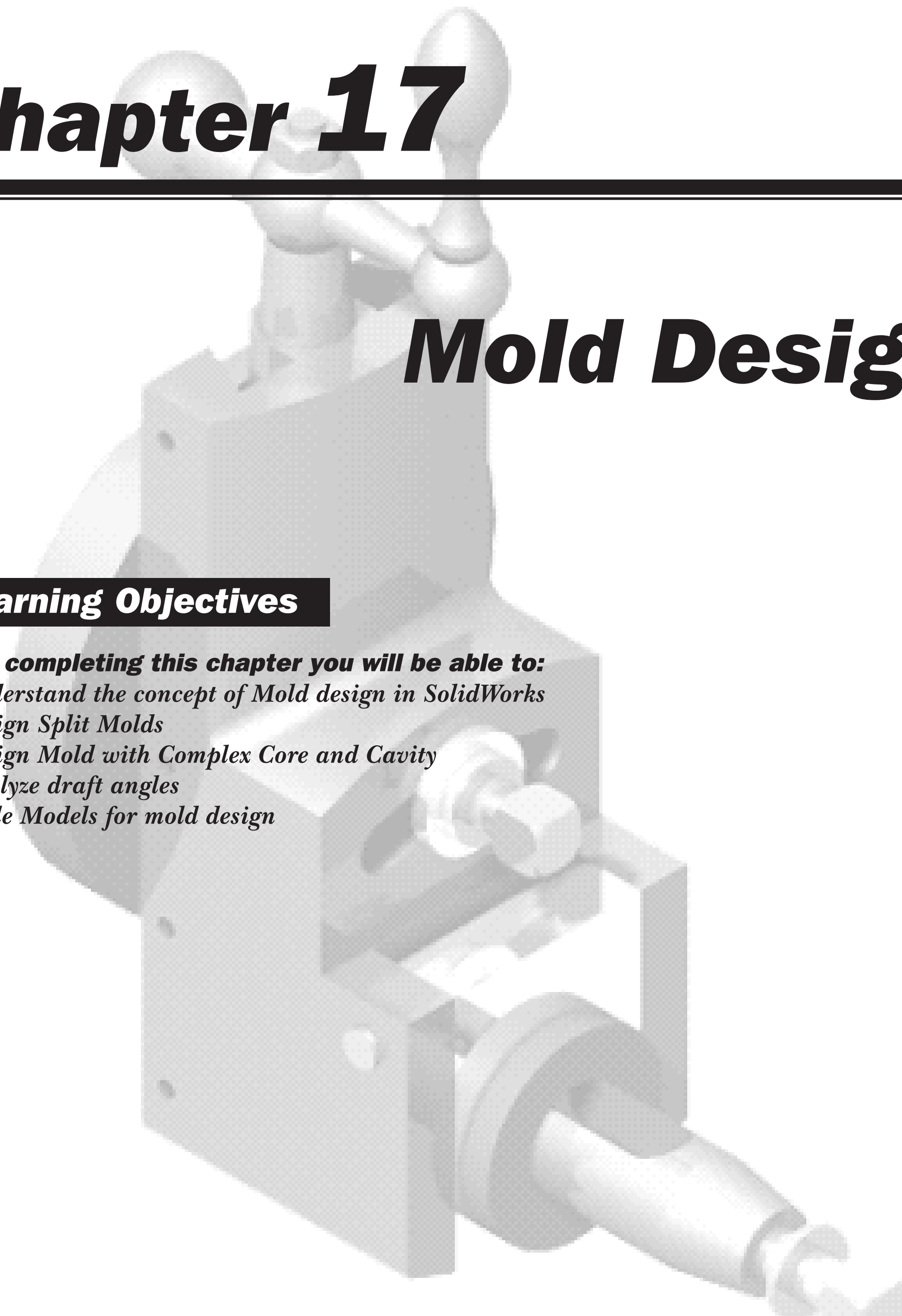
Chapter 17

Mold Design



Learning Objectives

After completing this chapter you will be able to:

- *Understand the concept of Mold design in SolidWorks*
- *Design Split Molds*
- *Design Mold with Complex Core and Cavity*
- *Analyze draft angles*
- *Scale Models for mold design*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

MOLD DESIGN

In this chapter, you will learn about the tools that you can use to design the molds for a casted or plastic molded components. Generally, the complete mold design consists of designing of mold base, designing of gates and runners, designing of ejection mechanism, and designing of core and cavity. But in the mold design using SolidWorks, you will only learn about design of core and cavity. In other words, you can say, extraction of core and cavity from a part created in the **Part** mode. You will also learn about creating the inserts for extracting the core and cavity of complex parts. Figure 17-1 shows a soap case created in the **Part** mode of SolidWorks. Figure 17-2 shows the core for manufacturing this part and Figure 17-3 shows the respective cavity.

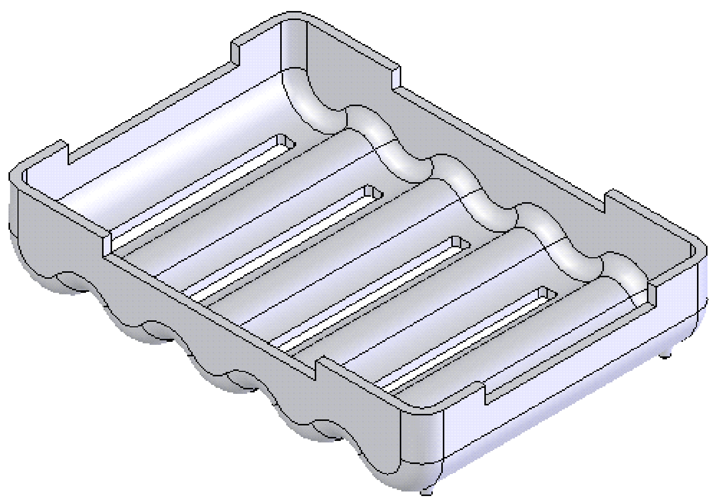


Figure 17-1 *Soap Case created in the* ***Part*** *mode of SolidWorks*

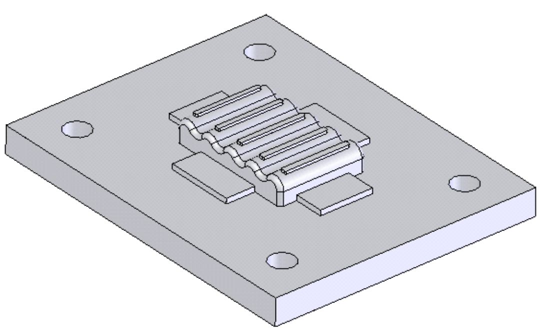


Figure 17-2 *Core of the mold for manufacturing this Soap Case.*

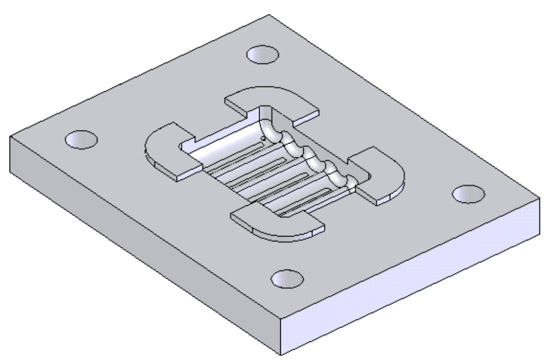


Figure 17-3 *Cavity of the mold for manufacturing this soap case.*

Before you learn about mold design, it is necessary that you first learn about some of the major terms and definitions related to mold design.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Mold Assembly

Mold assembly is an assembly created by assembling a number of parts such as top plate, back plate, core back plate, cavity back plate, ejector mechanism, guide bushes, guide pillars, other fasteners, cooling mechanism, and so on. The complete molds are designed by considering the part to be molded. Its design also depends on the complexity of the part.

