold Assembly tab** > **Boolean panel** > **Mold Boolean** 

6 The **Cutting tool**  selection tool should be active. Select the insert as shown in the following image.

NOTE Be careful to select the insert and not the core as the cutting tool.



7 The **Body**  selection tool should be active. Select the core.

8 On the Mold Boolean dialog box, click **OK** to remove the volume of the insert from the core.

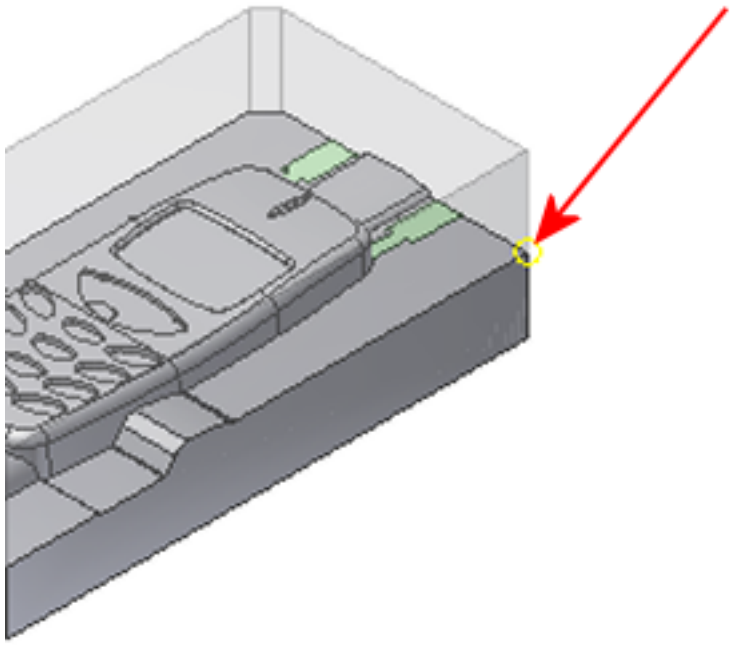
9 On the browser, expand the **Representations** > **View** nodes, and double-click the **Default** view.

Previous (page 122) | Next (page 128)

Placing the Mold Base

In this section, you place the mold base into the assembly. The mold base must be added to the assembly before the slider and lifter assemblies can be generated.

1. On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Mold Base** .
2. On the Mold Base dialog box, the **Placement Ref Point** selection tool should be active. Select the point shown in the following image.



3. On the **Mold Base** dialog box, expand the Vendor and Type menu.
4. From the Vendor menu, select **DME**.
5. Click the Query button.

6 From the window, select **E**.

7 From the Size menu, select **296 mm x 346 mm**.

8 Expand the **Mold Base** dialog box by clicking the arrows at the top or bottom of the right-hand side of the dialog box.

9 In the Component list, click **E 400 296x346x56**.

10 On the right of the selected component, click **Property Settings** 
.

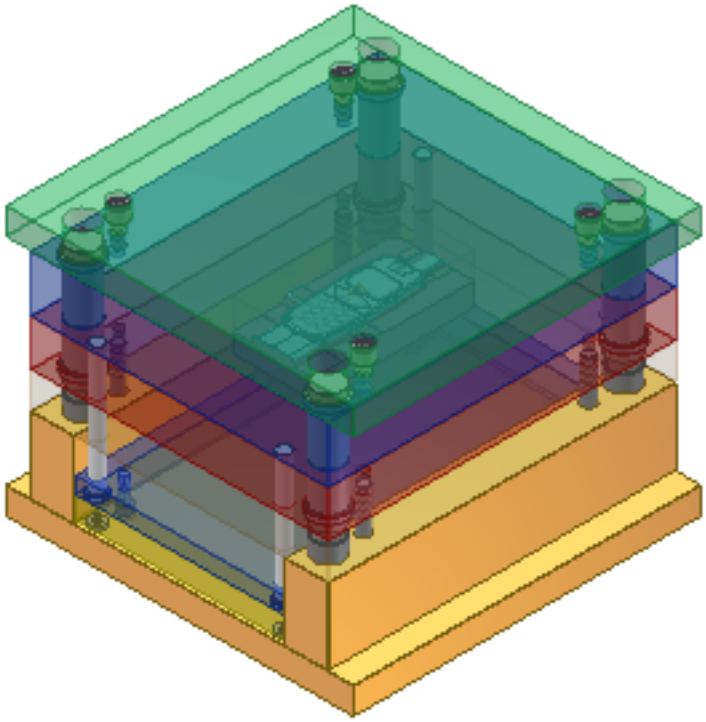
11 In the E 400 dialog box, modify the H_ parameter value to **76 mm**.

12 Click **OK** on the E 400 dialog box.

13 Click **OK** on the **Mold Base** dialog box.

14 Click **OK** on the **File Naming** dialog box to accept the defaults and generate the mold base.

15 The full mold base assembly appears as shown in the following image.

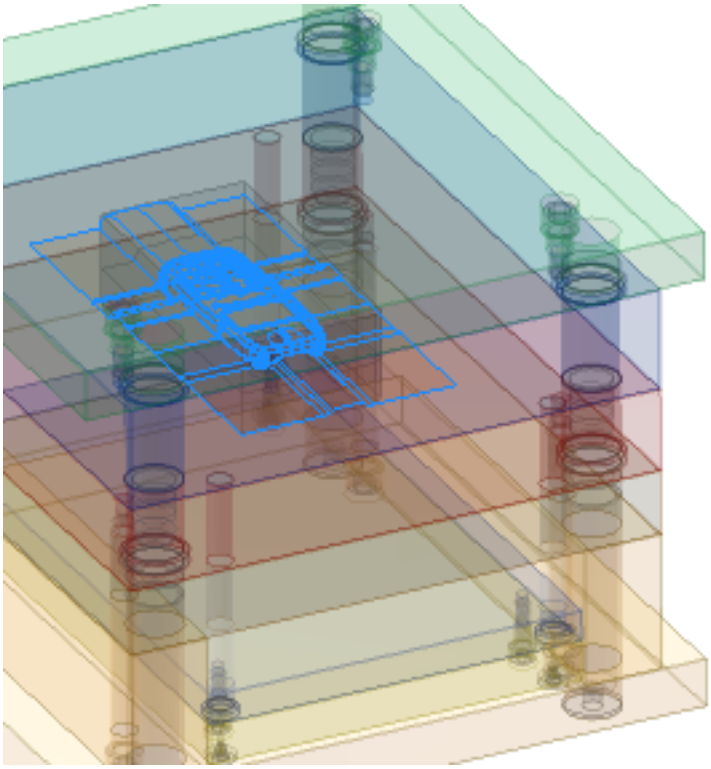


Previous (page 126) | Next (page 130)

Placing the Slider Assembly

In this section, you place the slider assembly. A slider helps to release the plastic part by pulling the side core from the mold.

1. On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Slider**  .
2. Orient the model as shown in the following image. The inserts and heels created for the side core are at the front of the view.



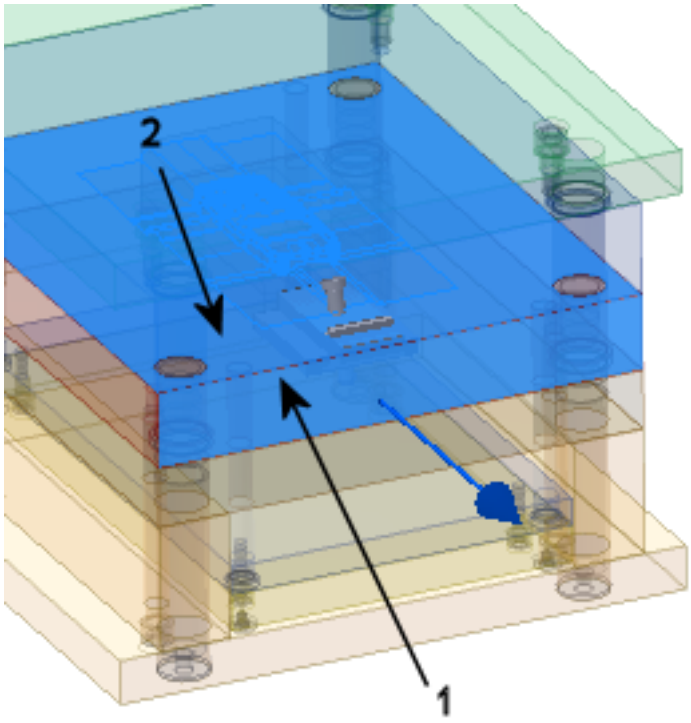
3. On the Slider dialog box, review the preview image to understand the required references.

4 Select the surface labeled 1 on the following image as the **Pull Direction** reference.

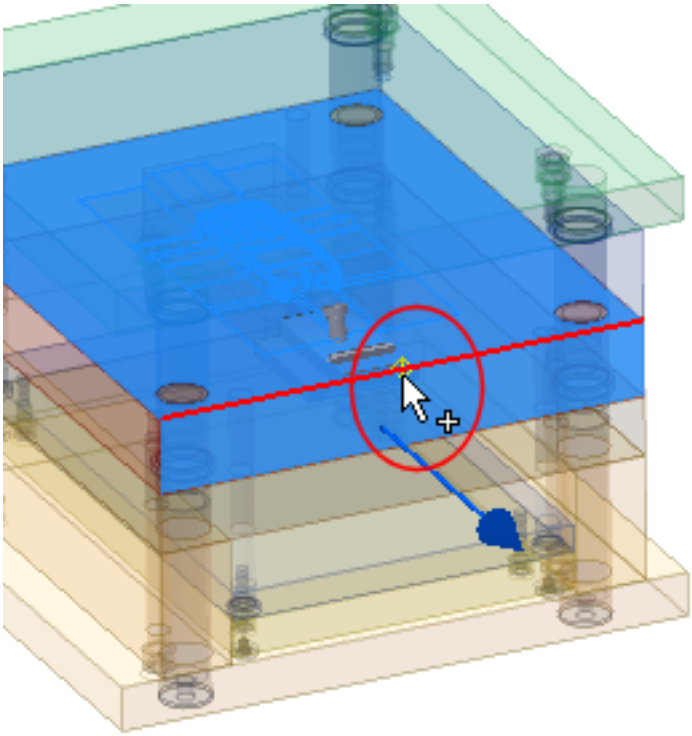
5 Modify the Pull Direction Offset to **-93 mm**.

6 Select the surface labeled 2 in the following image as the **Base Plane** reference.

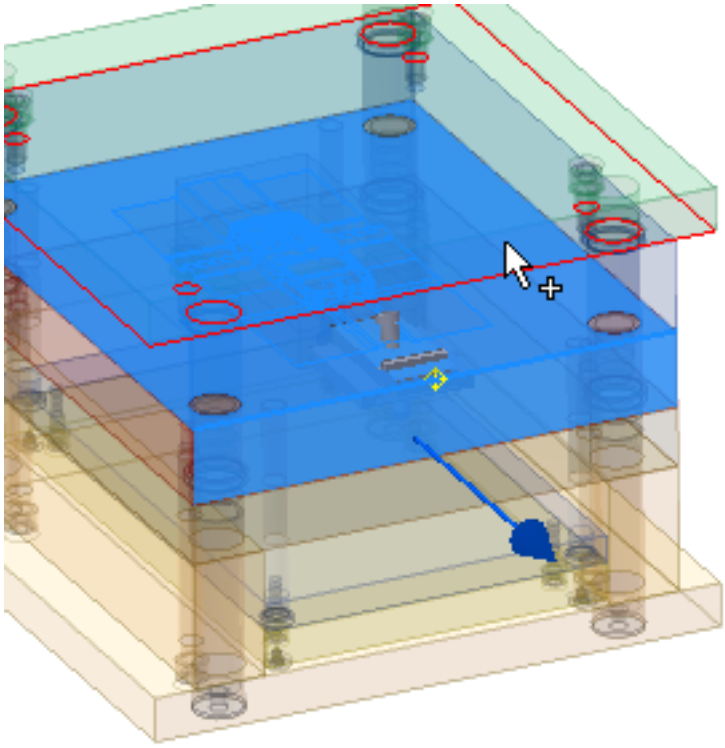
7 On the Slider dialog box, click **Flip Side**  to flip the base plane orientation.



8 Select the midpoint of the edge highlighted in the following image as the **Base Point** reference. When you move the mouse pointer near the center of this edge, it will automatically adjust to this point, and a yellow dot will appear in the display.



9 Select the bottom surface of the top plate as the **Pin Plane** reference, as shown in the following image.

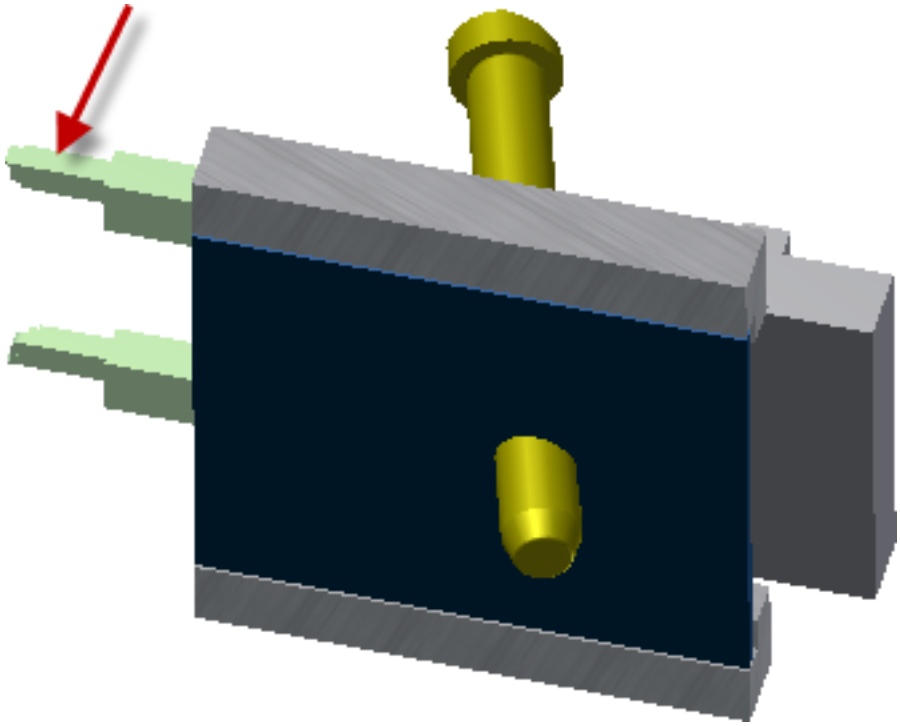


10 On the Slider dialog box, click **Generic detailed preview** .

11 Rotate the model to view the bottom of the Slider assembly.

12 On the **Slider** dialog box, click **Side Core** . Select one of the two side cores (insert or heel features) as the **Side Core** reference, as shown in the following simplified image.

You only need to select one of the inserts because they are both located the same distance away from the slider.



13 On the **Slider** dialog box, click **Update Travel** .

14 At the top of the **Slider** dialog box, click **Customize** to modify the slider assembly parameters.

15 From the Components list, select **Angle Pin**.

16 On the right of the selected component, click **Selection Properties** .

17 Modify the AP_l parameter to **85 mm**.

18 Click **OK** on the **Angle Pin** dialog box.

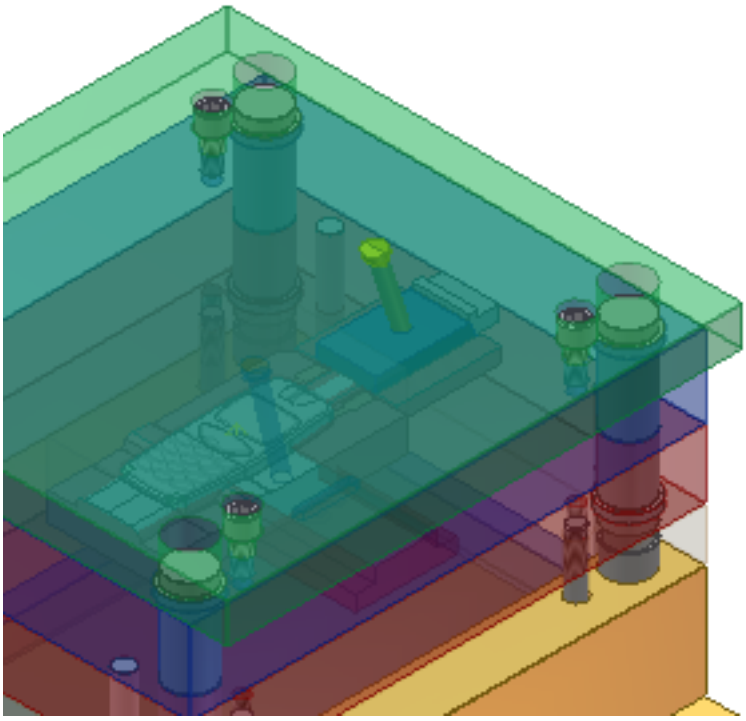
19 Modify the other components as shown in the following table.

Component	Parameter	Value
Slider	SL_b1	50 mm

Component	Parameter	Value
Slider	SL_b2	59 mm
Guide Strip (#1)	SL_b1	50 mm
Guide Strip (#1)	GS_es1	10 mm
Guide Strip (#2)	SL_b1	50 mm
Guide Strip (#2)	GS_es1	10 mm
Locking Heel	LH_A	48 mm

20 On the **Slider** dialog box, click **OK.**

21 Click **OK** on the **File Naming** dialog box to accept the defaults and place the slider.



22 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 128) | Next (page 136)

Placing the Lifter Assembly

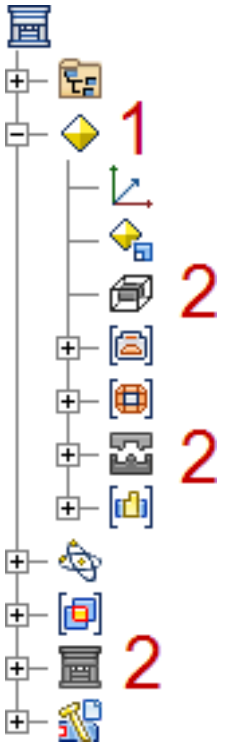
In this section and the next, you place the lifter assemblies.