Cavity

The cavity is the female part of the mold. It is hollowed from inside and the molten metal or the model plastic is poured so that it can take the shape of the cavity to create the component.

Core

The core is the male part of the mold and it defines the area where the molten metal should not reach. The core generally helps in defining the holes, pockets and other cut out features of a part while molding.

Parting Line

Parting line is a line from where the molds split and open in their respective directions. Figure 17-4 shows an assembly of core and cavity with the parting line. Figure 17-5 shows the exploded view of the core cavity assembly with the component that will be created using this combination of core and cavity.

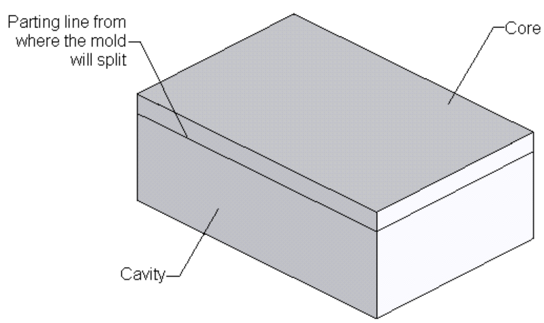


Figure 17-4 *Assembly of core and cavity with the parting line*

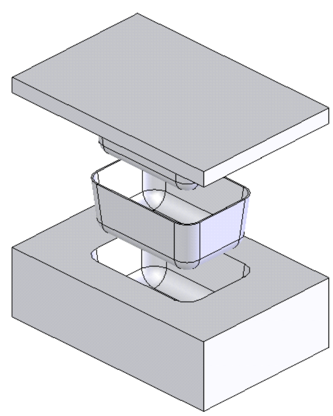


Figure 17-5 *Exploded view of the core and cavity assembly with the resultant component.*

Scale factor

Scale factor is defined as the amount by which the molded or casted component shrinks, depending on the type of material and the shape of the mold. The scale factor is defined in terms of percentage.

Draft Angle

In the molded or casted component, the draft angles are of great importance. It is very important to add draft in the core and cavity because if the faces of the core and cavity do not

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

have any draft angle, then you cannot take out the molded or casted component from the mold. Therefore, it is very important that you should add the required draft angles to the component in the **Part** mode so that the same draft is also added in the core and cavity.

MOLD DESIGNING USING SOLIDWORKS

In SolidWorks, there are two methods of designing the mold. The first method is generally used to design the split molds and the second method is generally used to design the molds that have core and cavity. Both the methods of designing the core and cavity are discussed next.

Designing a Split Mold

In technical terms, a simple mold consisting of two cavities can be called a split mold. For designing this type of mold, you first need to create the part for which you need to design the mold. Figure 17-6 shows a part whose mold is to be designed. Next, you need to create the mold base, which is generally an extruded cube or cuboid. Figure 17-7 shows the mold base. The over all dimensions of the mold base are greater than the part. The dimensions of the mold base depend on the size of the part, injection pressure of the molten metal, temperature of the molten metal, and so on.

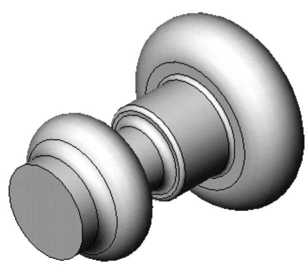


Figure 17-6 *Part whose mold is to be designed*

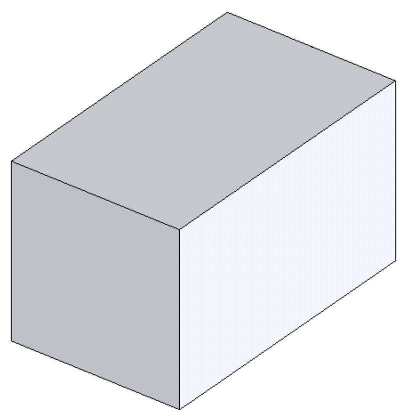


Figure 17-7 *Mold base*

The following are the steps required to create a split mold.

1. Creating an interim assembly.
2. Creating the cavity in the Mold base.
3. Creating the derived component.

Creating the Interim Assembly

After creating the part and the mold base, you need to create an interim assembly in which you will assemble the part with the mold base. Create a new assembly file and save it. Now, place the mold base in the assembly file. Select the mold base from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Component Properties** option from the shortcut menu. The **Component Properties** dialog box is displayed. Choose the **Color** button in this dialog box to invoke the **Assembly Instance Color** dialog box. Choose the **Advanced** button from this dialog box to invoke the **Advanced Properties** dialog box. Using the **Transparency** slider, increase the value of transparency and choose the **OK** button from the **Advanced Properties** dialog box. Now, choose the **OK** button from the other dialog boxes to exit all the dialog boxes. The mold base will be displayed in transparency. Now, place the part in the assembly and assemble it using the assembly mates such that the part is placed in the middle of the mold base. Figure 17-8 shows the part placed in the middle of the mold base.

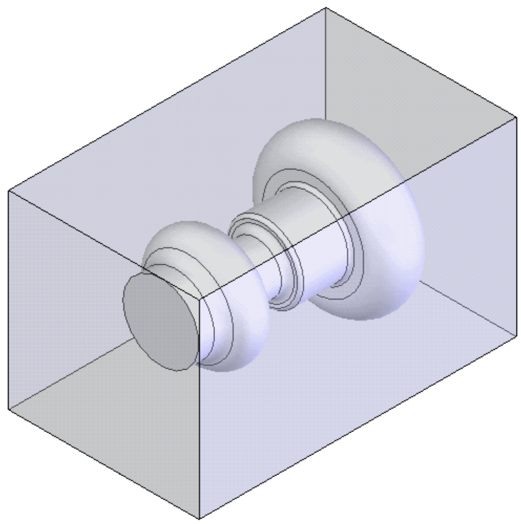


Figure 17-8 *Part placed and assembled in the middle of the mold base*

Creating the Cavity

Toolbar:	Mold Tools > Cavity
Menu:	Insert > Features > Cavity

After placing the part in the mold base, you need to create the cavity in the mold base. The cavity is created using the **Cavity** tool. To create a cavity, select the mold base and choose the **Edit Part** button from the **Assembly** toolbar. The **Part** modeling environment is invoked to edit the mold base part. Now, choose the **Cavity** button from the **Mold Tools** toolbar. If the **Mold Tools** toolbar is not available, choose **View** > **Toolbars** > **Mold Tools** from the menu bar to display this toolbar. When you invoke the **Cavity** tool, the **Cavity** dialog box is displayed as shown in Figure 17-9.

After invoking the **Cavity** dialog box, select the part from the **FeatureManager Design Tree**. The name of the selected part is displayed in the **Design component** display area. After selecting the component, you need to define the scale factor to mold cavity. The scale factor is defined to create the cavity larger than the part. This is done because after solidification, the resultant component shrinks. Since the **Uniform** check box is selected in the **Scale factor in %** area, the cavity will be scaled uniformly. Using the spinner provided in the **Scale factor in %** area, set the value of the scale factor. If you need to apply a nonuniform scaling to the

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

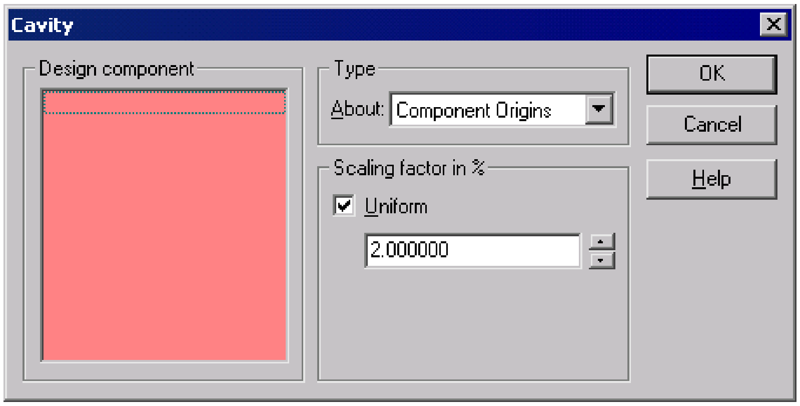


Figure 17-9 *The **Cavity** dialog box*

cavity, then clear the **Uniform** check box. The **X**, **Y**, and **Z** spinners are displayed to specify the different scale factors in all the axes.

The options available in the **About** drop-down list provided in the **Type** area are used to define the origin about which you need to scale the cavity. After setting all the parameters, choose the **OK** button from the **Cavity** dialog box. You will observe that a **Cavity1** feature is displayed in the **FeatureManager Design Tree**. Choose the **Edit Part** button from the **Assembly** toolbar to return to the **Assembly** environment.

Creating the Derived Component

After creating the cavity in the mold base, you need to create the derived component of the mold base that will be used to create the cavity. To create the derived component, select the mold base from the **FeatureManager Design Tree** and choose **File > Derive Component Part** from the menu bar. A new part file is created that contains the mold base with the cavity created inside the mold base. Since the mold base is displayed in shaded, therefore, you first need to change the model display to wireframe. The derived component after changing the model display is shown in Figure 17-10.

Select the front face of the derived component and invoke the sketching environment. Create a single line sketch as shown in Figure 17-11.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

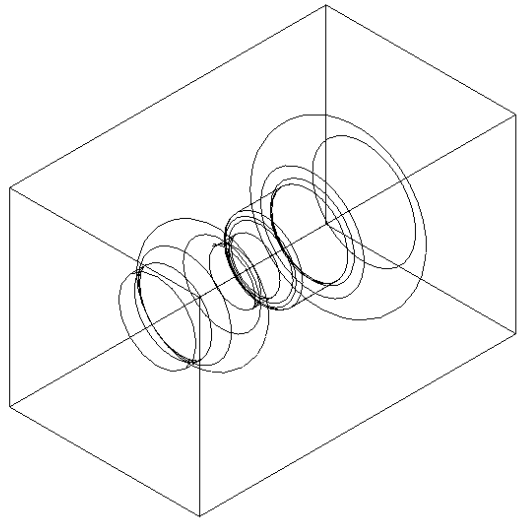


Figure 17-10 *Derived part with the model display set to wireframe*

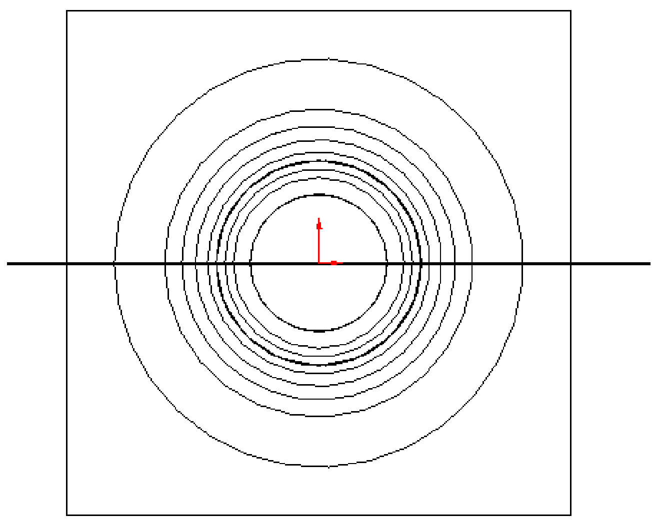


Figure 17-11 *Sketch for creating the cut feature*

Now, invoke the **Extrude-Cut PropertyManager** and create an extruded cut using the **Through All** option. The derived component after creating the cut feature is displayed in Figure 17-12.

Save this part with the name **Cavity1.** Now, choose **File > Save As** from the menu bar to invoke the **Save As** dialog box and save this part using the name **Cavity2.** After saving it, select the **Cut-Extrude1** feature from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Edit Definition** option from the shortcut menu to invoke the **Cut-Extrude1 PropertyManager**. Select the **Flip side to cut** check box from the **Direction 1** rollout and exit the **PropertyManager**. Again save the part file. This creates the top and bottom plates of the split mold.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

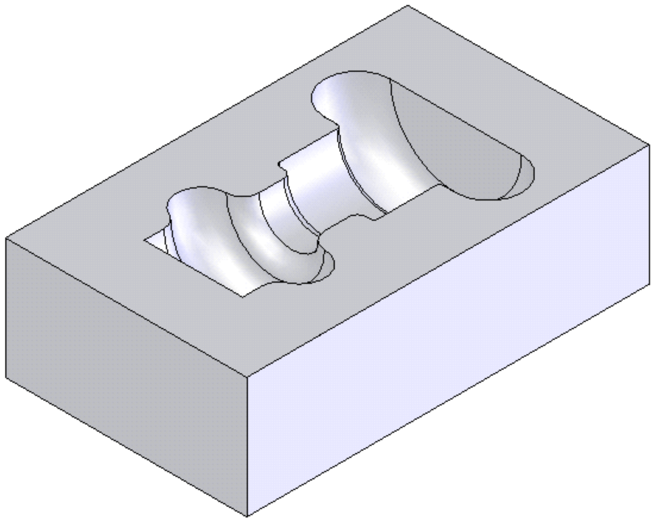


Figure 17-12 *Cavity*

To create the gates, runners, and other features, you can use the standard modeling tools.

Designing a Mold Having Core and Cavity

The second type is the one that has core and cavity. To create such a mold, you need to create the part in the **Part** mode. Make sure that you add the required draft angle while creating the part so that the component is easily extracted from the mold. You also need to make sure that the required scaling is added to the model so that the same scale factor is also reflected in the core and cavity. Consider an example of the carburetor cover shown in Figure 17-13. Figure 17-14 shows the other side of the model. Figure 17-15 shows the views and dimensions of the Carburetor Cover. This type of component requires a mold with core and cavity. The method of designing such a mold is discussed next.