A lifter is an angled mold component that helps release undercut sections of the plastic part. The lifter is attached to the ejector plate and as the ejector plate moves forward, the lifter moves along an angled channel to release the undercut.

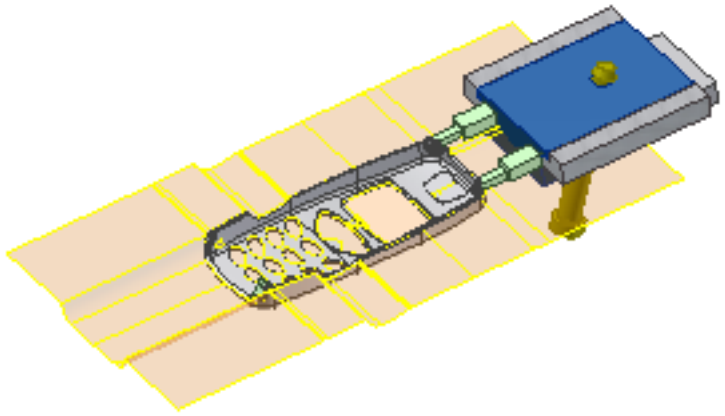
1 On the Quick Access toolbar, click **Design View** > **View1**.

2 On the Mold Design browser, right-click and click **Visibility** as required to match these settings:
   - Plastic part (1) visibility enabled

- Workpiece, Core Cavity, Mold Base (2) visibility off



Orient the model as shown in the following image. View1 is needed for the selection of references in a later step.

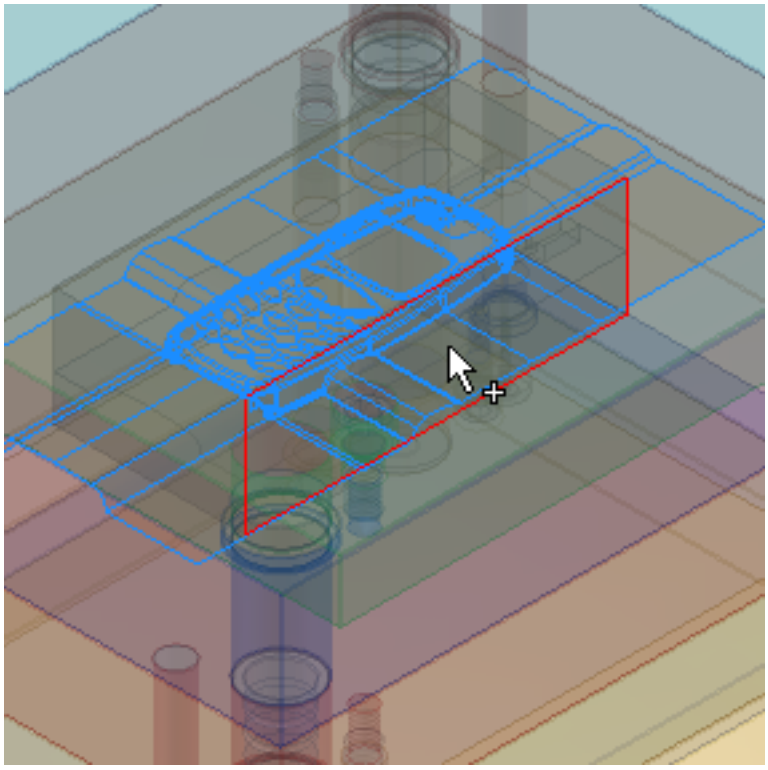


3 Click **Design View** > **Default** to return to the default design view.

4 On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Lifter**  .

5 On the keyboard, press **F6** to return to the Home View.

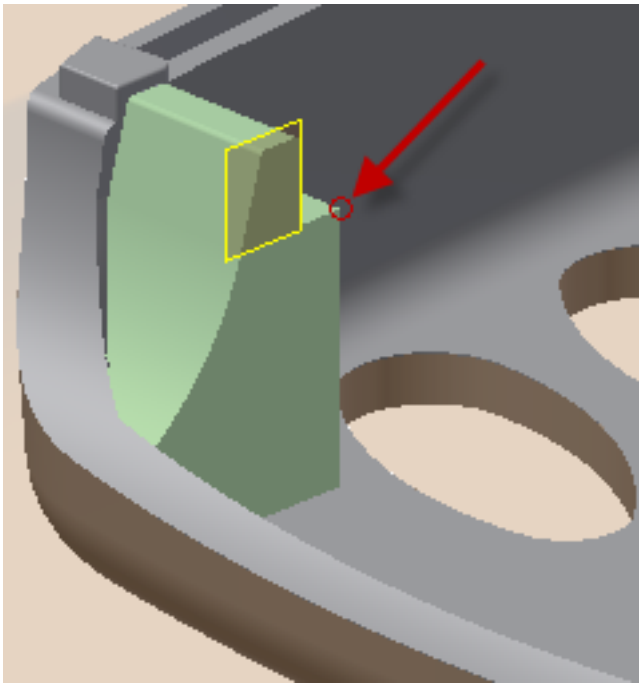
6 The **Select Pull Direction**  tool should be active. Select the side of the workpiece, as shown in the following image.



7 On the Quick Access toolbar, click **Design View** > **View1**.

8 The **Base Point** tool should be active. Select the corner of the insert closest to the highlighted surface shown in the following image.

NOTE The image is simplified for clarity.



9 Modify the following values in the **Lifter** dialog box:
   - Pull Direction **-7 mm**
   - Base Point **-3.75 mm**

10 On the **Lifter** dialog box, click **OK.**

11 Click **OK** on the **File Naming** dialog box to accept the defaults and place the lifter.

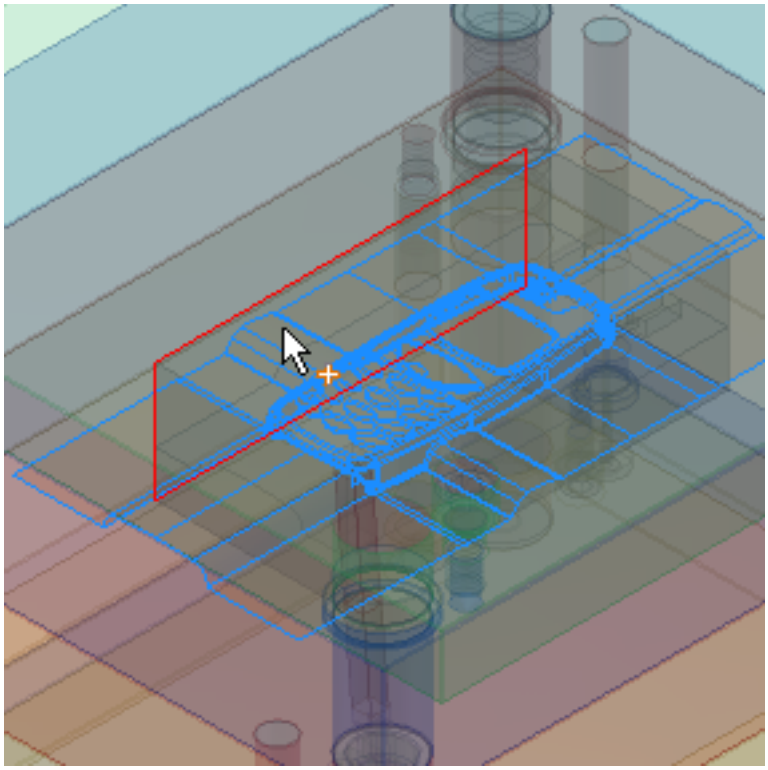
12 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 130) | Next (page 141)

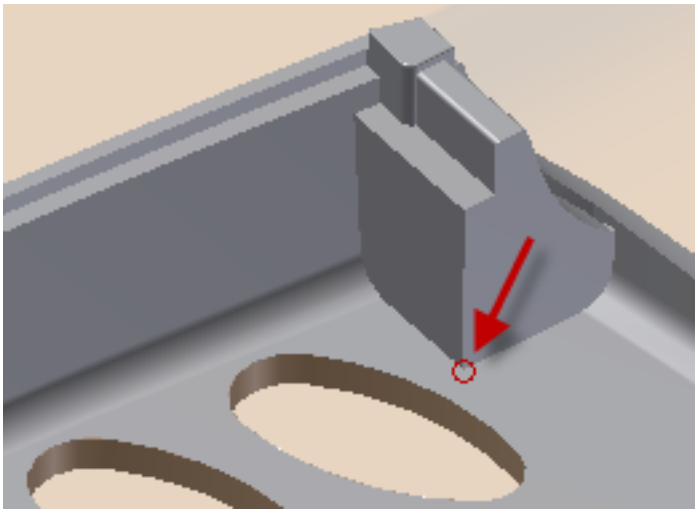
Placing the Second Lifter Assembly

In this section, you add another lifter assembly to the mold.

1. On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Lifter** .
2. On the keyboard, press **F6** to return to the Home View.
3. The **Select Pull Direction** tool should be active. Select the side of the workpiece, as shown in the following image.



4. On the Quick Access toolbar, click **Design View** > **View1**.
5. The **Base Point** tool should be active. Orient that model and select the corner of the insert as shown in the following simplified image.

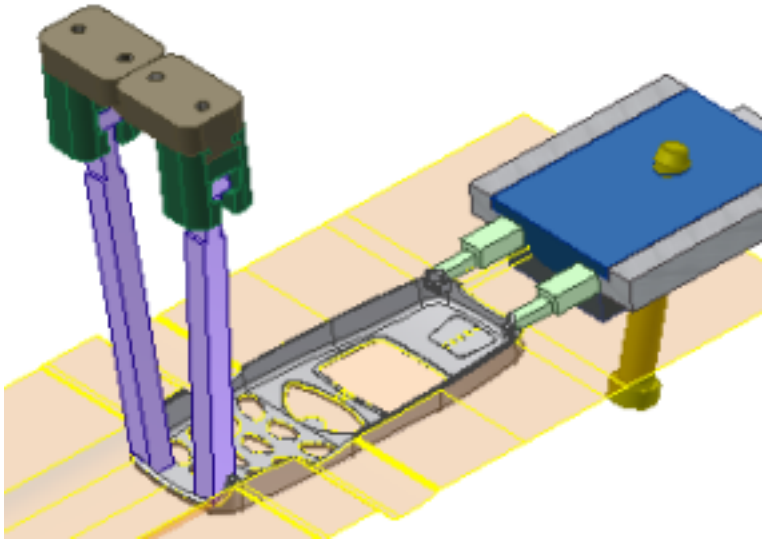


6 Modify the following values in the **Lifter** dialog box:
   - Pull Direction **-7 mm**
   - Base Point **3.75 mm**

7 On the **Lifter** dialog box, click **OK.**

8 Click **OK** on the **File Naming** dialog box to accept the defaults and place the lifter. With the View1 design view active, the model components appear as shown in the following image.

NOTE The orientation of the view may vary.



9 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 136) | Next (page 143)

Performing a Boolean Operation for the Lifter Assembly

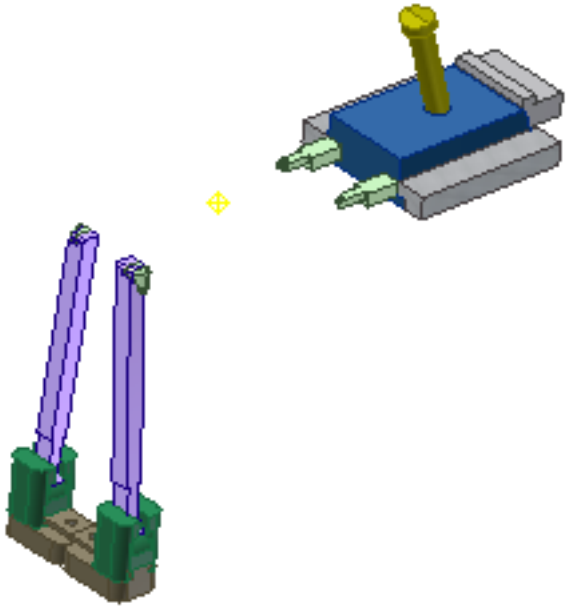
In this section, you perform a Boolean operation that join the inserts to the lifters.

The Mold Boolean command is used to check for interference in the Mold Design assembly, cut a component body by using the cutting tool components, or unite two components into one.

1 On the Quick Access toolbar, click **Design View** > **View1**.

2 On the Mold Design browser, turn off the visibility of Top.

3 Orient the model as shown in the following image.

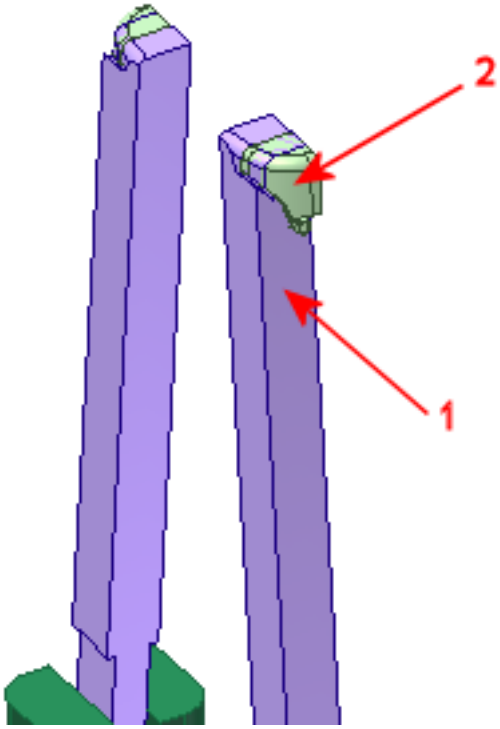


1 On the ribbon, click **Mold Assembly tab** > **Boolean panel** > **Mold Boolean** .

2 On the **Mold Boolean** dialog box, click **Add** .

3 Select the lifter labeled 1 in the following image as the **Unite Body** reference.

4 Select the insert labeled 2 in the following image as the **Join Body** reference.



5 On the Mold Boolean dialog box, click **Apply** to unite the selected bodies.

6 Repeat the steps to unite for the other lifter and insert and then click **OK** to finish the operation.

NOTE You are not required to create a Mold Boolean for the slider assembly in this tutorial.

Previous (page 141) | Next (page 145)

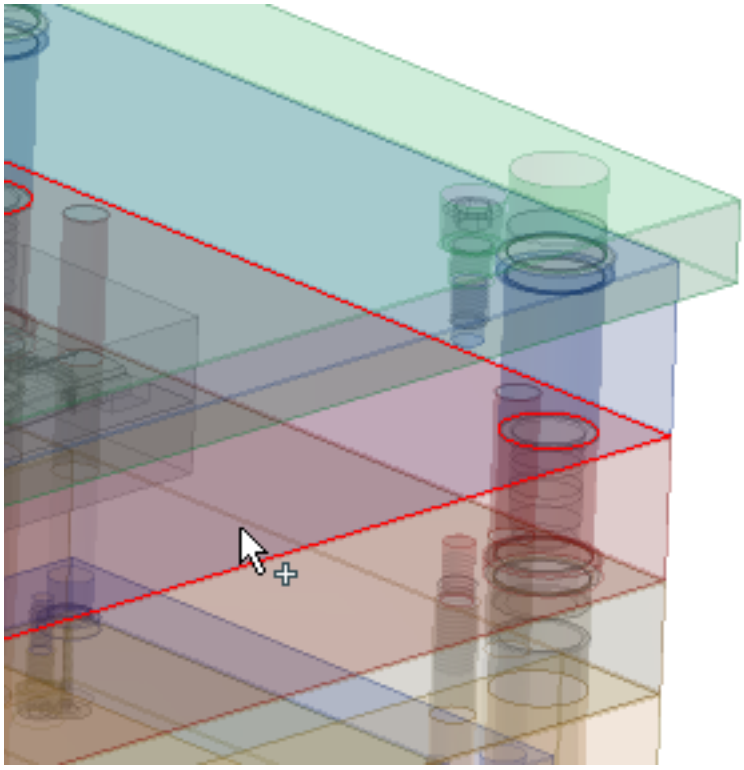
Placing the Lock Sets

In this section, you place two side lock sets. A lock set ensures accurate alignment between the two halves of the mold. The two types of locks sets that can be created are side locks and interlocks.