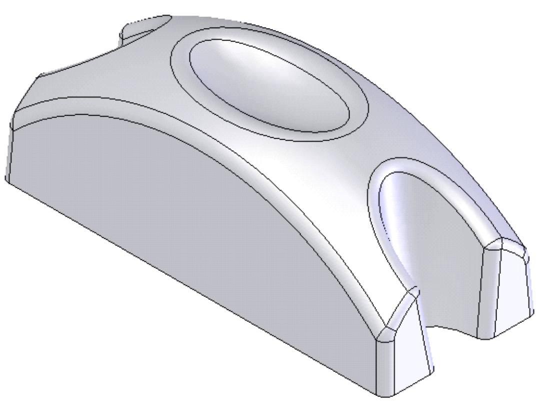


Figure 17-13 *Model of the Carburetor Cover*

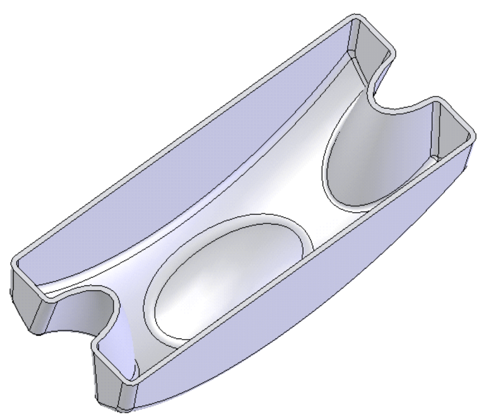


Figure 17-14 *Rotated view to display the other side of the Carburetor Cover*

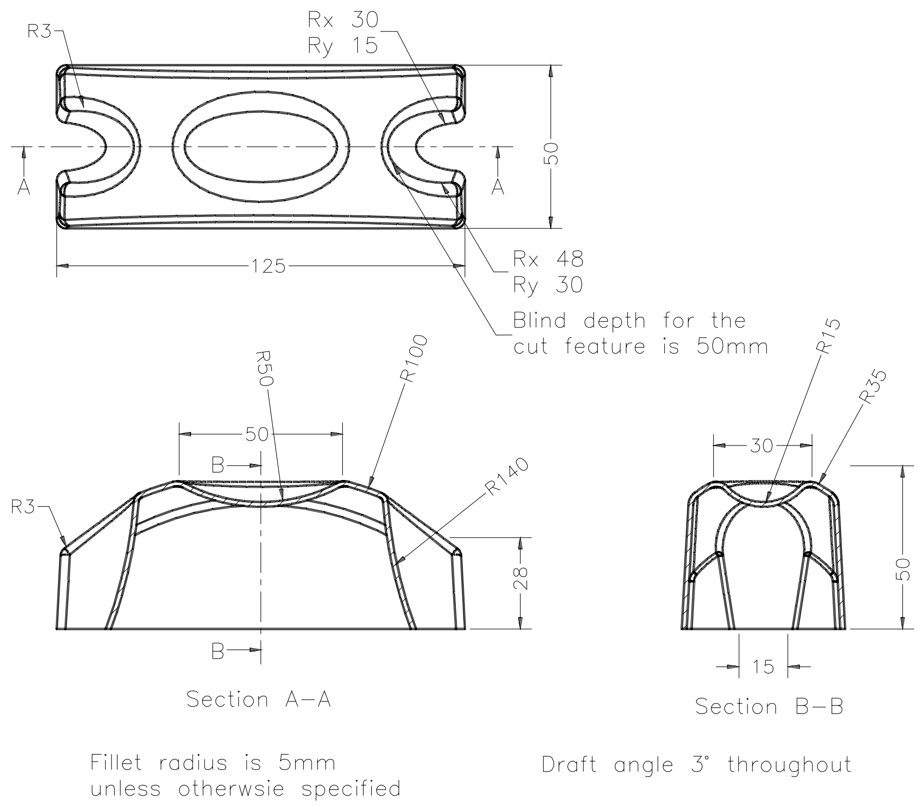
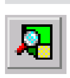


Figure 17-15 *Views and dimensions of the Carburetor Cover*

Analyzing the Draft Angles

Toolbar:	Mold Tools > Draft Analysis
Menu:	Tools > Draft Analysis



Before generating the core and cavity, you need to analyze the drafts applied to the model. The **Draft Analysis** tool is used to analyze the draft angles of the model. Choose the **Draft Analysis** button from the **Mold Tools** toolbar. The **Draft Analysis PropertyManager** is displayed as shown in Figure 17-16. First you need to select a plane, face, or an edge that will define the direction of pull. The name of the selected entity is displayed in the **Direction of Pull** display area and you are provided with the **Reverse Direction** button. The default draft angle value is displayed in the **Angle** spinner. This draft angle is specified as the required angle of draft for the analysis. You can set the value of the draft angle using the **Angle** spinner. Choose the **Calculate** button available in the **Analysis Parameters** rollout to analyze the draft angle using the default option selected in the **Draft Analysis PropertyManager**. The faces of the model will be displayed in different colors specifying the status of drafted faces. The faces displayed in green color indicate that they have a positive draft. The faces displayed in yellow indicate that they require a draft angle. The faces displayed in red indicate that they have a negative draft angle. Choose the **OK**

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

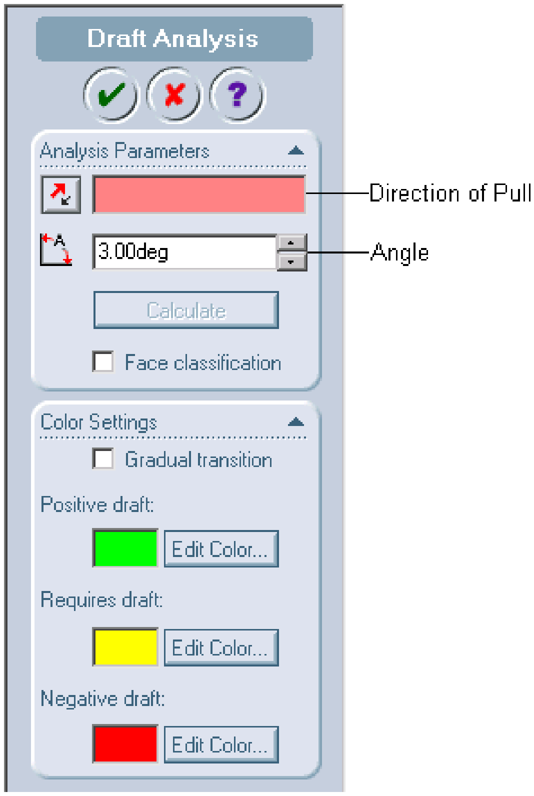


*Figure 17-16 The **Draft Analysis PropertyManager***

button from the **Draft Analysis PropertyManager** and add the draft to the faces that were displayed in yellow. The other options available in the **Draft Analysis PropertyManager** are discussed next.

Gradual transition

The **Gradual transition** check box is selected to display color variations specifying the draft angles on the model. This displays the range of angles starting from positive draft through negative draft depending on the value of the angle specified in the **Angle** spinner.

Face classification

The **Face classification** check box available in the **Analysis Parameters** rollout is used to refine the draft analysis, depending on the type of draft. When you select this check box, the **Hide/Show** buttons are provided with the **Positive draft**, **Requires draft**, **Negative draft**, **Straddle faces** areas. Using this **Hide/Show** button, you can display the type of faces that need to be analyzed.

When you select this check box, you will observe that the **Straddle faces** area is displayed. The straddle faces are the faces that have a combination of both positive and negative drafts. The display area available in this area is used to display the number of straddle faces displayed in the draft analysis.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Find steep faces

The **Find steep faces** check box is selected to analyze the positive steep faces and the negative steep faces. This check box is only available if you select the **Face classification** check box.

Scaling the Model

Toolbar:	Mold Tools > Scale
Menu:	Insert > Features > Scale

After creating the model and analyzing it for the draft angles, you need to scale the model so that the resultant core and cavity will be created bigger than the component that will be manufactured using that core and cavity. As discussed earlier, the size of the core and the cavity is always bigger than the resultant component because the component will shrink after solidification. For scaling the model, you need to invoke the **Scale** tool. Choose the **Scale** button from the **Mold Tools** toolbar. The **Scale PropertyManager** is displayed as shown in Figure 17-17.

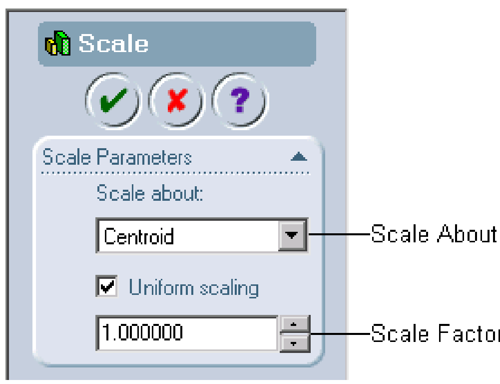
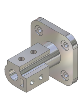


*Figure 17-17 The **Scale PropertyManager***

Using the options available in the **Scale about** drop-down list, you can specify the option about which you need to scale the model. Using the **Scale Factor** spinner, you can specify the scale factor for scaling the model.

The **Uniform scaling** check box is selected by default and the model is uniformly scaled. If you clear this check box, then the **X Scale Factor**, **Y Scale Factor**, and **Z Scale Factor** spinners are provided to specify the scale factor individually along all the three directions.



Note

*If you have more than one solid or surface bodies, then you need to select the bodies that are to be scaled using the **Scale PropertyManager**.*

Creating the Interim Assembly to Generate the Core and Cavity

After creating, analyzing, and scaling the part, you need to create an interim assembly in which you will place the component and then extract the core and cavity. Create a new SolidWorks document in the **Assembly** mode and then place the component in the

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

assembly coincident to the assembly origin. The component placed coincident to the assembly origin is displayed in Figure 17-18.

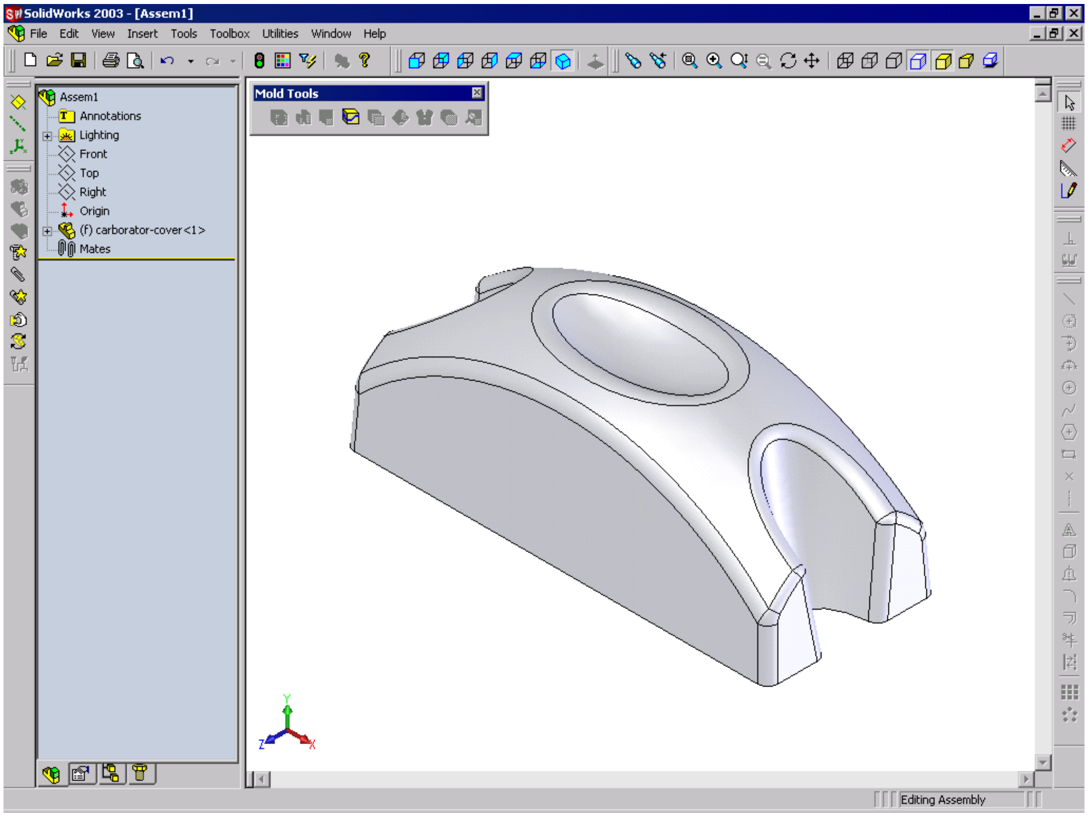


Figure 17-18 Carburetor Cover placed coincident to the assembly origin

Extracting the Core

After placing the component in the assembly, you need to create a new part document in which you will extract the core and cavity. The extraction of the core and cavity is a top-down assembly design approach. Therefore, you will create a new part document in the assembly file. Choose **Insert > Component > New Part** from the menu bar. The **Save As** dialog box is displayed, specify the file name as **Core** and choose the **Save** button from the **Save As** dialog box. Place the component on the front plane by selecting the **Front** plane from the **FeatureManager Design Tree**. The sketching environment is invoked and the Carburetor Cover is displayed in transparent. Exit the sketching environment.

Now, you need to create a radiated surface that will be used as the parting surface for extracting the core. Invoke the **Radiate Surface** tool and select the **Top** plane as the radiated surface. Now, select the outer edge of the base as the edge to radiate. Select the **Propagate to tangent faces** check box. Figure 17-19 shows the radiate direction and the edges to be radiated. Set the value of the **Radiate Distance** spinner to **80** and choose **OK**. Figure 17-20 shows the resultant radiated surface.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

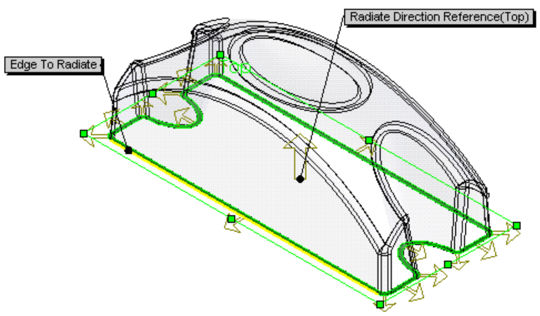


Figure 17-19 *Radiate direction and the edges to radiate*

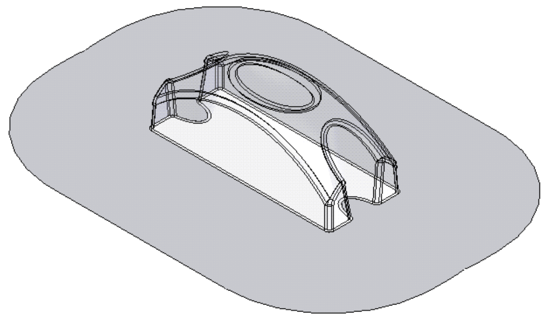


Figure 17-20 *Resultant radiated surface*

Next, you need to knit the radiated surface with the seed surface for creating a knit surface that will be used to generate the core. Invoke the **Knit Surface** tool. Select the radiated surface as the surface to knit and the seed surface, refer Figure 17-21. Choose the **OK** button from the **Knit Surface PropertyManager**.

Now, exit the part editing environment. Select the Carburetor Cover from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose **Hide components** from the shortcut menu. Figure 17-22 shows the resultant knitted surface.

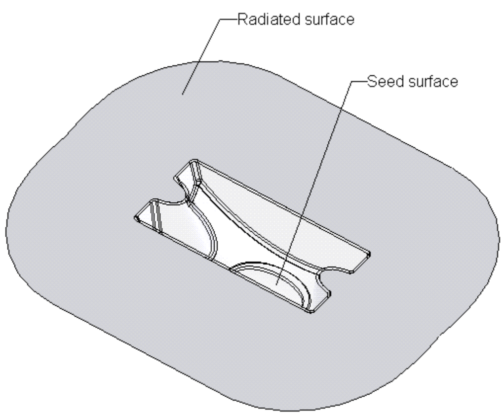


Figure 17-21 *Radiated surface and the seed surface*

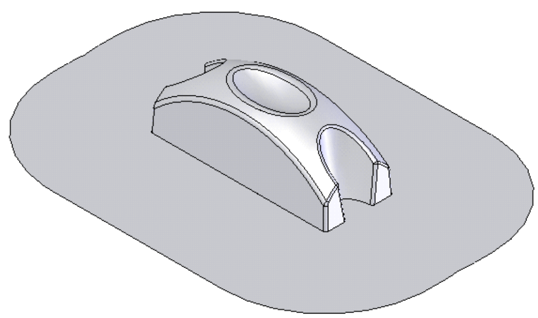


Figure 17-22 *Resultant knit surface*

Now, create a new plane at some offset distance below the radiated surface. Select the newly created plane as the sketching plane and invoke the sketching environment. Create the sketch of the core base and extrude it using the **Up To Surface** options with the knitted surface as the reference. Hide the knitted surface. Figure 17-23 shows the final core.