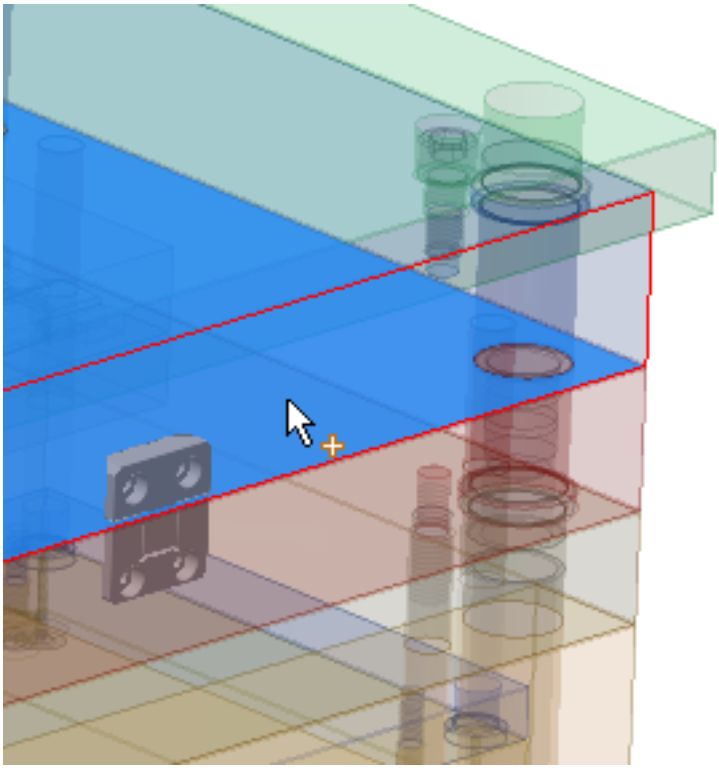
Side locks are machined on the centerline axis, both vertically and horizontally, on the outside of the mold to ensure alignment of the mold halves. Interlocks also align the mold halves, as well as lock the core and cavities. Interlocks maintain proper alignment while still permitting thermal expansion between the mold halves.

1. On the Quick Access toolbar, click **Design View** > **Default**.
2. On your keyboard, press **F6** to return to the Home View.
3. On the ribbon, click **Mold Assembly tab** > **Mold Assembly panel** > **Lock Set**  .

1. On the **Lock Set** dialog box, expand the **Type** menu.
2. From the Vendor menu, select **DME**.
3. From the Category menu, select **Side Lock**.
4. From the window, select **FW45**.
5. On the **Lock Set** dialog box, the **Base face**  tool should be active . Select the surface shown in the following image. The base face is the top or bottom surface of one of the mold base plates, which establishes the location of the parting plane.



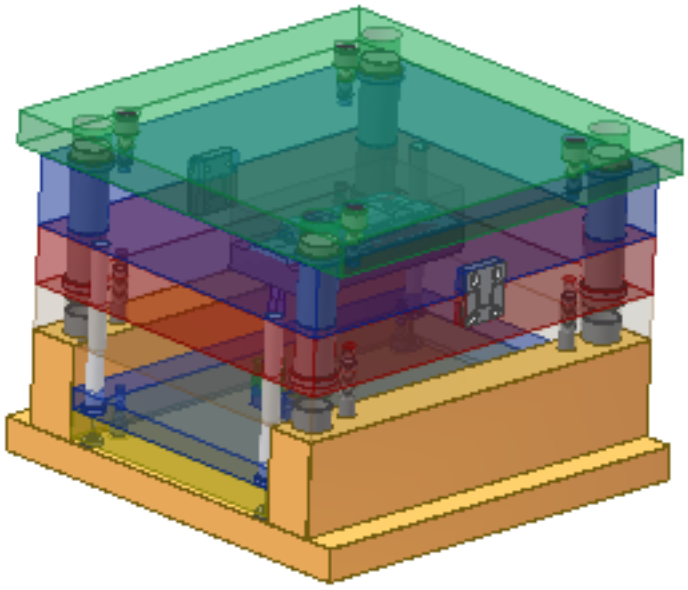
6 On the **Lock Set** dialog box, the **Reference face** tool should be active . Select the surface shown in the following image. The reference face on which the side lock will be attached must be perpendicular to the selected base face.



7 On the **Lock Set** dialog box, specify both the distance and the offset as **0mm**. The distance represents the measurement between the center of the reference face and the center of the side lock. The offset is the distance into the mold plate that the side lock will be embedded.

8 On the Lock Set dialog box, click the **Symmetrical Placement** option to create a second lock set that is symmetrical with the first lock set. Click **OK.**

9 Click **OK** on the **File Naming** dialog box to accept the defaults and place the lock sets. The model appears as shown in the following image.



Previous (page 143) | Next (page 149)

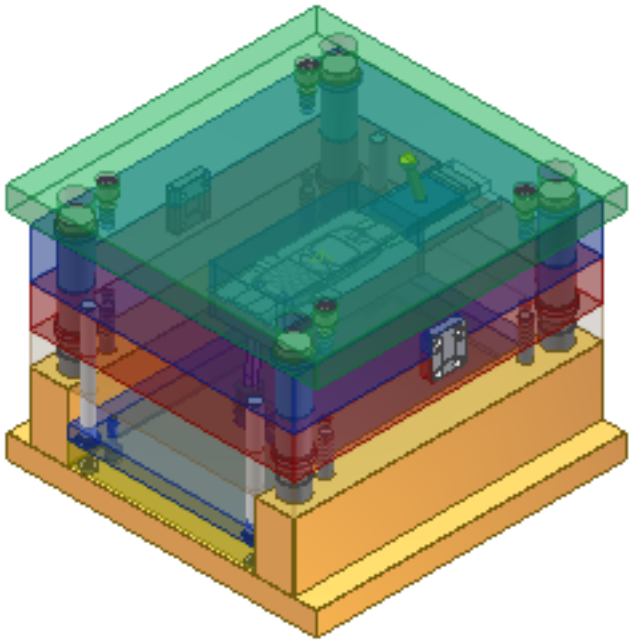
Generating the Workpiece Pocket

In this section, you will create a workpiece pocket. The workpiece pocket is a type of Boolean operation, which creates an excavated pocket in the mold base to hold the core and the cavity.

1. On the ribbon, click **Mold Assembly tab** > **Boolean panel** > **Workpiece Pocket**  .
2. On the **Workpiece Pocket** dialog box, select **Hole Clearance** from the Pocket Type menu.
3. Click **Chamfer**  . Create a chamfer on the top left-hand side, as indicated on the image in the **Workpiece Pocket** dialog box. The chamfer is added because the workpiece has a chamfer along this corner to create a foolproof design.

4 Modify the value of the chamfer to **5 mm** to match the value of the chamfer on the workpiece.

5 On the **Workpiece Pocket** dialog box, click **OK** to generate the workpiece pocket. The final mold assembly appears as shown in the following image.



NOTE It is difficult to see the workpiece pocket in the model. Select the **Workpiece Pocket** node in the Mold Design browser to highlight it on the model.

6 On the Quick Access toolbar, click **Save** to save the assembly.

Previous (page 145) | Next (page 150)

Summary

In this tutorial, you learned to incorporate inserts, lifter and slider assemblies, and lock sets into a mold assembly design.

You learned how to:

- Create and add existing inserts to a mold assembly.
- Create heels associated with existing inserts.
- Create lifter and slider assemblies.
- Perform Boolean operations on lifters and sliders.
- Place lock sets in a mold assembly.
- Generate the workpiece pocket.

What next?

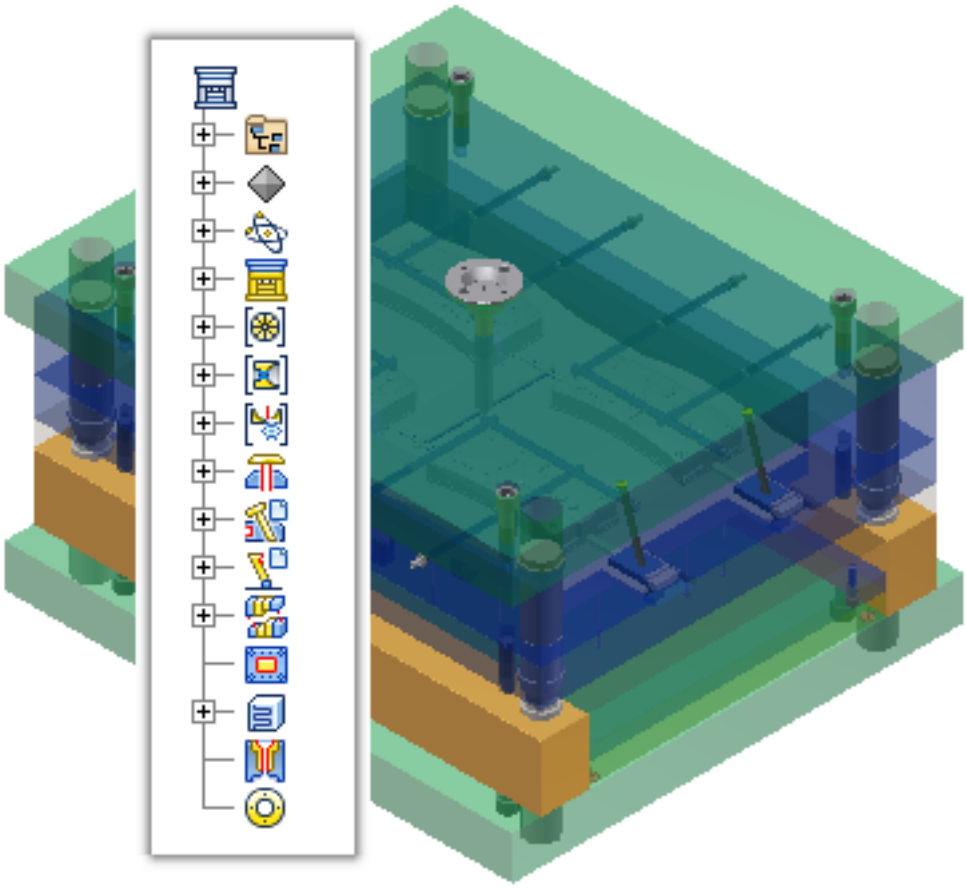
You can place ejector components for the mold assembly and also perform the Boolean operations on the slider assembly.

Previous (page 149)

Mold Assembly Organizational Structure

5

About this tutorial



Review the file organizational structure in a mold assembly.

Skill level	**Level 3**
Time required	60 minutes
Prerequisites	■ Completion of the Projects Tutorial from the Inventor Tutorial Suite.

	■ Completion of the About this tutorial (page 1)Tutorial. ■ Know how to set the active project and navigate the model space with the View commands. See the Inventor Help topic *Getting Started* for further information. ■ An understanding of the Inventor Content Center is also recommended.
Tutorial file used	CoverMold directory

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Navigation

Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Tutorial objectives

Inventor Mold Design generates a significant number of files when a mold assembly is created. In this tutorial, you examine these files and discover their naming conventions and functions. After completing this tutorial, you will be able to do the following:

- Identify the mold assembly files that are created and stored in the Project Location in Windows Explorer.
- Recognize the nodes that are added to the Mold Design browser display when features are added.
- Recognize the file hierarchy that is displayed in the Model browser for a mold assembly.

  NOTE This tutorial uses the default file naming scheme. Right-click the Mold Design node in the Mold Design browser and select **Prompt for File Names**. You can accept the default names and locations, or choose new file names and locations.

Next (page 156)

Understanding Project Files and Mold Libraries

In this topic, you examine project files, which are used in the Inventor Mold Design environment to manage and locate referenced files. You also examine the Mold Library and see how the Inventor Mold Design application accesses it to create components within mold assemblies.

When working with Inventor Mold Design, a significant number of files are created and referenced. Project files impact the mold environment in the following ways:

- The Templates folder identifies where the program stores file templates. The mold design templates available for use are Metric (DIN and mm) and English (in).
- The Content Center Files folder identifies the root folder for the Content Library files of the project. Some of the standard components used in Inventor Mold Design are managed by the Content Center. It is important that this path correctly specifies the location of the Content Center. Components that reference the Content Center include screws, guide pins, and guide bushings.

Inventor Mold Design supports a wide range of components from commercial catalog libraries, including Mold Base, Ejector, Slider, Lifter, Sprue Bushing, Locating Ring, and Lock Sets. These predefined components comply with the industry standards of D-M-E, Futuba, HASCO, Progressive, Punch, Misumi, Sideco, Meusburger, Strack, Pedrotti, and Rabourdin. The components are all created by the Inventor Mold Design application using the mold library, not the Content Center.

As with libraries in the part/assembly design environment, you must attach the necessary libraries to the project file when you use libraries in the mold environment. It is important to map the Mold Library to ensure that the Inventor Mold Design application can generate any required mold assembly components. In addition to these components, standard components are generated in Inventor Mold Design from the Content Center, and the Content Center manages these files.

Previous (page 154) | Next (page 157)

Activating the Project and Opening the Assembly

In this topic, you open a completed mold assembly.

The mold assembly and its associated files, including library files, are provided for you. The project file of the original designer is also included.

1. In Windows Explorer, navigate to the Mold tutorials and expand the CoverMold directory.
2. Notice that the folder contains the following subfolders to help organize the data files:
   - The Libraries folder contains the files that were generated from the Content Center. Because the assembly is no longer on the system on which it was created, this directory was provided with all the required Content Center data.
   - The Cover Mold folder contains all the mold assembly files that were generated when the mold assembly was created.
3. Minimize Windows Explorer. Launch Inventor if it is not yet running.
4. Close all open windows in Inventor  . Project files cannot be activated if files are open.
5. On the ribbon, click **Get Started tab** > **Launch panel** > **Projects**  .
6. Click **Browse**. Navigate to and select the project file: CoverMold.ipj.
7. Click **Open**. This project file, which was provided by the mold assembly designer, is listed in the **Projects** dialog box and is automatically set as the active project.
8. Click **Done** to close the **Projects** dialog box.
9. On the ribbon, click **Get Started tab** > **Launch panel** > **Open**  .
10. Navigate to and select CoverMold.iam.
11. Click **Open** to open the mold assembly. The **Resolve Link** dialog box appears, indicating that the mold assembly has failed to open. It is because it cannot find components that were used in the original assembly. If you searched for the missing file, you would find it in the

Libraries directory that has been provided. The assembly cannot find this directory due to problems in the project file.

12 Click **Cancel** in the **Resolve Link** dialog box.

Previous (page 156) | Next (page 158)

Resolving the Retrieval Failure

In this topic, you edit the project file to resolve the failure experienced when opening the mold assembly in the previous topic.

1 On the ribbon, click **Get Started tab** > **Launch panel** > **Projects**  .

2 Ensure that the active project is still **CoverMold**.

3 Expand **Folder Options** at the bottom of the dialog box. Notice that this points to the default location for the Content Center. Because the files were generated on another system, they no longer exist in that directory. The path must be edited.

4 Select **Content Center Files = [Default]**, right-click, and select **Edit**.

5 Click **Browse**  .

6 In the install location for the tutorial files, select **...\Inventor 2011\Tutorial Files\Mold\CoverMold\Libraries\Content Center Files**.

7 Click **OK**.

8 It is important to ensure that the project files are accessing all the libraries required for the design. In the **Projects** dialog box, click **Configure Content Center Libraries**  .

9 In the **Configure Libraries** dialog box, scroll to the Inventor Mold libraries in the list, and ensure that the **In Use** check box is enabled. Click **OK**.

NOTE If you do not enable the required mold libraries, you can still open the mold assembly, but you cannnot add features that access the Mold Library.

10 Click **Save** and **Done**.

11 On the **Launch** panel, click **Open**  .

12 Navigate to and select **CoverMold.iam**. This time, the mold assembly opens without any failures.

Previous (page 157) | Next (page 159)

Generating Files When Starting a New Mold Assembly

In this topic, you start a new mold assembly to help you understand the directory and naming conventions used when creating a mold assembly.

You then cancel the mold assembly creation and begin reviewing the files in the provided mold assembly. You review the files up to the point where the moldable part is added to the assembly.

1 Click **Inventor** 

to start a new mold assembly.

2 Select **New** to open the **New File** dialog box.

3 From the **Metric** tab of the **New File** dialog box, click **Mold Design (mm).iam** and click **OK**. The **Create Mold Design** dialog box appears, enabling you to assign the Mold Design File Name and its location. The fields for the Cover Mold assembly were defined as follows:

- Mold Design File Name: **CoverMold.iam**
- Mold Design File Location: This lists the default location in the Tutorial directory.

4 This assembly will not be created so click **Cancel**.

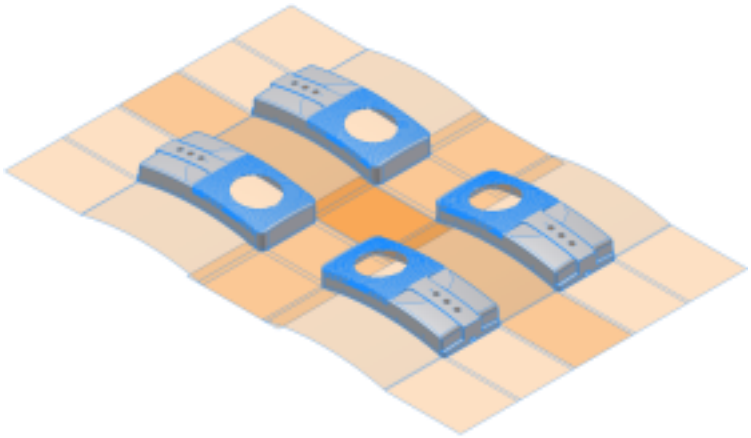
5 You now review the completed model that is currently open in Inventor. In Windows Explorer, click the directory to display its contents. Expand all the directories for easy viewing.

Previous (page 158) | Next (page 160)

Understanding the Mold Design Browser Nodes

In this topic, you review the Mold Design browser nodes that are created when the mold assembly is created and the moldable part is added.

1. Return to the Inventor Mold Design application and ensure that the browser display is set to Mold Design. The first node in the browser is the Representations node. Like standard assembly design functionality, it enables you to create view representations for component display purposes.
2. Below the Representations node is the cover node. Expand this branch. The cover node represents the moldable part (_MP). All the subnodes indicate the commands that have been executed directly on the molded part.
3. Right-click the **cover** node and click **Visibility** to turn on its display.
4. Right-click the **cover** node again, and click **Isolate** to clear all other components from the display. It is the core/cavity for the moldable part, as shown in the following image.