Choose the **Edit Part** button from the **Assembly** toolbar to return to the assembly environment. Save the assembly file.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

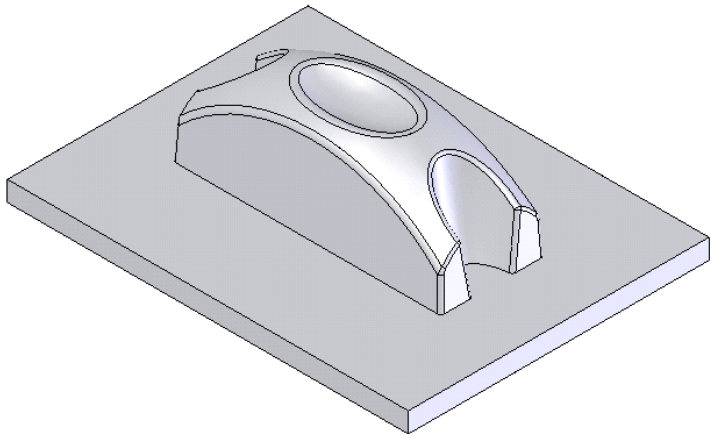


Figure 17-23 Final Core

Extracting the Cavity

Next, you need to generate the cavity of the mold. Before proceeding further, hide the Core and then unhide the Carburetor Cover. Now, create a new SolidWorks part document in the assembly file and save it with the name cavity. Place the cavity on the **Front** plane. As discussed earlier, create a radiated surface. Next, you need to create a knit surface. Invoke the **Knit Surface PropertyManager** and select the radiated surface and the seed surface, refer to Figure 17-24. Choose the **OK** button from the **Knit Surface PropertyManager**. Figure 17-25 shows the resultant knit surface.

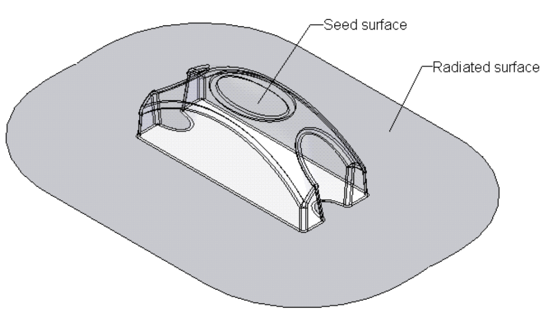


Figure 17-24 Radiated surface and the seed surface

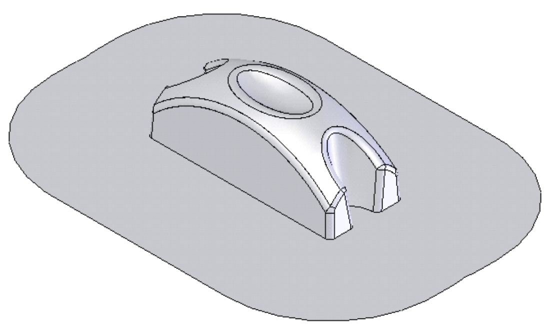


Figure 17-25 Resultant knit surface

Now, create a new plane offsetted vertically upwards to the radiated surface. Remember that the offset distance should be more than the total height of the carburetor cover, refer to Figure 17-26. Select this plane as the sketching plane and create the sketch for the cavity base and extrude it using the **Up To Surface** option. Now, hide the surface body. The final cavity is shown in Figure 17-26.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

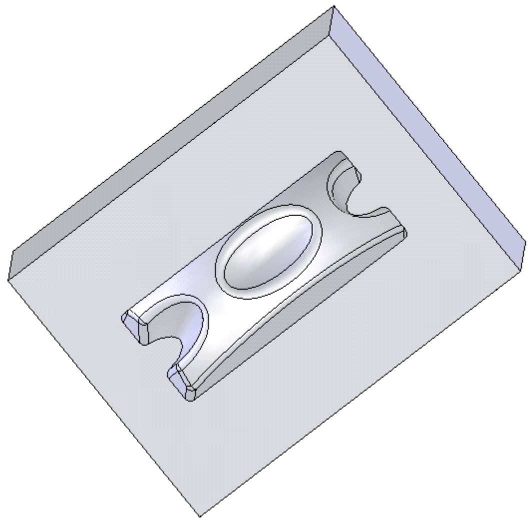


Figure 17-26 *Final cavity*

Now, unhide all the other components and create the exploded view using the **ConfigurationManager** as shown in Figure 17-27.

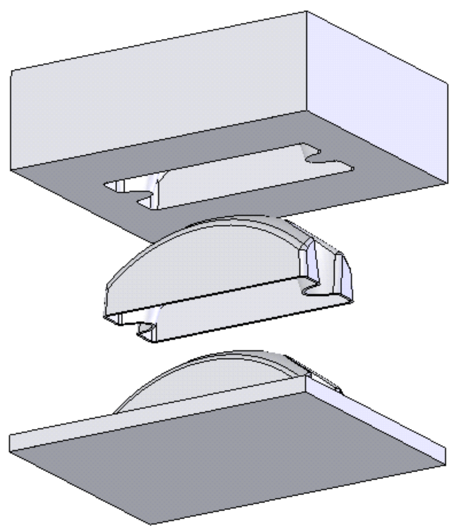


Figure 17-27 *Exploded view of the final assembly*

Save the assembly. The separate part documents of the Core and Cavity are also created. You can use these part documents for further use such as toolpath generation or for performing some analysis. You also need to use the standard modeling tools to create the gates, runners, and so on.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Note

Sometimes, you may need to place some inserts in the mold. Inserts are placed where you need to restrict the flow of metal in the mold. While designing the mold in SolidWorks, you need to create the inserts using the top-down approach.

TUTORIALS

Tutorial 1

In this tutorial, you will extract the core and cavity of the Plastic Cover shown in Figure 17-28. The resultant core is displayed in Figure 17-29 and the cavity is displayed in Figure 17-30. Figure 17-31 shows the exploded view of the Mold base assembly. Figure 17-32 shows the views and dimensions of the Plastic Cover. Figure 17-33 shows the dimensions of the Core Plate. The dimensions of the Cavity Plate are displayed in Figure 17-34. **(Expected time: 1.5 hrs)**