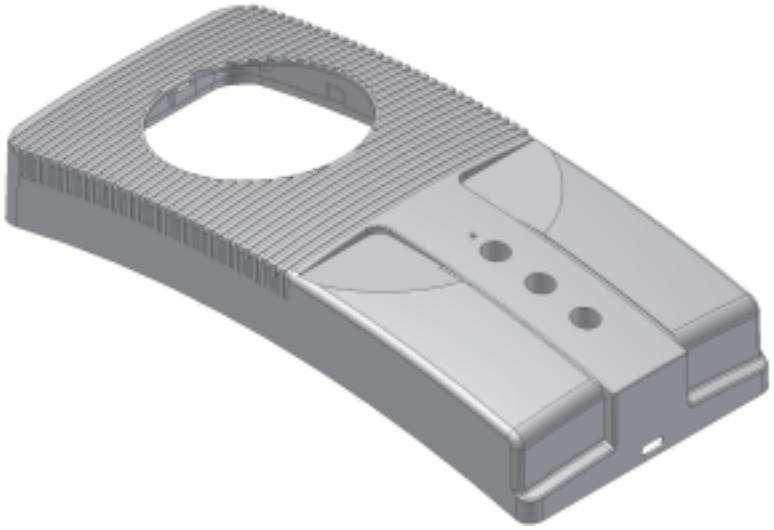


The cover node of the browser contains significant information about such things as the orientation of the model, shrinkage, core and cavity, patterns, and gate locations. This information is stored in the file. The _MP appended to the end of the file name indicates it is the moldable part file.

1. Right-click the **cover** node and click **Open**. The CoverMold_cover_MP.ipt model opens. Review the browser. The _MP file contains all the information related to the core and cavity design.

In this file, you can see all the surface patches and runoff surfaces that were used to create the parting surface.

2. The fourth node in the _MP file is the original moldable part file. Right-click **cover.ipt** in the browser and click **Open Base Component**. The original moldable part file opens, displaying all the features used to create the model.



The cover.ipt model can be modified to make product changes, which are reflected in the moldable part. The designer can also edit the CoverMold_cover_MP moldable part to satisfy any mold design requirements.

1. Click **Close** in the top right-hand corner of the cover.ipt window to close the part models.
2. Close the **CoverMold_cover_MP.ipt** file. The CoverMold.iam file should be the active file.
3. Right-click the **cover** node and select **Undo Isolate**.

Previous (page 159) | Next (page 162)

Understanding the Layout and Part Zone Assemblies

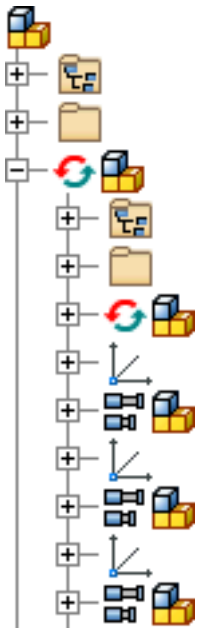
In this topic, you review the layout and part zone assemblies that are created when a moldable part is added to the mold assembly.

Assigning a material to the mold assembly and the effects it has on the mold files will also be discussed.

The **_LY** (layout) and **_PZ** (part zone) files are two additional files that are created after moldable parts are added to the mold assembly and the assembly is saved. These files are used in the following ways:

- The CoverMold_cover_LY.iam assembly is a top-level layout assembly (_LY). If multiple different components are added to the mold assembly, an _LY.iam file is generated for each unique component.
- The CoverMold_cover_PZ.iam assembly is a part zone assembly (_PZ) in the layout assembly, which contains the moldable part design file. For each instance in a family mold pattern, a PZ assembly is created.

1. At the top of the browser, change the display to Model and then expand the **CoverMold_cover_LY** assembly. Notice that there are four CoverMold_cover_PZ.iam assemblies in the mold assembly, one for each of the four patterned family mold instances. The browser appears as follows:



The next step in the mold design workflow is to define the material type. No files are generated during material definition, but a Material node is added to the Mold Design browser.

1. At the top of the browser, change the display back to Mold Design. The Material node is created when the material is assigned.
2. Expand the **Material** node to review the material type.

Previous (page 160) | Next (page 163)

Examining the Files that are Generated for Core and Cavity and Mold Base Creation

In this topic, you examine all the files that are created in the mold assembly to generate the core and cavity. You also examine the files created by the addition of the mold base.

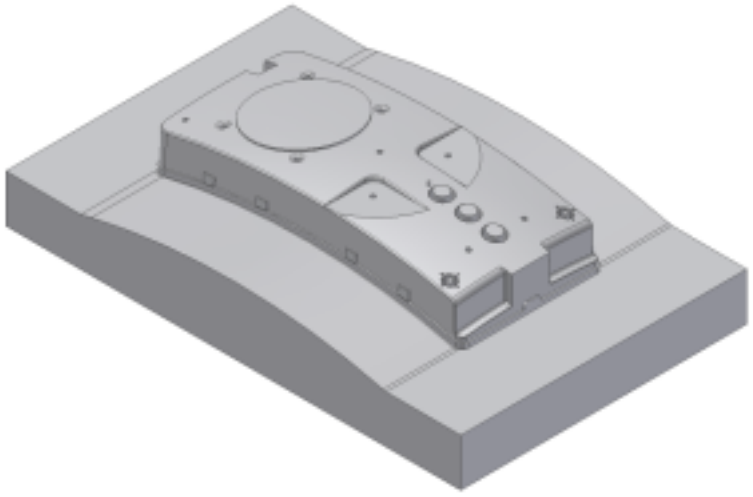
1. Review the Mold Design and Model browser displays. With the addition of the workpiece, a _WP file is generated when the assembly is saved. This _WP part file contains a single extrude feature that creates the

workpiece geometry. In the cover node in the Mold Design browser, a Workpiece node indicates that the workpiece was added. In the Model browser, the workpiece file is located in the Part Zone (_PZ) assembly.

2 When creating the parting surface, you first create a patch surface and then you create the runoff surfaces. Although no new files are created during this step, the _MP file is modified. The only change to the Mold Design browser display is that Patching Surfaces and Runoff Surfaces nodes are now listed in cover.

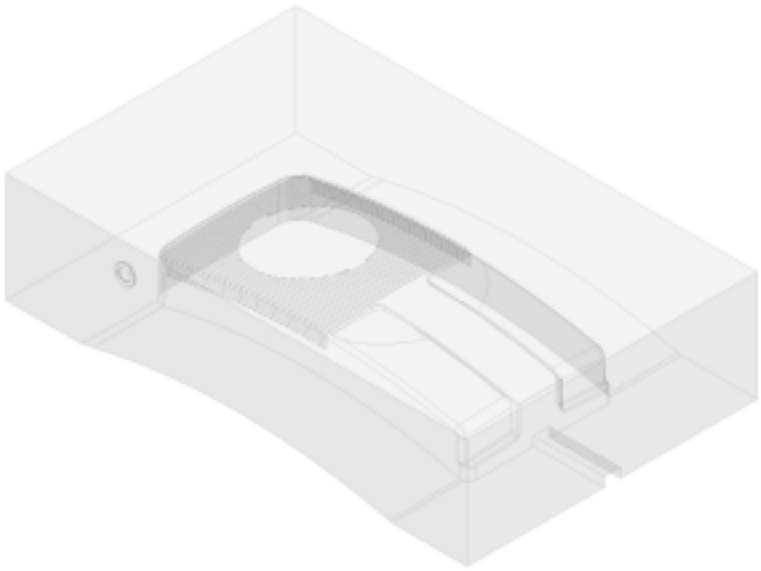
3 With the creation of the core and cavity, a Core file (_CR) and a Cavity file (_CV) are generated, and a Core Cavity node is added to the cover node in the Mold Design browser. These files are located in the Part Zone (_PZ) assembly.

4 In the Model browser, expand the **_LY** and **_PZ** assemblies, right-click the **CoverMold_cover_CR** file, and click **Open**.



CoverMold_cover_CR

5 To return to the mold assembly, click the **CoverMold.iam** tab at the bottom of the working window. Right-click the **CoverMold_cover_CV** file and click **Open**. Review the geometry in both these parts. The geometry of these files was based on the moldable part.



CoverMold_cover_CV

6 Click **Close** in the top right-hand corner of the working window to close the two part models. The mold assembly should now be the active model. The geometry in these models can be modified directly within their individual part files. However, if changes are required to the geometry of the core/cavity that was generated by the moldable part, return to the _MP file for editing.

7 The CoverMold_cover_IN1 part file that is listed in the Cover Mold directory is an Insert. It was created when the Insert was added to the moldable part. The _IN1 file is located in the _PZ assembly. Switch to the Mold Design browser display. The Insert is included at the bottom of this list in the cover node because it was applied to the moldable part.

The addition of a mold base to the mold assembly creates a substantial number of files. To manage these files, a single _MB assembly file is added to the mold assembly, and all its files are stored in their own assembly directory. This directory is created in the assigned mold Project file location and is named MB. A subfolder in this directory indicates the Vendor of the mold base that was selected, and all the files are generated from the mold library for this vendor. A Mold Base node is added to the Mold Design browser display.

Previous (page 162) | Next (page 166)

Examining the Runner System Files and Component Directories

In this topic, you examine the files that are created for the mold base runner system, and some other mold assembly component commands.

To create the runner system in a mold assembly, define the gate location, runner sketch, runner, gate, and cold well. With the initial definition of the gate location, an _RG (Runner) file is created in the Cover Mold directory. Review the CoverMold_RG.ipt file in the Model browser. The _RG file is an adaptive file, as indicated by the adaptive symbol at the beginning of the file name  . As each item that makes up the runner system is created, the _RG file updates all the information.

1. Right-click **CoverMold_RG.ipt** and select **Open**. The runner appears as shown in the following image.

2. Click **Close**  to close this model.
3. Review the Mold Design browser. The Runner, Gate, and Cold Well nodes were added to the model when these commands were executed on the mold assembly.

   When you add Ejecting Components, Sliders, Sprue Bushings, Locating Rings, and Lifters, directories are also created for all their generated

components. For example, the following directories were created in this assembly:

- Ejectors
- Lifters
- Locating Ring
- Sliders
- Sprue Bushing

The files in each directory are generated from the mold libraries. For any command that can generate multiple components, multiple groups are created in its directory. For example, a Group N.0 folder is generated for each ejector pin grouping that is added to the assembly. There is only one grouping of ejector pins in the CoverMold.iam mold assembly so the directory appears as follows:

- Ejectors (1)
- Type (2)
- Group (3)

4 In Windows Explorer, navigate to the Cover Mold Ejectors directory to review the files and discover the naming conventions for files generated from the libraries. Also expand and review the following directories:
   - Lifters
   - Locating Ring
   - Sliders
   - Sprue Bushing

5 Review the Model browser display and notice the following near the bottom of the browser:
   - The _AG 156-7-0 file was created when the sprue bushing was inserted.
   - The _DHR 21 100-12 file was created when the locator ring was inserted.

6 Expand the **_LY** and one of the **_PZ** assemblies. The six Group 1 _Ejector Pin# models are located here because they were used to eject the plastic part.

7 Review the Mold Design browser display and notice the nodes in the browser. The order in which the following nodes are listed indicates the order in which they were added to the mold assembly: Ejecting Components, Slider Assemblies, Sprue Bushing, Locating Ring, Lifter Assemblies.

Previous (page 163) | Next (page 168)

Examining the Files Generated by Combining Cores and Cavities and Adding Cooling and Heating Components

In this topic, you examine the files that are created when using the Combine Cores and Cavities command, and when cooling and heating components are added to the mold assembly.

When working with a family mold design or a multi-cavity mold design, the Combine Cores and Cavities command is used to combine separate cores and cavities into one core and cavity. In this assembly, the cover model was assembled and patterned to create a family mold with four moldable parts. The _Combined CV1 and _Combined CR1 files were generated from this command, and are added to the Cover Mold directory for the project.

1 Review the Model and Mold Design browser displays. These two files are listed in the Model display of the browser at the top level of the mold design assembly. The Combined Core/Cavity node is listed in the Mold Design browser.

2 Review the Cover Mold directory in Windows Explorer to identify the location of the _Combined CR1 and _Combined CV1 files. The _AsmContainer.iam file is a container file for the Combine Cores and Cavities command. This file contains information about the placement of the source cores and cavities, and other data and information. It is a temporary assembly that is not shown in the Mold Design or Model browser displays.

3 Review the Mold Design browser display and notice the Workpiece Pocket node. The Workpiece Pocket command is used to create an excavated pocket in the mold base, which holds the core and cavity. This command

does not create any files, but it does generate modifications to the mold base files.

4. The final node in this mold assembly represents the Cooling and Heating Components, which is comprised of the cooling channel and its components. In the Mold Design browser, expand the **Cooling & Heating Components** node to display all the channel segments and the components.
5. Review the Model browser display. In this assembly, the creation of the cooling and heating components generated 21 files, all which were created in the top-level assembly. The _CH file captures the cooling channel information, and the other 20 files represent the components. In Windows Explorer, review the following files associated with the cooling and heating components:

   - Cooling Components folder (1)
   - Cooling component part files(2)

6. In Windows Explorer, compress the directories to review the top-level Cover Mold directory. You have now examined all the files and directories that were generated in this mold assembly.

Previous (page 166) | Next (page 169)

Summary

In this tutorial, you learned about the files and directories that are created when you create a mold assembly with Inventor Mold Design. You reviewed the files that are created, and discovered how they are stored on your system. You also used the Mold Design and Model browser display views to see how all the files are structured in the mold assembly.

You learned how to:

- Identify the mold assembly files that are created and stored in the Project Location in Windows Explorer.

- Recognize the nodes that are added to the Mold Design browser display when features are added.
- Recognize the file hierarchy that is displayed in the Model browser for a mold assembly.

What next?

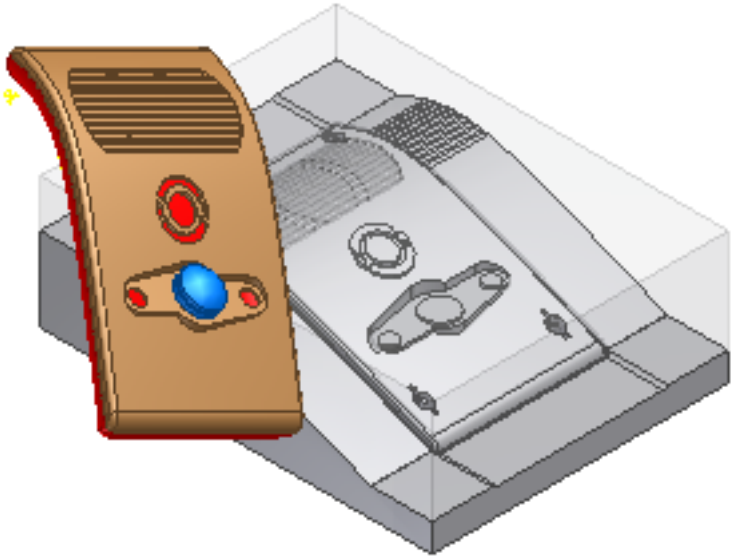
You can now review the files created in the other mold tutorials with a better understanding of the structure. You can also try any of the mold tutorials you have not completed yet.

Previous (page 168)

Resolving Plastic Part Design Issues

6

About this tutorial



Resolve tooling issues for plastic parts.

Category	**Tooling**
Time Required	60 minutes

Tutorial File Used	Mold.ipj Control_Button_Solid.ipt PFTutorial_Complete.ipt

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Before you begin, open and review the supplied plastic part. In this tutorial, you open an Inventor part file in the mold application. You use it to create the initial core and cavity of a mold design. To create the core and cavity, you require patching and runoff surfaces. To create these surfaces in the mold application, you modify some of the plastic part geometry. The edits do not change the shape of the part but are necessary for correct parting and patching surface generation.

You can complete the tutorial in segments by saving your work before you exit.

Objectives

- Insert one body from a multi-body plastic part in a mold assembly.
- Edit part features for use in the mold application.
- Generate patching and runoff surfaces.
- Generate a core and cavity.

Prerequisites

- Intermediate level of understanding part modeling.
- Completion of the General Mold Workflow Tutorial.
- Experience with Plastic Features and Multi-body part design.
- Activate the **Mold.ipj** project file.

Navigation Tips

- Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Next (page 173)

Getting Started

In this section, you review a multi-body part model and start a new mold design assembly.

1. On the ribbon, click **Get Started tab** ➤ Launch **panel** ➤ Projects  .
2. Navigate to the mold tutorials folder and select **Mold.ipj**.
3. Click **Open.**
4. Click **Configure Content Center Libraries**  .
   Make sure that the Inventor Mold libraries are selected.
5. Click **OK** to exit the Configure Libraries dialog box, and then click **Save** and **Apply** to activate the project if necessary.
6. Click **Done** to close the Projects dialog box.
7. On the ribbon, click **Get Started tab ➤ Launch panel** ➤ Open.
8. Navigate to the **Plastic Parts and Features** directory and open **PFTutorial_complete.ipt**. It is the part created in the plastic and multi-body tutorial.
9. Expand the Solid Bodies node in the Model browser and notice that this part was modeled using four separate solid bodies. Select each solid body in the Model browser to highlight them on the screen.
10. Leave the model open without changing anything. In later topics, you return to this window to work with the model.
11. On the Quick Access toolbar, click **New**  to start a new file.
12. Click the Metric folder in the Create New File dialog box, click **Mold Design (mm).iam**, and then click **Create**.
13. In the Create Mold Design dialog box, name the file **Tooling_Fix.iam**. Click **OK** to accept the default file location. The Mold Design environment is now available.

Previous (page 171) | Next (page 174)

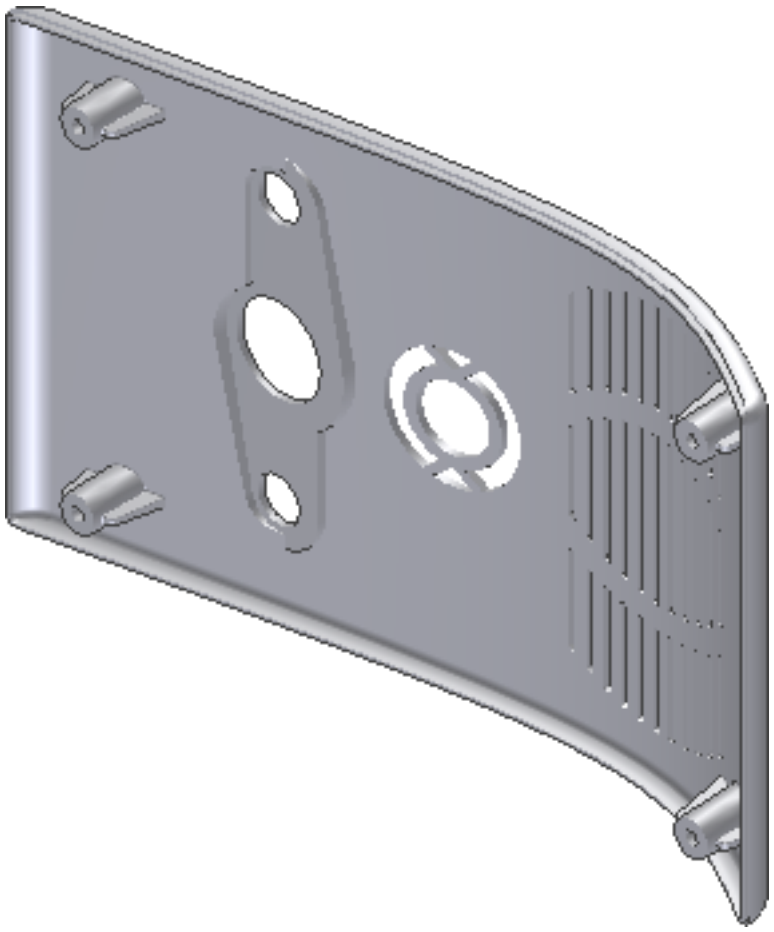
Adding a Body From a Multi-Body Part

In this section, you add a plastic part to a new mold design assembly. The plastic part contains multiple solid bodies and the required body must be derived into the mold assembly.

1. On the ribbon, click **Mold Layout tab ➤ Mold Layout panel** ➤ Plastic Part  .
2. In the Plastic Part dialog box, select the **PFTuorial_complete.ipt** part file and click **Open** to add the plastic part to the mold assembly.

   NOTE Click **Yes** to close the message box that states the file is not in the active project path.

3. A warning message appears indicating there is more than one solid body in the part. Click **Yes** to continue.
4. Right-click PFTutorial_complete_1 in the left frame and select **Rename**. Enter **PFTuorial_Top_Shell** as the new name.
5. Clear the check marks near Solid4, Solid5, and Solid7. The only Solid Body selected in the right-hand frame is **Solid2**.
6. Click **OK** to derive the plastic part.
7. Left-click in the graphics window to place the part with the default alignment option. The plastic part appears, as shown in the following image.
8. Alternately, if you have enabled **Prompt for File Names**, click **OK** to accept the defaults in the **File Naming** dialog box to place the part.



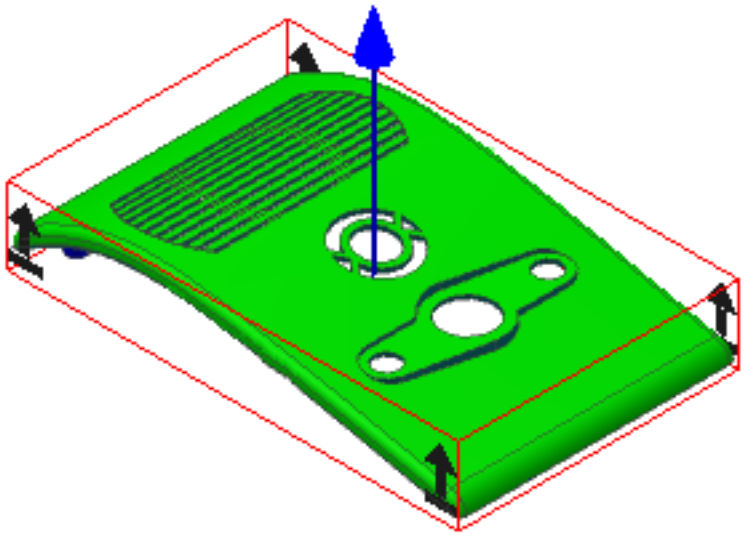
Previous (page 173) | Next (page 175)

Defining the Mold Parameters

In this section, you define the mold orientation and material type.

1 The default orientation of the plastic part is incorrect to obtain the required opening direction. On the ribbon, click **Mold Layout tab ➤ Mold Layout panel ➤** Adjust Orientation .

2 Maintain **Align with axis** as the default Method for adjustment and ensure that **Z Axis of Rotation** is enabled. Select the flat circular face of any of the four bosses. Select **Flip moldable part**. Make sure that the part matches the following image.



Click **Done** to close the Adjust Orientation dialog box.

3 On Mold Layout tab, click **Select Material** .

4 From the list of Manufacturers, select **CMOLD Generic Estimates**.

5 From the list of Trade names, select **ABS Generic Estimates**.

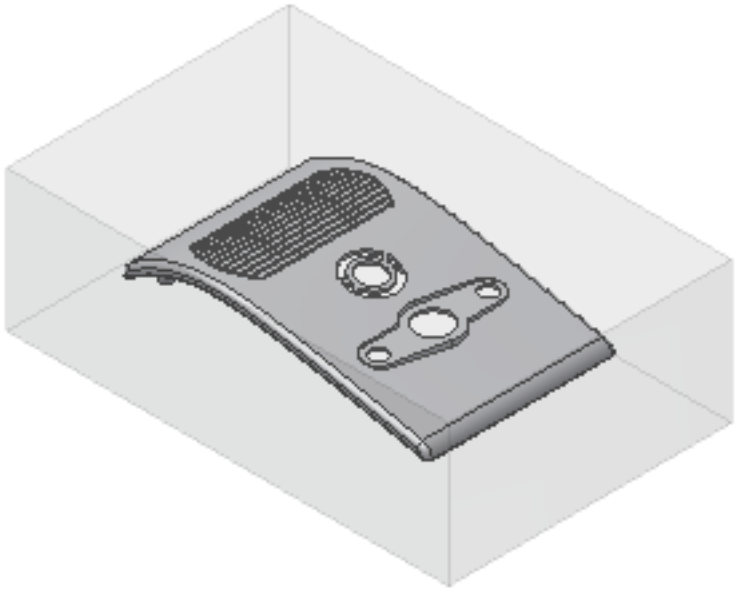
6 Click **OK** to accept the selected material and close the Select Material dialog box.

Previous (page 174) | Next (page 177)

Defining the Workpiece

In this section, you define the workpiece used to create the core and cavity for the mold.

1. On the ribbon, click **Core/Cavity tab ➤ Parting Design panel** ➤ Define Workpiece Setting  .
2. Maintain the default workpiece type. In the **X_total** field, enter **250 mm**. In the **Y_total** field, enter **160 mm**. In the **Z_total** field, enter **80 mm**.
3. Click **OK** on the Define Workpiece Setting dialog box to accept the parameters and close the dialog box. The workpiece appears as shown in the following image.
4. Alternately, if you have enabled **Prompt for File Names**, click **OK** to accept the defaults in the **File Naming** dialog box to place the workpiece.



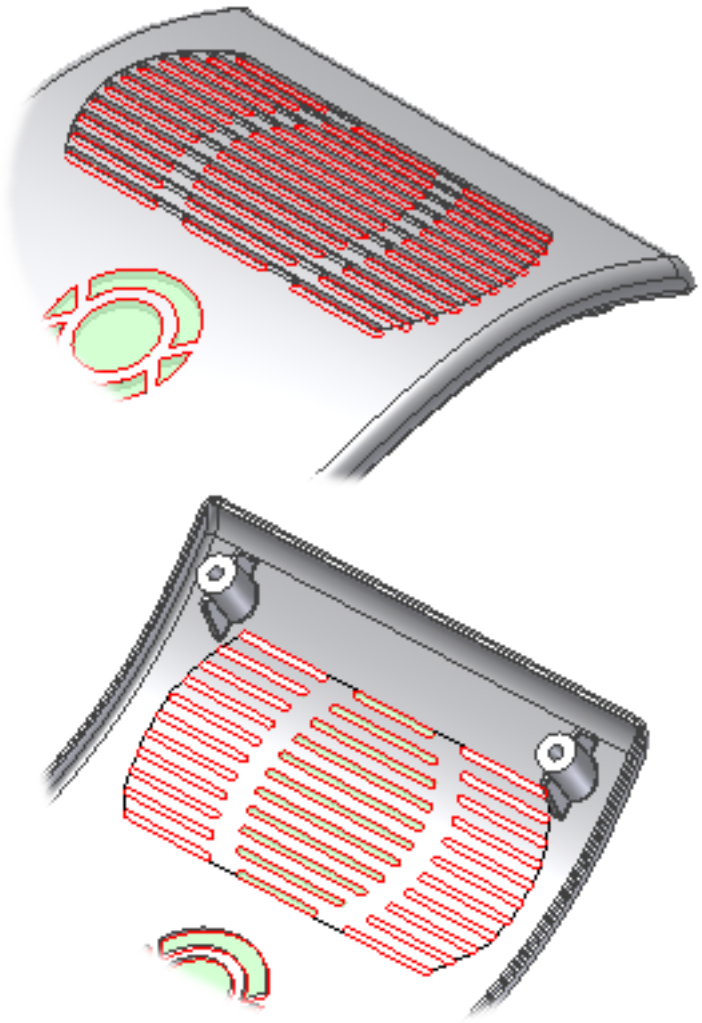
Previous (page 175) | Next (page 178)

Creating the Patching and Runoff Surfaces

In this section, you create the patching and runoff surfaces for the mold assembly. Once created it is obvious that there are issues with creating the patching and runoff surfaces caused by the geometry in the model.

1. On the ribbon, click **Core/Cavity tab ➤ Parting Design panel** ➤ Create Patching Surface .

2. On the Create Patching Surface dialog box, select Auto Detect to create the internal patches automatically. The system detects as many internal patches as possible and lists them by name in the Creating Patching Surface dialog box. The patches are highlighted on the model in green. Rotate the model. Notice that the center section of the grill contains valid patches, but the two outer areas do not as shown in the following image.



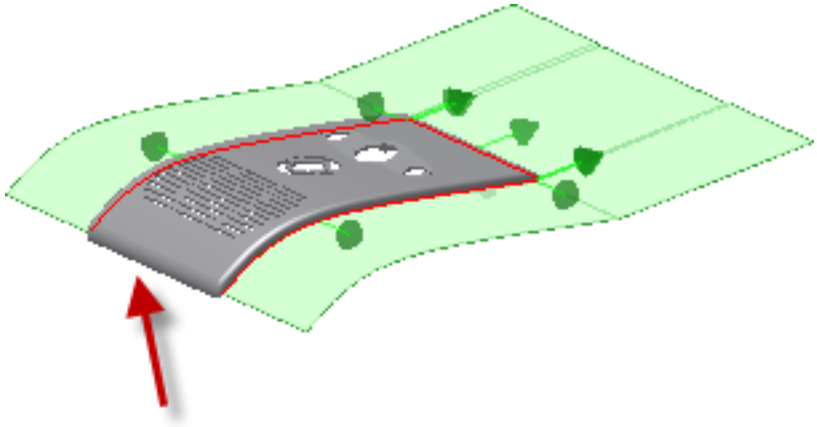
NOTE The missing surface patches can be created by deleting the bad patches on the grill and then using **Click to add** to select the edges manually. In the next section, you return to the part model and make an edit that allows automatic patching of all grill openings.

3 Click **Cancel** on the Create Patching Surface dialog box to exit without creating the patching surfaces.

4 On the ribbon, click **Core/Cavity tab ➤ Parting Design panel** ➤ Create Runoff Surface .

5 On the Create Runoff Surface dialog box, select Auto Detect to create the runoff surfaces automatically.

6 The preview shows the runoff surface is not complete. Examine the preview and notice the change in curvature.



NOTE The missing surface could be created manually by selecting edges. However, the runoff preview points out that a slight design change provides better parting results. In the next section, you return to the part model and make an edit to provide better parting results and automatic runoff surface generation.

7 Click **Cancel** on the Create Runoff Surface dialog box to exit without creating the runoff surfaces.

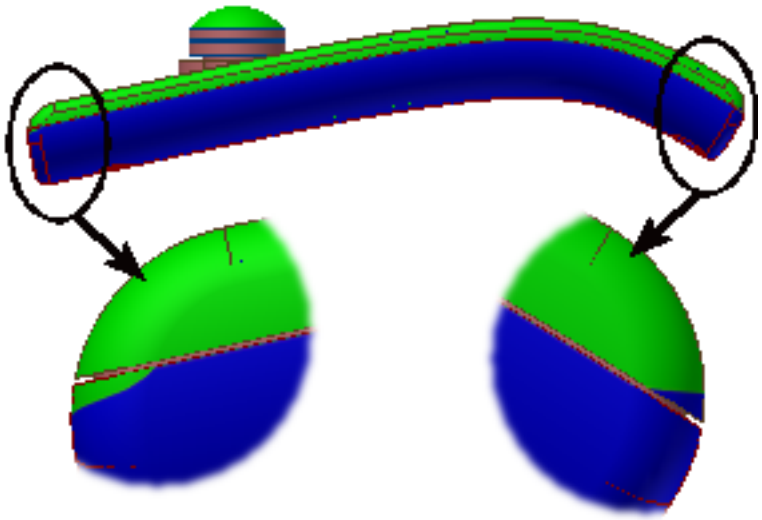
8 Click **Finish Core/Cavity** and save the assembly.

Previous (page 177) | Next (page 181)

Perform a Draft Analysis on the Plastic Part Model

In this section, open the source part model and perform a Draft analysis to identify curvature issues in the model.

1. Open **PFTutorial_complete.ipt**. If the window tab is still open, activate it.
2. On the ribbon, click **Inspect tab ➤ Analysis panel ➤ Draft** .
3. Enter **-0.1 deg** and **0.1 deg** as the range for the analysis.
4. Expand the Origin browser node and select the Y axis as the pull direction. Click **OK** to perform the analysis.
5. Rotate the model and zoom in to see the ends of the model, as shown in the following image.



Notice the draft analysis shows an undercut on each end of the model. It must be fixed to create a proper runoff surface.

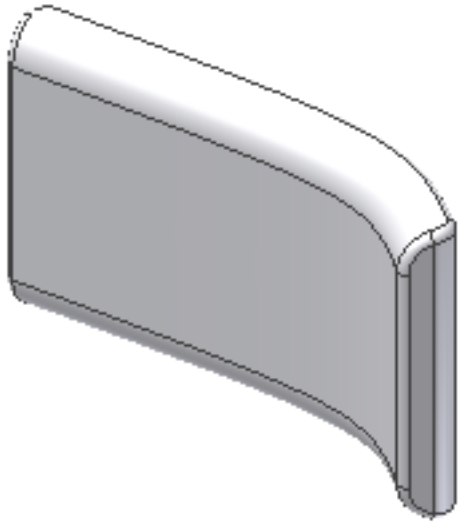
6 On the ribbon, click **View tab ➤ Visibility panel ➤ Analysis** to turn off the draft analysis display.

Previous (page 178) | Next (page 182)

Add a Boundary Patch to the Plastic Part Model

In this section, you add a planar surface to the plastic part model. It is used as a reference for the model changes required to eliminate the undercut.

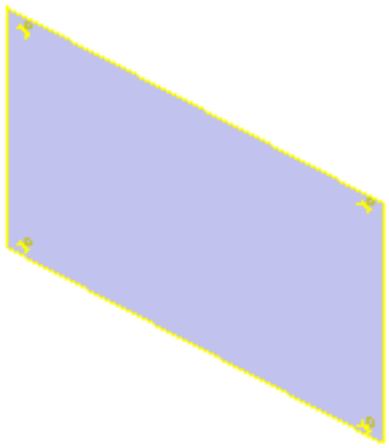
1 Drag the **End of Part** marker from the bottom of the Model browser list to just after the Work Point4 feature. The model appears as shown in the following image.



2 Turn off the visibility of **Solid1** in the Solid Bodies node.

3 Turn on the visibility of **Work Point1**, **Work Point2**, **Work Point3**, and **Work Point4**.

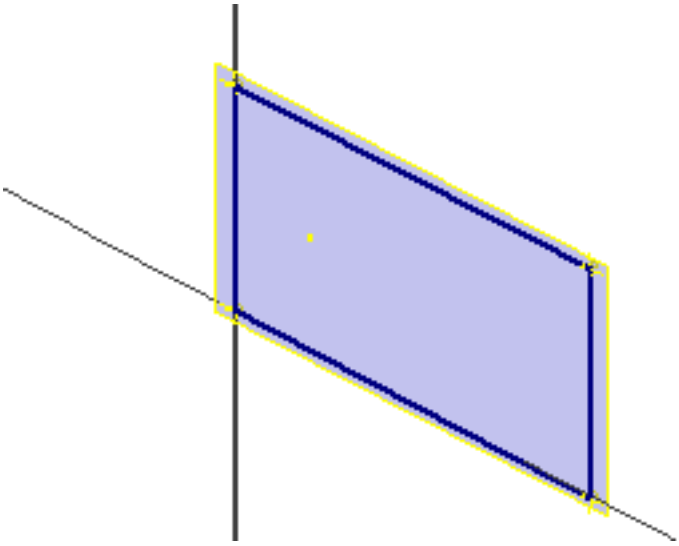
4 On the ribbon, click **3D Model tab ➤ Work Features panel ➤ Work Plane**. On the browser, select **Work Point1**, **Work Point2**, and **Work Point3** to create a plane. The model appears as shown in the following image.