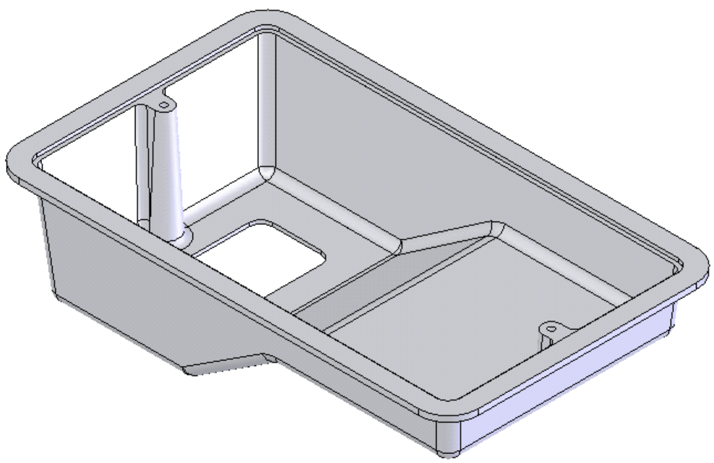


Figure 17-28 *Plastic Cover*

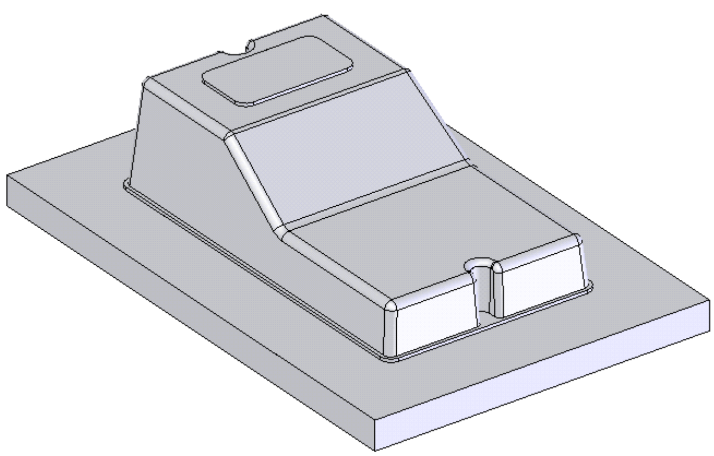


Figure 17-29 *Core of the Plastic Cover*

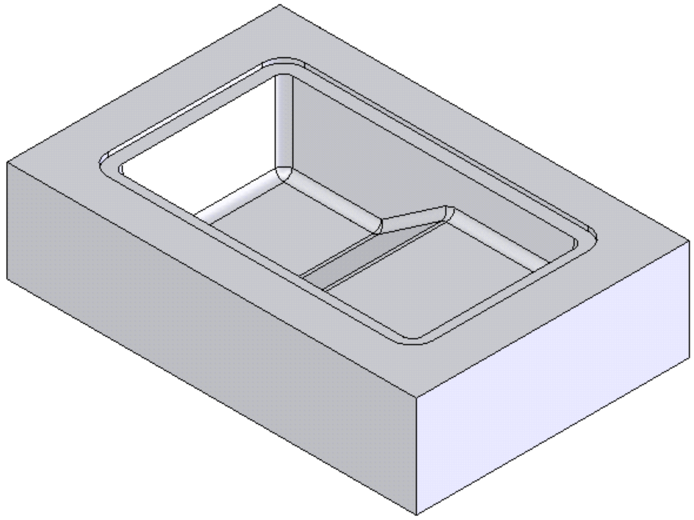


Figure 17-30 *Cavity of the Plastic Cover*

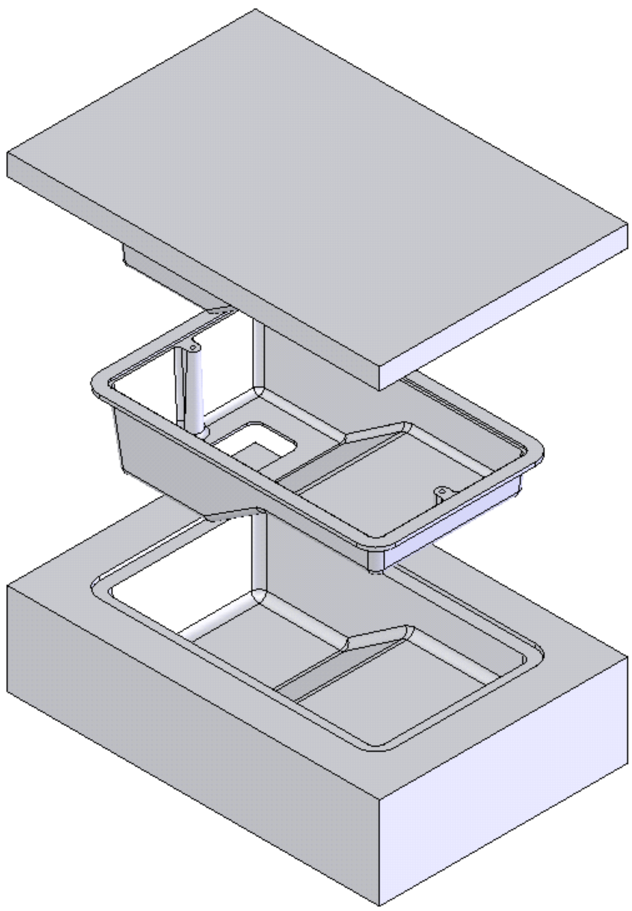


Figure 17-31 *Exploded view of the Mold Base assembly*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

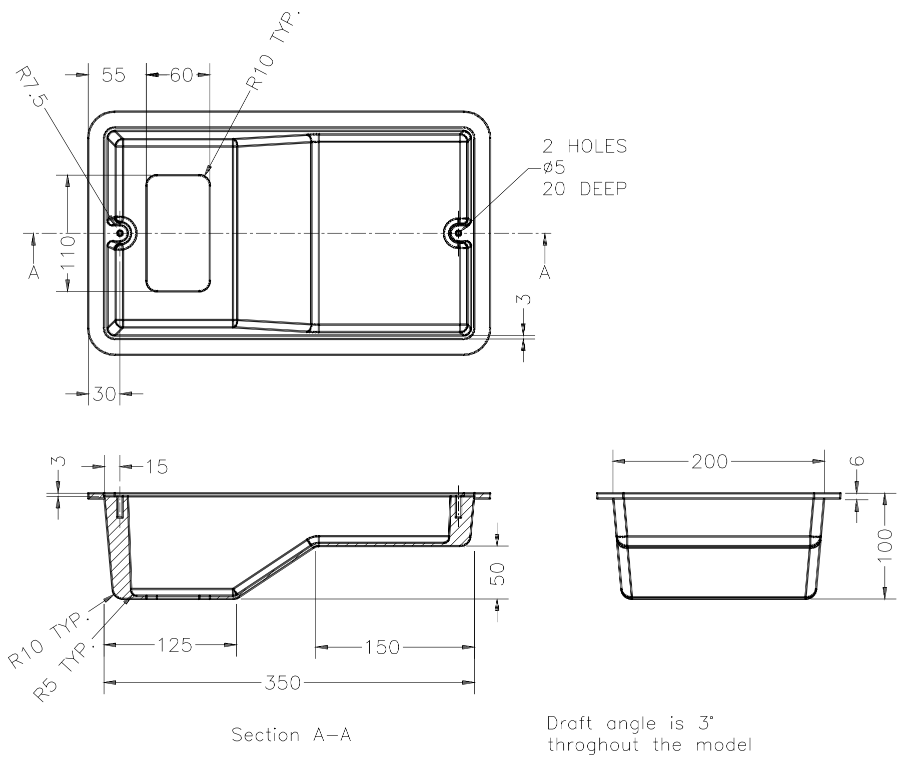


Figure 17-32 *Views and dimensions of the Plastic Cover*

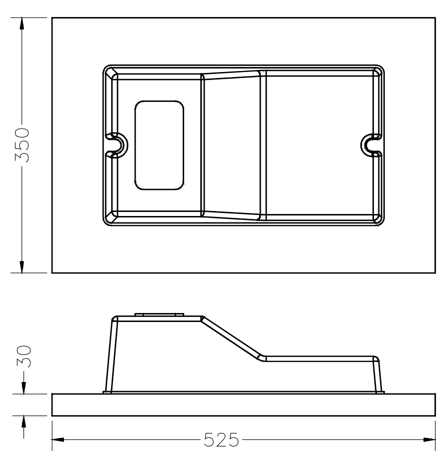


Figure 17-33 *Dimensions of the Core Plate*

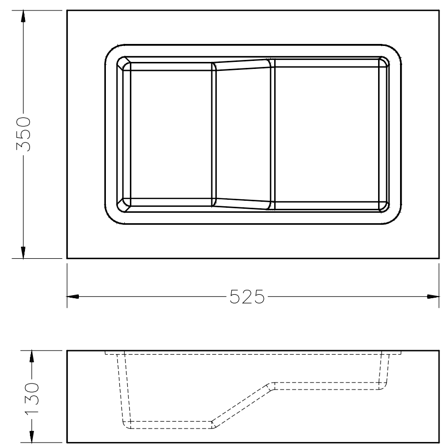


Figure 17-34 *Dimensions of the Cavity Plate*

The steps to be followed to complete this tutorial are discussed next.

a. Create the new SolidWorks document and create the model of the Plastic Cover, refer to Figure 17-35.
b. Analyze the model for draft angles using the Draft Analysis.
c. Scale the model.
d. Create the shut off surfaces, refer to Figures 17-36 and 17-37.
e. Create a new SolidWorks document in the **Assembly** mode and place the Plastic Cover in the assembly document.
f. Using the top-down approach, extract the core and cavity of the Plastic Cover, refer to Figures 17-38 through 17-46.
g. Explode the assembly, refer to Figure 17-47.