5 On the ribbon, click **3D Model tab ➤ Sketch panel ➤ Create 2D Sketch** and select the edge of the plane you created as the sketch plane reference.

6 On the ribbon, click **Sketch tab ➤ Draw panel ➤ Project Geometry** and select **Work Point1**, **Work Point2**, **Work Point3**, and **Work Point4**.

7 Sketch four lines connecting the projected points, as shown in the following image.



8 Finish the sketch.

9 On the ribbon, click **3D Model tab ➤ Surface panel ➤ Patch** and select the closed boundary loop created by the sketch. Click **OK**.

10 Turn off the visibility of **Work Point1**, **Work Point2**, **Work Point3**, **Work Point4**, and the **Work Plane.** The new surface is the only feature displayed. The surface was added to the model to use as a reference for new work points. The new work points properly locate the bosses in the modified model.

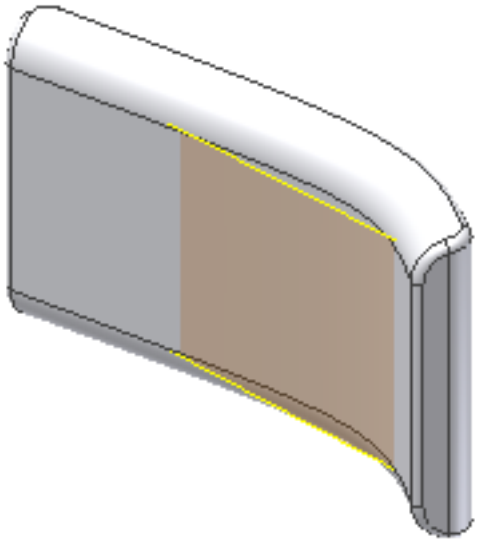
Previous (page 181) | Next (page 184)

Move Bodies in the Plastic Part Model

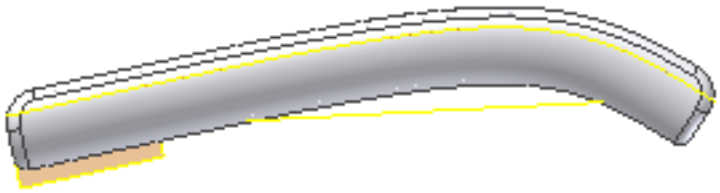
In this section, you use the Move Bodies command to rotate the solid body and surfaces by 10 degrees. It adjusts the parting surface for the model and avoids the undercuts.

1 Drag the **End of Part** marker after the Shell1 feature.

2 Return to the Solid Bodies node and turn on the visibility of **Solid1**, as shown in the following image.



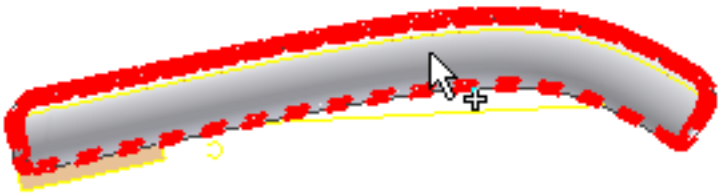
3 Reorient the model to a Front view. Turn on the visibility of the **Parting Surface** and **BatterySplitSurface**. The model appears as shown in the following image.



4 On the ribbon, click **3D Model tab ➤ Modify panel ➤ Move Bodies**

.

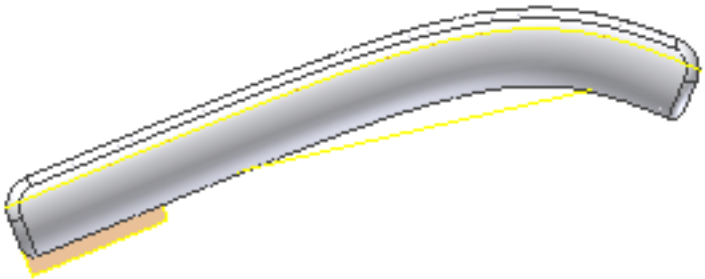
5 Select **Solid1** from the Solid Bodies browser node or in the graphics window as shown in the following image.



6 Click Free Drag in the Move Bodies dialog box to activate the drop-down arrow. Scroll the list and select Rotate about line in the drop-down list. Select the Z axis in the Origin node as the rotation axis. Enter **10 deg** in the Angle box.

7 Pick the Bodies select arrow again and pick **Srf1**, **Srf2**, and **Srf5** in the Surface Bodies folder.

8 Click **OK**. The model appears as shown in the following image.



9 Turn off the visibility of the **Parting Surface**, **BatterySplitSurface**, and **Solid1**.

10 Reorient the model to the Home view.

11 On the ribbon, click **3D Model tab ➤ Work Features panel ➤ Point** and select one of the four corners of the boundary patch to create a work point. Repeat the command until there are work points at all four corners of the boundary patch. These points are required because the Move Bodies command cannot rotate points. The original reference points were used to locate the bosses in the solid body. Because the points were not rotated, the bosses are incorrectly positioned. Using these new points the bosses can be redefined for proper placement.

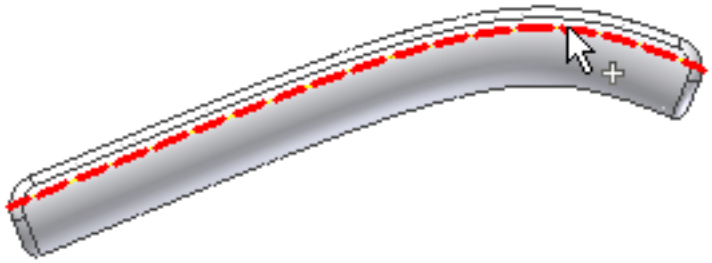
12 Turn off the visibility of the **Boundary Patch**.

13 Turn on the visibility of **Solid1** and **PartingSurface**.

14 On the ribbon, click **3D Model tab ➤ Modify panel ➤ Move Bodies**

.

15 Select **Srf1** from the Surface Bodies node or directly from the model to satisfy the Bodies selection, as shown in the following image.



16 Maintain the Free Drag option for moving the bodies and enter **0 mm** as the X offset and **-0.25 mm** as the Y Offset. Notice the small change in the position of Solid1 relative to the PartingSurface. This surface was moved down to correct the parting line for the model.

17 Click **OK**.

18 Turn off the visibility of **PartingSurface**.

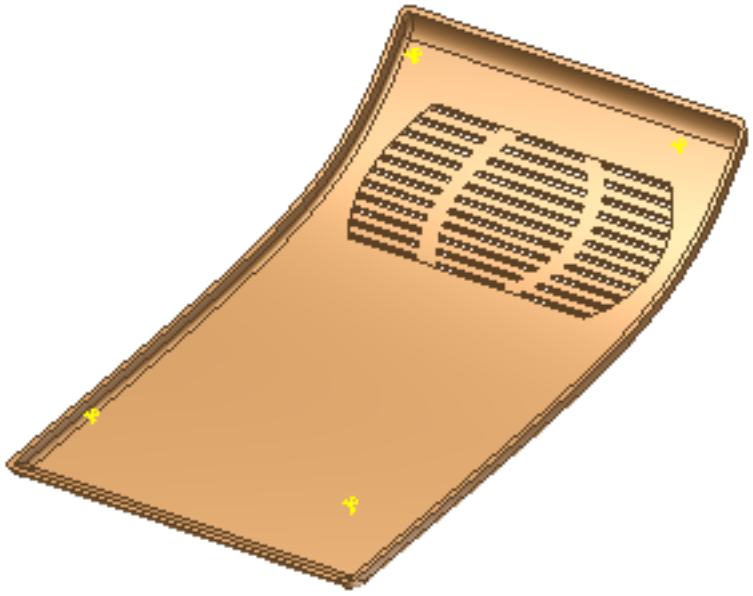
Previous (page 182) | Next (page 187)

Delete and Heal a Face on the Plastic Part Model

In this section, you delete a face on the plastic part model. It resolves the design issue that prevented the automatic patching surfaces from being created for the grill. When the grill was created, multiple faces were produced. The edges of the grill holes do not belong to the same face and therefore are not picked up with the automatic patching. Once the face is deleted and healed, resolve a failure to a rule-based fillet. Removing the face caused the failure.

1 Drag the **End of Part** marker after the Grill1 feature.

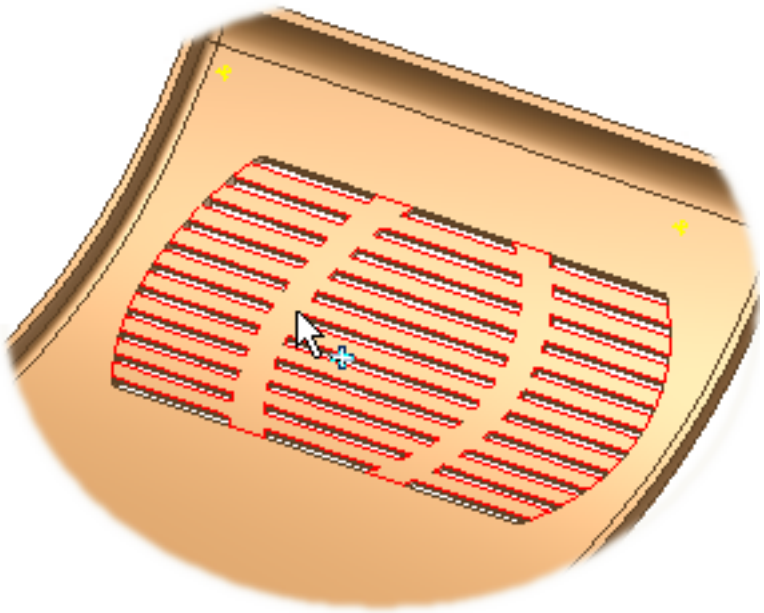
2 Turn off the visibility of the **Solid3**. Rotate the model as shown in the following image.



3 On the ribbon, click **3D Model tab ➤ Surface panel ➤ Delete Face** .

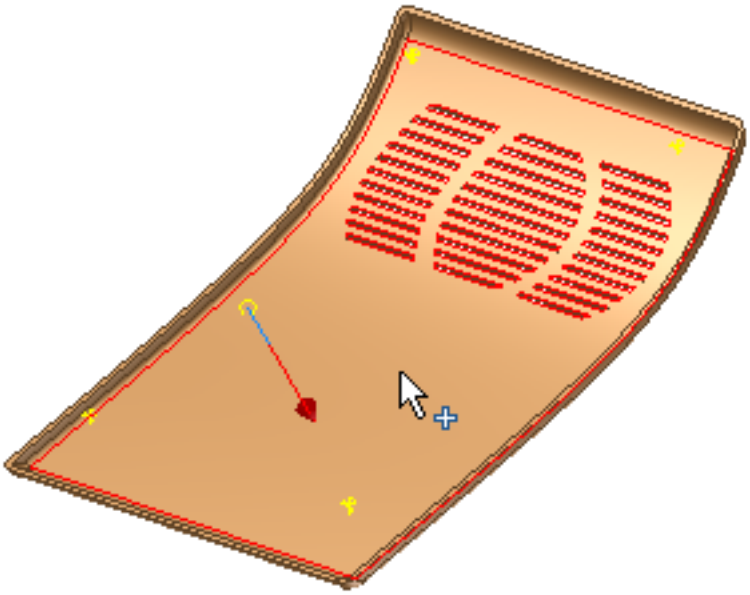
4 Click **Heal** in the Delete Face dialog box.

5 Zoom in on the model and select the face shown in the following image as the face to delete. When the process finishes, select the inside face again and notice there is only one interior face.



6 Drag the **End of Part** marker after the RuleFillet1 feature. The feature fails indicating that the feature needs at least one valid rule. Click Accept.

7 Double-click RuleFillet1 to access the Rule Fillet dialog box. Notice that there is no surface selected. It is because the surface was deleted. Select the surface shown in the following image as the reference surface. Click **OK** to complete the feature. The failure has been resolved.



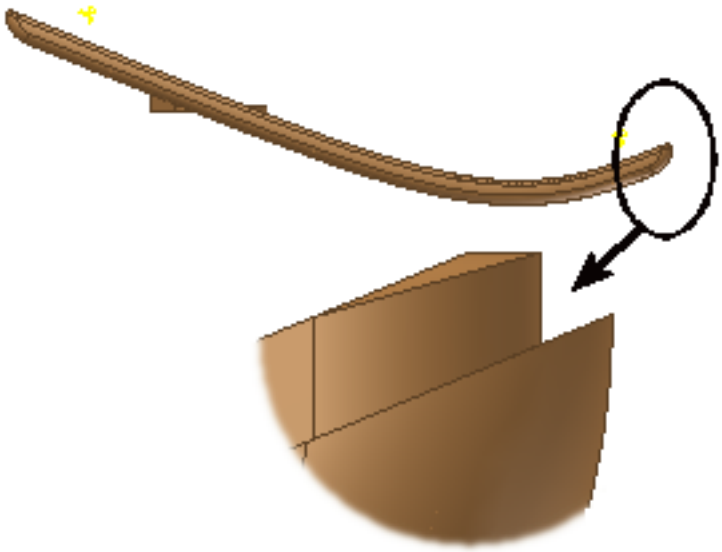
Previous (page 184) | Next (page 190)

Change the Lip of the Plastic Part Model

In this section, you modify the size of the lip features to adjust the parting surface. It avoids small undercuts in the model.

NOTE Some of the operations in the next sections take time for the system to update.

1. Drag the **End of Part** marker after the Lip1 feature.
2. Reorient the model to the Back view and zoom in as shown in the following image.



3 On the ribbon, click **View tab ➤ Visibility panel ➤ Analysis** to turn on the results display for the Draft analysis that was created earlier. It helps identify the undercut.

4 Double-click Lip1 to access the Lip dialog box.

5 Select the Lip tab to access the size parameters for the lip geometry.

6 Change the 0.5-mm dimension to **0.7 mm**. Click **OK**.

7 Drag the End of Part marker after the Lip2 feature.

8 Turn on the visibility of the **Solid3**. Notice that the separation between the two bodies is now a little larger.

9 On the ribbon, click **View tab ➤ Visibility panel ➤ Analysis** to turn off the results display for the Draft analysis.

Previous (page 187) | Next (page 192)

Change the Boss Features

In this section, you redefine the boss features to set the references for the changes that were made in the model.

1. Drag the End of Part marker after the **Boss1** feature.
2. Reorient the model to the Front view as shown in the following image.

3. Turn on the visibility of **Work Point1**, **Work Point2**, **Work Point3**, and **Work Point4**.
4. Double-click Boss1 to access the Boss dialog box. Notice that the boss no longer intersects with the model now that the changes have been made using the Move Bodies command.
5. Ensure that the Centers command is active in the dialog box. Select **Work Point1**, **Work Point2**, **Work Point3**, and **Work Point4** in the browser to clear them from the selection set.
6. Ensure that the Centers command is still active. Select **Work Point7**, **Work Point8**, **Work Point9**, and **Work Point10**. These points align the boss locations with the changes made to the model.
7. Click **OK**. The boss feature is now properly located relative to the existing geometry, as shown in the following image.

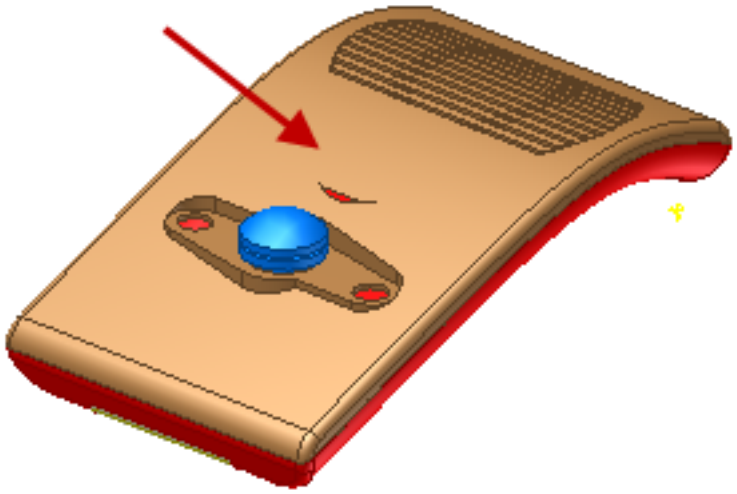
8. Drag the **End of Part** marker after the Boss2 feature. The model fails due to a modeling failure while trimming. Click **Accept**.
9. Double-click Boss2 to access the Boss dialog box.
10. Ensure that the Centers command is active in the dialog box. Select **Work Point1**, **Work Point2**, **Work Point3**, and **Work Point4** in the browser to clear them from the Centers selection set.
11. Ensure that the Centers command is still active. Select **Work Point7**, **Work Point8**, **Work Point9**, and **Work Point10** to align their location with the model changes.
12. Click **OK**. The boss feature is now properly located relative to the modified geometry and the failure is resolved.

Previous (page 190) | Next (page 194)

Modify the Control Button to Cut the Model

In this section, you open the Control_Button_Solid.ipt component and modify its depth so that it cuts through the entire solid body.

1 Drag the **End of Part** marker after the last feature, Revolution1. The model appears as shown in the following image. Notice that the cutout in the center of the model is no longer cutting through the entire model.



Right-click Control_Button_Solid.ipt in the Model browser and select **Open Base Component**.

2 Double-click Extrusion1 and change the extrusion extent to **20 mm**. Click **OK**. Save the model and close it. Return to the PFTutorial.ipt model if not already active.

3 On the Quick Access toolbar, click **Local Update**  . The model appears as shown in the following image.

4 Save the file.

Previous (page 192) | Next (page 195)

Create the Runoff, Parting Surfaces, and Core and Cavity

In this section, you return to the mold assembly and automatically create the patching and runoff surfaces on the modified model geometry. To complete the section you create the core and cavity based on the patching and runoff surfaces.

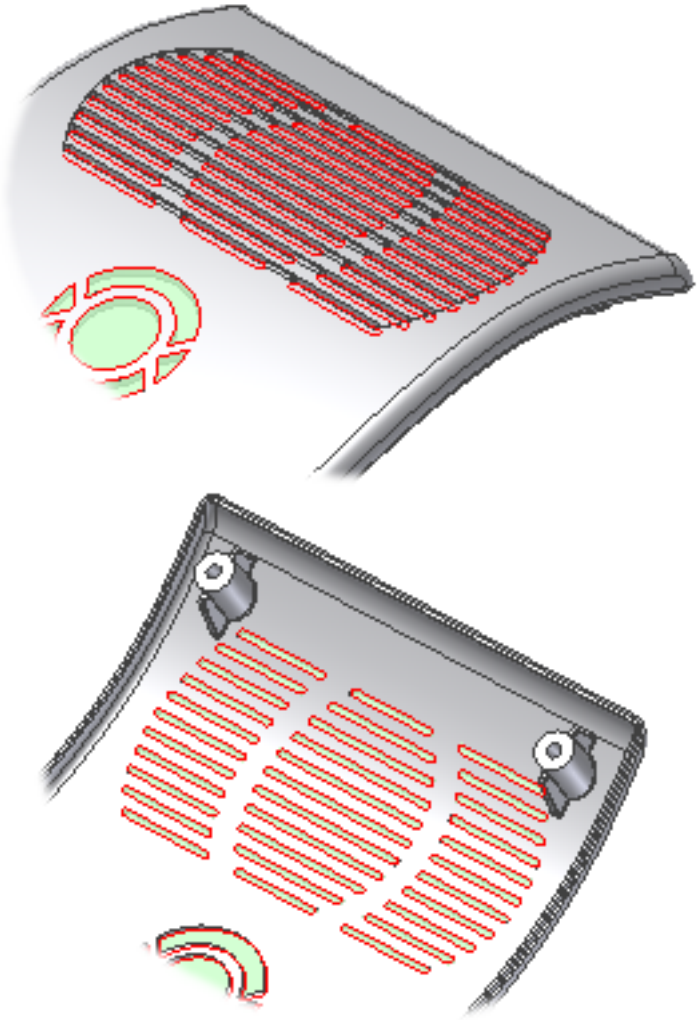
1 Activate the mold assembly.

2 On the Quick Access toolbar, click **Local Update** .

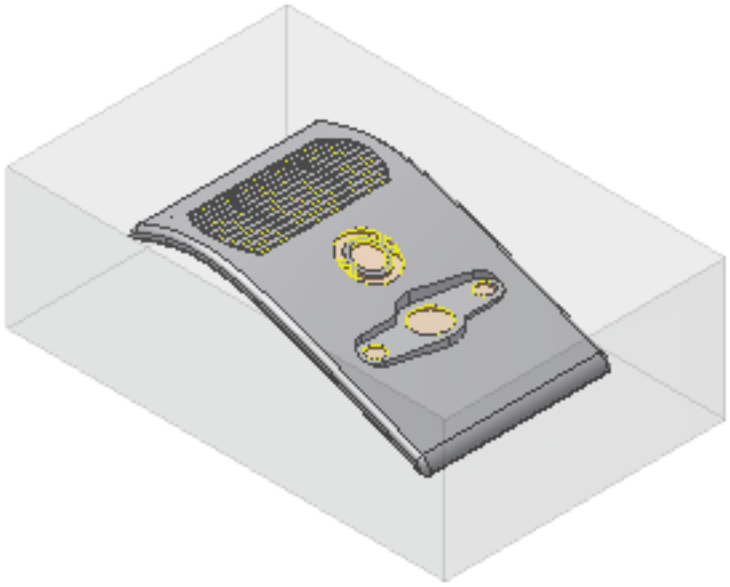
3 On the ribbon, click **Core/Cavity tab ➤ Parting Design panel ➤ Create Patching Surface** .

NOTE Click Core/Cavity in the Mold Layout panel to activate the Core/Cavity tab. You can also double-click the plastic part in the Mold Design browser.

4 On the Create Patching Surface dialog box, select Auto Detect  to create the internal patches automatically. Now that the model has been modified the system automatically detects all the surfaces for the Grill, as shown in the following image.



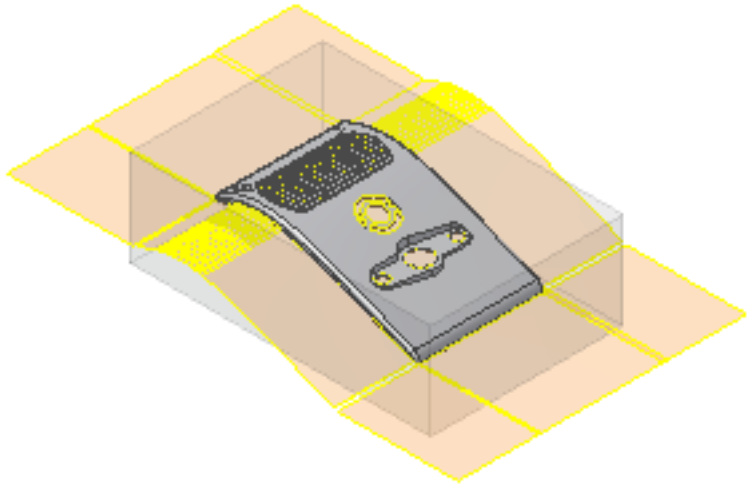
5 Click **OK**. The model appears as shown in the following image.



6 On the ribbon, click **Core/Cavity tab ➤ Parting Design panel** ➤ Create Runoff Surface .

7 On the Create Runoff Surface dialog box, select Auto Detect to create the runoff surfaces automatically.

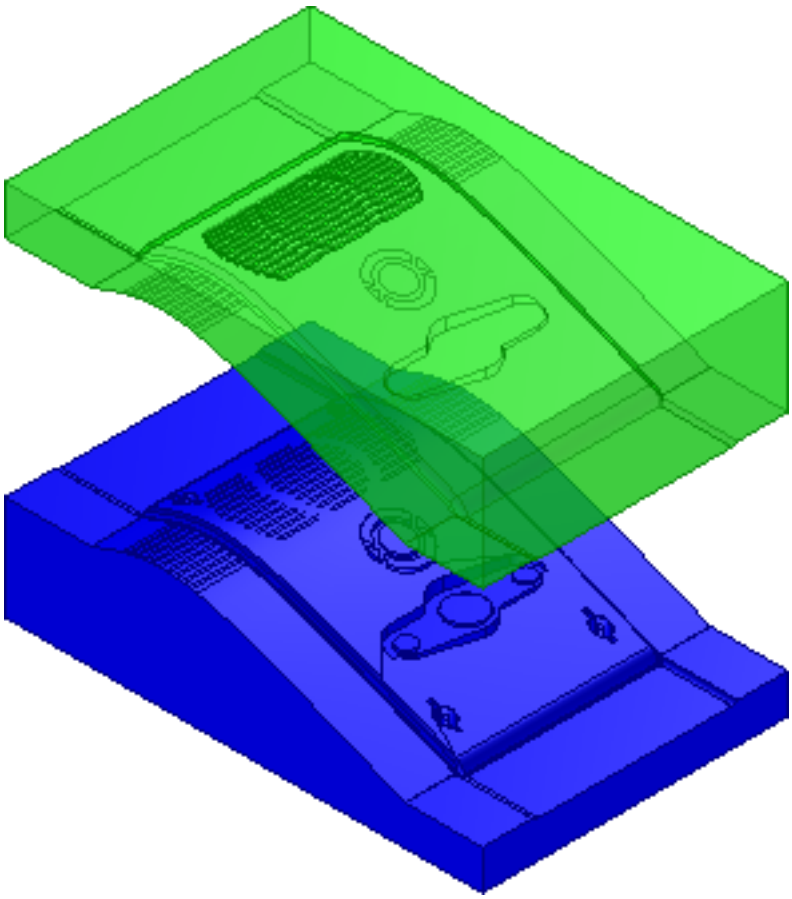
8 Click **OK** to generate the runoff surfaces. The model appears as shown in the following image.



9 On the ribbon, click **Core/Cavity tab ➤ Parting Design panel ➤ Generate Core and Cavity** .

10 Click **Preview/Diagnose** to perform parting diagnostics and enable the preview controls.

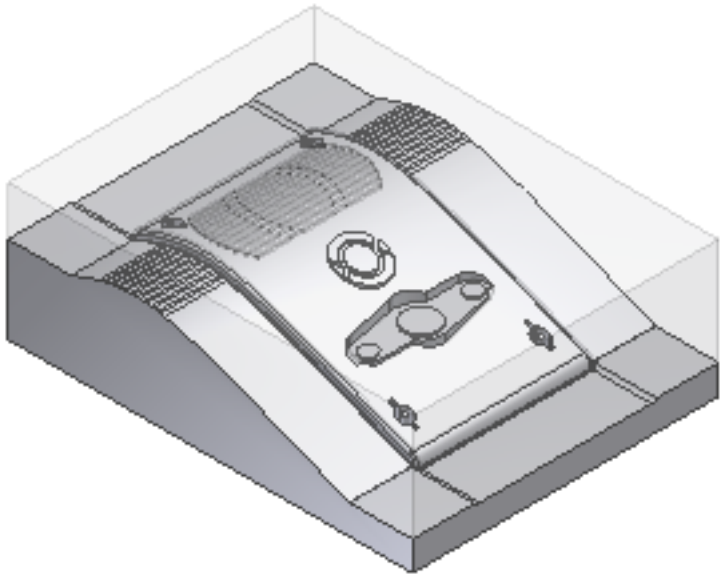
11 Use the **Body Separation** slider in the Preview tab to examine the core and cavity before creation as shown in the following image.



NOTE The Parting Diagnostics tab report does not indicate any serious errors.

12 Click **OK** to generate the core and cavity. The bodies are generated as shown in the following image.

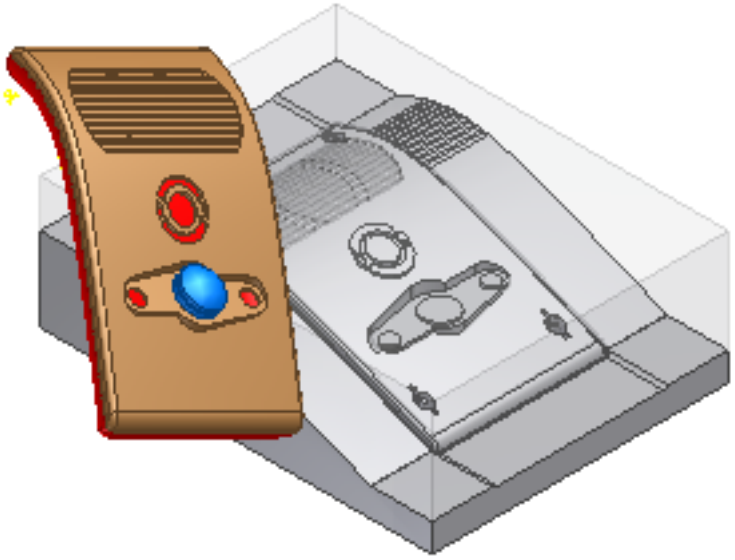
13 Optionally, if **Prompt for file names** is enabled, click **OK** in the File Naming dialog box. The core and cavity generates with the default settings.



14 Save the file.

Previous (page 194) | Next (page 201)

Summary



In this tutorial, you created a mold assembly using an Inventor multi-body part file. When working on the parting design, you recognized part modeling issues which prevented patching and runoff surfaces from being created. Working in the plastic multi-body part file, you redesigned the model for better results in the mold application.

You also learned how to:

- Place a single body from a multi-body part file in a mold assembly.
- Use draft analysis tools to analyze the model.
- Resolve body errors in a part model.
- Generate automatic patching and runoff surfaces.
- Generate a core and cavity with no errors.

What Next?

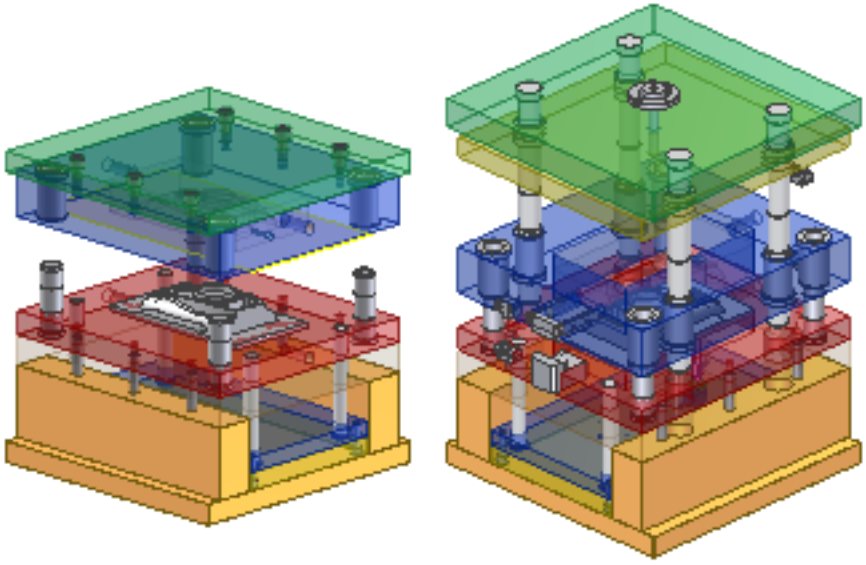
You can add the feeding and cooling systems, place the mold base, and add additional mold components to complete the mold assembly design.

Previous (page 195)

Mold Tutorial Kinematics

7

About this tutorial



Work with mold base kinematics.

Category	**Tooling**
Time Required	60 minutes
Tutorial File Used	Mold_Kinematics.ipj

NOTE Click and read the required Tutorial Files Installation Instructions at*http://www.autodesk.com/inventor-tutorial-data-sets* . Then download the tutorial data sets and the required Tutorial Files Installation Instructions, and install the datasets as instructed.

Objectives

- Examine the default positional representations created by 2 and 3 plate mold bases.
- Learn how to activate positional representations.
- Understand the constraints that control positional representations.
- Drive positional representations.

Prerequisites

- Intermediate level of understanding part modeling.
- Completion of the General Mold Workflow Tutorial.
- Experience with Plastic Features and Multi-body part design.
- Activate the **Mold_Kinematics.ipj** project file.
- Ensure that the Mold library content is enabled in the project file.

Navigation Tips

- Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

Next (page 204)

Getting Started

In this section, you review a multi-body part model and start a new mold design assembly.

1. On the ribbon, click **Get Started tab** ➤ Launch **panel** ➤ Projects  .
2. Navigate to the Tutorials ➤ Mold ➤ Mold Kinematics folder, and select **mold_kinematics.ipj**.

3 Click **Open.**

4 Click **Configure Content Center Libraries**  .
Make sure that the Inventor Mold libraries are selected.

5 Click **OK** to exit the Configure Libraries dialog box and then click **Save** and **Apply** to activate the project if necessary.

6 Click **Done** to close the Projects dialog box.

7 On the ribbon, click **Get Started tab ➤ Launch panel ➤** Open.

8 Navigate to the **Mold Kinematics** directory and open **Mold_Kinematics.iam**. The assembly file already contains a plastic part, a core/cavity, and patching and runoff surfaces.

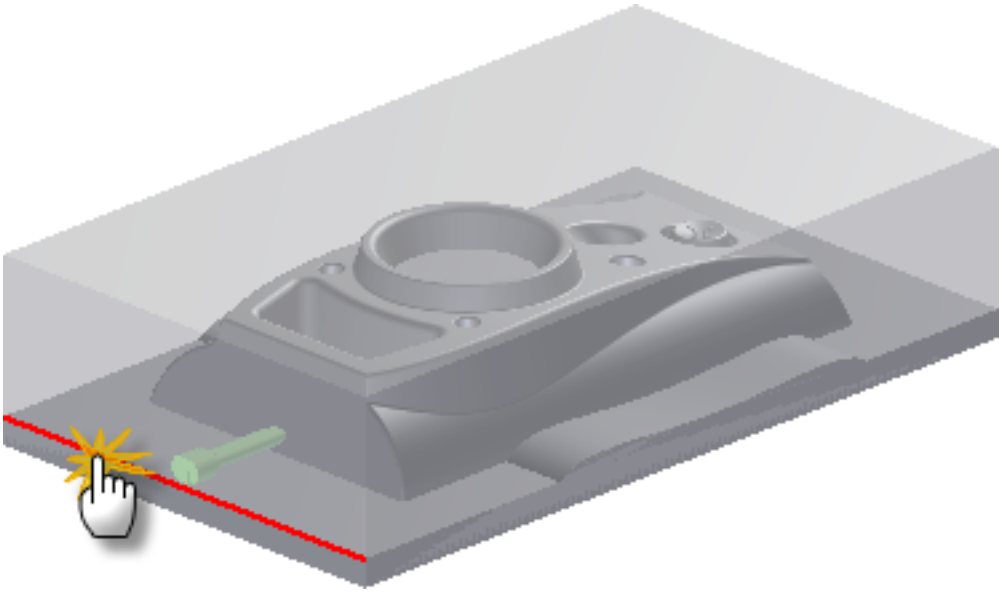
Previous (page 203) | Next (page 205)

Place a Two-Plate Mold Base

In this section, you place a two-plate mold base.

1 On the ribbon, click **Mold Assembly tab ➤ Mold Assembly panel ➤ Mold Base**  .

2 On the Mold Base dialog box, make sure that the Placement selection is active. Select the edge that separates the core and cavity to locate the Placement Ref Point as shown in the following image.

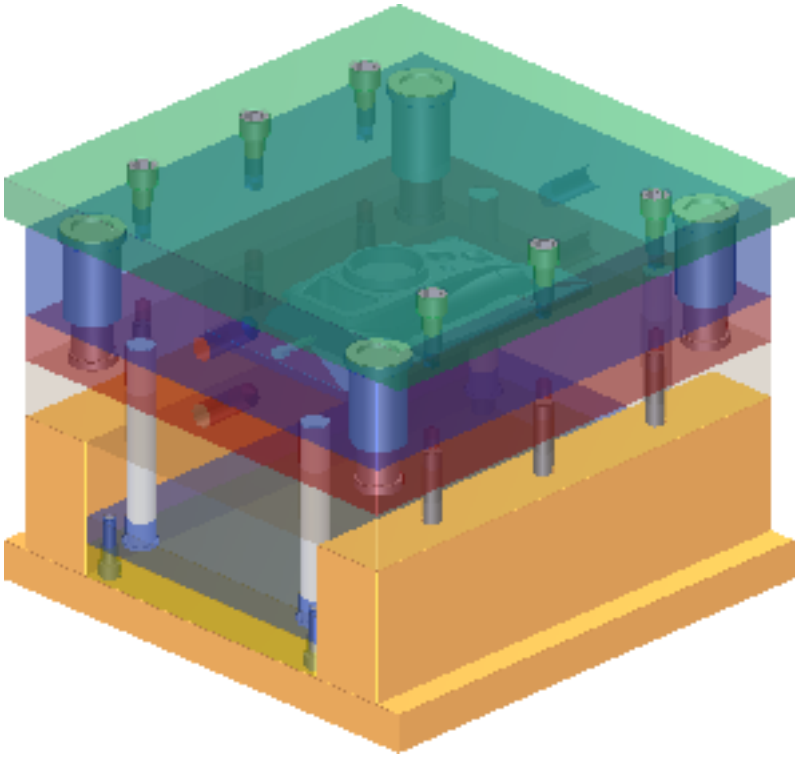


3 On the Mold Base dialog box, specify the Vendor and Type as **FUTABA-SA-S.**

4 Set the Size to **400 x 450 mm**.

5 Click **OK** to generate the mold base as shown in the following image.

6 Optionally, if you have enabled Prompt for File Names, click **OK** in the File Naming dialog box to accept the defaults and generate the mold base.



Previous (page 204) | Next (page 207)

Two-Plate Mold Base Kinematics

In this section, we examine the kinematic constraints of a two-plate mold base. A two-plate mold base contains two key constraints:

- **PL1** between the core plate (BP and the cavity plate (AP).
- **Ejection** between the ejector plate (EP) and the bottom clamp plate (BCP).

1. Change to the Model browser and expand the Mold_Kinematics _MB (Mold Base) assembly node.

   TIP Choose the drop-down at the top of the browser to switch between Mold Design, Model, and Representations browser display.

2. Expand Mold_Kinematics_MB_SA-S-AP to view the constraints

Notice the mate constraint named **PL1**. Right-click PL1 and select **Other Half** in the context menu to locate the second half of the constraint.

The browser exposes the second half of the constraint contained in Mold_Kinematics_MB_SA-S-BP.

3 Expand Mold_Kinematics_MB_SA-EP to view the constraints.

Notice the mate constraint named **Ejection**. Right-click Ejection and select **Other Half** in the context menu.

The browser exposes the second half of the constraint contained in Mold_Kinematics_MB_SA-BCP.

4 On the Model browser, expand the Representations node.

5 Expand the Position node. Notice there are five representations; Master, Product Open, Ejection, Close, and Free Drag.

- Right-click the Product Open node and click Activate, Observe the position change.
- Right-click the Ejection node and click Activate. Observe the position change.
- Right-click the Close node and click Activate. Observe the position change.
- Right-click the Free Drag node and click Activate. Drag the movable components on the graphics screen.

6 Right-click the Master node and click Activate.

7 **NOTE** Edits to the mold are disabled unless the mold is in the closed position and Master representation.

8 Enable the Representations browser.

9 Expand the Positional Representations node. Notice that the representations are also located in this node.

10 Make sure that the Master representation is active and then activate the Mold Design browser.

11 Close the file without saving your changes and reopen it.

In the next section, you insert a 3-plate mold base.

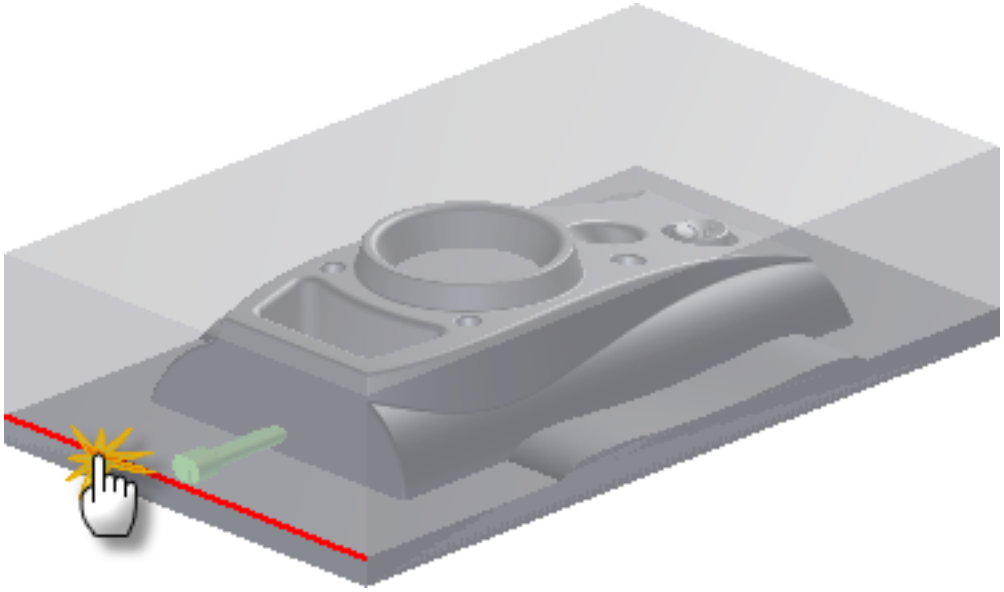
If you saved the file, on the Mold Design browser, right-click the Mold Base node and choose Delete.

Previous (page 205) | Next (page 209)

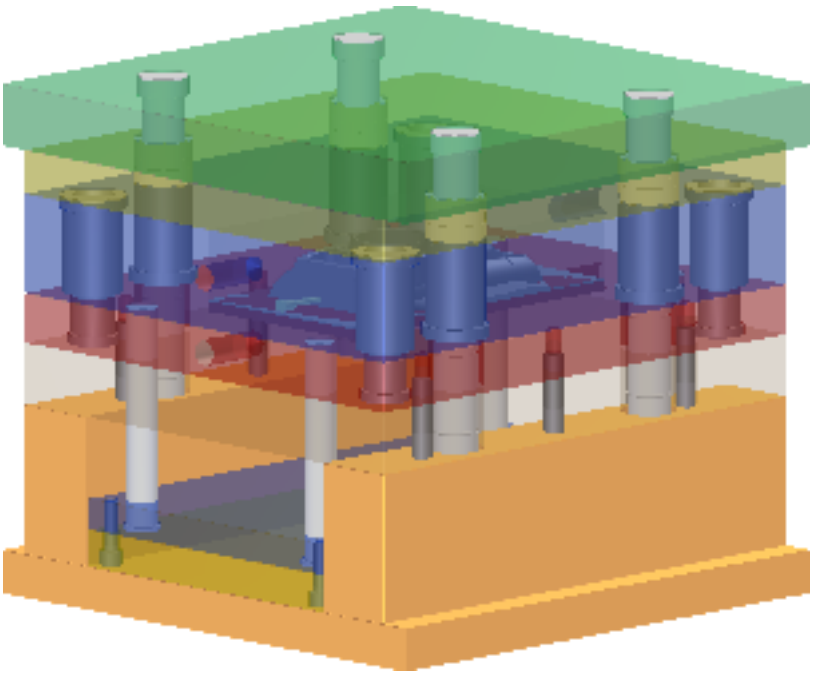
Place a Three-Plate Mold Base

In this section, you place a three-plate mold base.

1. On the ribbon, click **Mold Assembly tab ➤ Mold Assembly panel ➤ Mold Base** .
2. On the Mold Base dialog box, make sure that the Placement selection is active. Select the edge that separates the core and cavity to locate the Placement Ref Point as shown in the following image.



3. On the Mold Base dialog box, click the Vendor and Type drop down. Select **Futaba** as the Vendor, and **Three Plate Mold Bases** for the Category.
4. Select FUTABA-DA-S-IH from the list.
5. Set the Size to **400 x 450 mm**.
6. Click **OK** to generate the mold base as shown in the following image.
7. Optionally, if you have enabled Prompt for File Names, click **OK** in the File Naming dialog box to accept the defaults and generate the mold base.



Previous (page 207) | Next (page 210)

Three-Plate Mold Base Kinematics

In this section, we examine the kinematic constraints of a three-plate mold base. A three-plate mold base contains four key constraints:

- **PL1** = Constraint between runner stripper plate (RSP) and cavity plate (AP)
- **PL2** = Constraint between runner stripper plate (RSP) and top clamping plate (TCP).
- **PL3** = Constraint between core plate (BP) and cavity plate (AP).
- **Ejection** = Constraint between the ejector plate (EP) and bottom clamping plate (BCP).

1 Change to the Model browser and expand the Mold_Kinematics _MB (Mold Base) assembly node.

TIP Choose the drop-down at the top of the browser to switch between Mold Design, Model, and Representations browser display.

2. Expand Mold_Kinematics_MB_DA-IH-RSP to view the constraints.

   Notice the mate constraint named **PL1**. Right-click PL1 and select **Other Half** in the context menu to locate the second half of the constraint.

   The browser exposes the second half of the constraint contained in Mold_Kinematics_MB_DA-IH-AP.

3. Right-click **PL1** in either location and select **Drive Constraint**.
4. On the Drive Constraint dialog box, leave the **Start** value as **0**, and the **End** value as **150 mm** and select the **Forward** play button.
5. Select the **Reverse** play button to return the mold to the closed position.
6. Click **OK** to close the dialog box.

   NOTE If you close the dialog box before returning to the closed position, the active constraint value is changed to 150 mm.

7. Notice the **PL2** constraint is also located under Mold_Kinematics_MB_DA-IH-RSP. Right-click PL2 and select **Other Half** in the context menu to locate the second half of the constraint.

   The browser exposes the second half of the constraint contained in Mold_Kinematics_MB_DA-IH-TCP.

8. Expand Mold_Kinematics_MB_DA-IH-BP to view the constraints.

   Notice the mate constraint named **PL3**. Right-click PL3 and select **Other Half** in the context menu.

   The browser exposes the second half of the constraint contained in Mold_Kinematics_MB_DA-IH-AP.

9. Expand Mold_Kinematics_MB_DA-IH-EP to view the constraints.

   Notice the mate constraint named **Ejection**. Right-click Ejection and select **Other Half** in the context menu.

   The browser exposes the second half of the constraint contained in Mold_Kinematics_MB_DA-IH-BCP.

10. On the Model browser, expand the Representations node.
11. Expand the Position node. Notice there are seven representations; Master, Runner Open, Sprue Open, Product Open, Ejection, Close, and Free Drag.
    - Right-click the Runner Open node and click Activate, Observe the position change.
    - Right-click the Sprue Open node and click Activate, Observe the position change.
    - Right-click the Product Open node and click Activate, Observe the position change.

- Right-click the Ejection node and click Activate. Observe the position change.
- Right-click the Close node and click Activate. Observe the position change.
- Right-click the Free Drag node and click Activate. Drag the movable components on the graphics screen.

12 Right-click the Master node and click Activate.

13 **NOTE** Edits to the mold are disabled unless the mold is in the closed position and Master representation.

Previous (page 209) | Next (page 212)

Edit the Kinematic Constraint Value

In this section, you will the value of a positional representation using a table.

1 On the Model browser, right-click **Mold_Kinematics_MB:1** and choose Edit.

2 Switch to the Representations browser.

3 In the upper part of the browser, select **Edit positional representation table**.

4 On the worksheet, change the value of **PL1** from **155 mm** to **160 mm**.

5 Save and close the worksheet.

6 Return to the Model browser.

7 Activate the **Product Open** representation.

8 Expand Mold_Kinematics_MB_DA-IH-RSP to view the constraints.

9 The **Product Open** override value is now 160 mm between the runner stripper plate (RSP) and cavity plate (AP).

NOTE You can also activate a positional representation and edit the override value in the browser.

10 Choose Return to activate the top assembly.

11 Activate the **Master** representation.

12 Save and close the file.

Previous (page 210) | Next (page 213)

Summary

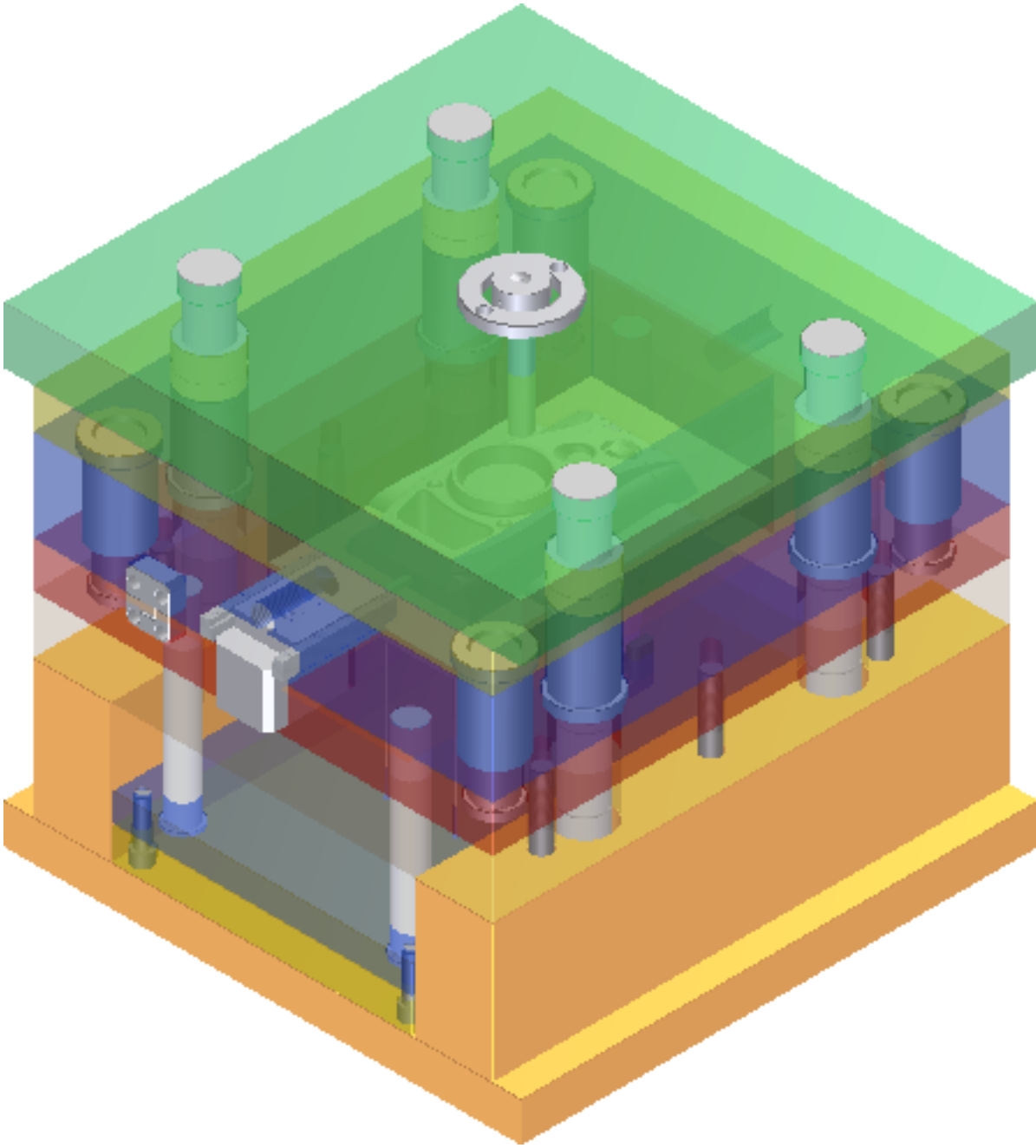
In this tutorial, you inserted 2-plate and 3-plate mold bases. You observed the default positional representations for each type.

You also learned how to:

- Delete a mold base.
- Activate positional representations.
- Drag components in the **Free Drag** representation.
- Drive a constraint.
- Edit a representation value in a table.

What Next?

- Open the assembly in the Inventor Studio environment and animate the positional representations to simulate mold sequences.
- Activate the **Free Drag** representation and then enter the Dynamic Simulation environment to perform a mold analysis.
- Add additional mold components to complete the mold assembly design.



Previous (page 212)

Index