Creating the Plastic Cover

First, you need to create the Plastic Cover in the **Part** mode of SolidWorks.

1. Create a new SolidWorks document in the **Part** mode.

2. Using the standard modeling tools, create the model of the Plastic Cover. For dimensions, refer to Figure 17-32.

3. Save the model. Figure 17-35 shows the final model created using the standard modeling tools.

Analyzing the Model for Draft Angles

As discussed earlier, before extracting the core and cavity, you need to make sure that the required draft angles are applied to the model so that the component is easily removed from the model after solidification of the plastic. Therefore, you will analyze the model using the **Draft Analysis** tool.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

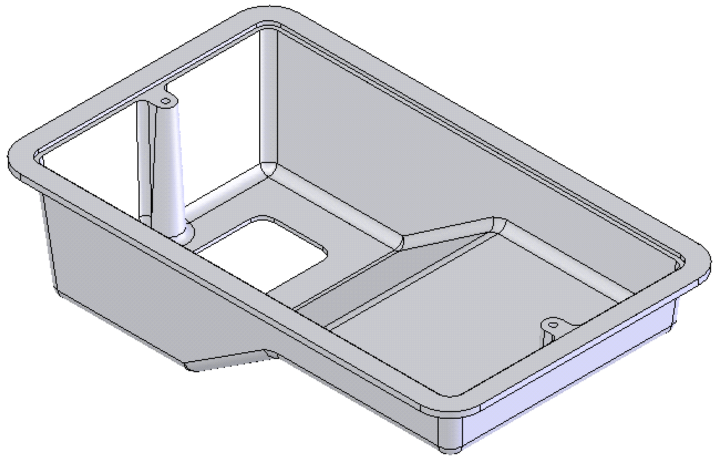
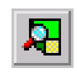


Figure 17-35 *Final model of the Plastic Cover*

1. Choose the **Draft Analysis** button from the **Mold Tools** toolbar. The **Draft Analysis PropertyManager** is displayed.



   First, you need to select the direction of pull along which the mold will be pulled out.

2. Select the **Top** face of the Plastic Cover to define the direction of pull. The selected face is highlighted in red.

3. Choose the **Calculate** button from the **Analysis Parameters** rollout to analyze the draft angles.

   You will notice that faces having the positive draft and the negative draft are displayed in green and red color respectively. You will also notice that some small faces are displayed in yellow color, suggesting that the draft angle is required on these faces. But you will notice that those faces are very small in height as compared to other faces of the part. Therefore, the draft angle is not required on those faces. You will also notice that the hole placed on one of the top faces of the part also displays its inner faces in yellow color. But you do not need to add draft in the inner faces of these holes because these holes will not be included in the core. These type of holes are drilled in the final molded part.

   If any face other than the faces having minor thickness are displayed in yellow, then you need to add the required draft angle on those faces.

4. Choose the **OK** button from the **Draft Analysis PropertyManager**.

Scaling the Model

After analyzing the model for draft angles, you need to scale the model. As discussed earlier, the scaling of the model is done in order to create the core and cavity larger than the component to be manufactured using this tool. This is done because after solidification, the component always shrinks. The scale factor always depends on the material to be used for molding or casting and also on the type of mold.

1. Choose the **Scale** button from the **Mold Tools** toolbar to invoke the **Scale PropertyManager**.

2. Set the value of the scale factor in the **Scale Factor** spinner to **1.1**.

3. Choose the **OK** button from the **Scale PropertyManager**.

Creating the Shutoff Surfaces

You will observe that there is a rectangular slot on the bottom face of the Plastic Cover. It is always recommended that the cavity should be smooth and all the impressions or embosses are on the core. Therefore, you need create a shutoff surface so that you can get the impression of the slot on the core. You also need to create shutoff surfaces to close the holes.

1. Rotate the model such that the bottom face of the model is clearly visible.

2. Choose the **Planar Surface** button from the **Mold Tools** toolbar to invoke the **Planar Surface PropertyManager**.

3. Select the edges to create a shutoff surface as shown in Figure 17-36.

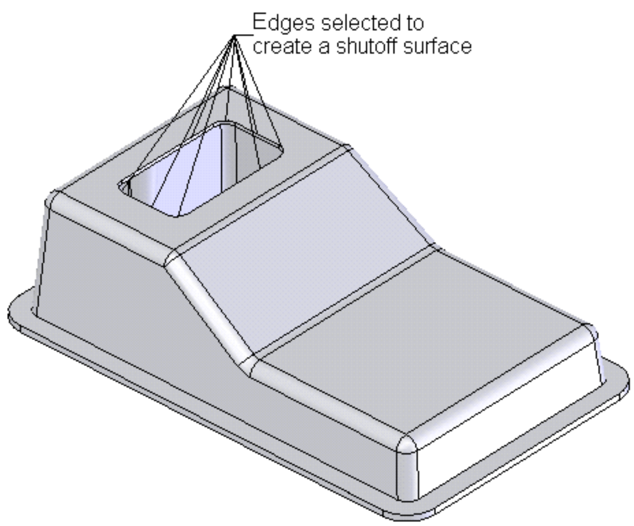


Figure 17-36 *Edges selected to create a shutoff surface.*

4. Choose the **OK** button from the **Planar Surface PropertyManager**. Model after creating the shutoff surface using the **Planar Surface** tool is shown in Figure 17-37.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

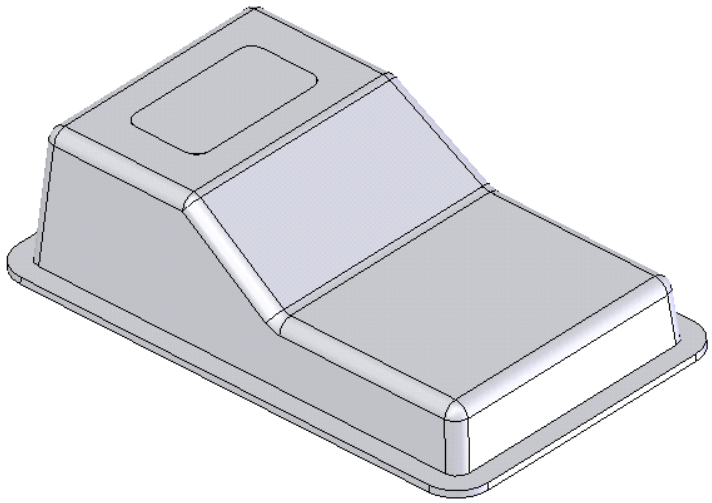


Figure 17-37 *Model after creating the shutoff surface*

5. Similarly, using the **Planar Surface** tool, create the shutoff surfaces on the edges of the holes created on one of the top faces of the model.

6. After creating the surfaces, save the model.

Creating the Interim Assembly

After finalizing the model, you need to create an interim assembly. Using this interim assembly and the top-down approach, you will extract the core and cavity of the model. To create the interim assembly, you first need to create a new SolidWorks document in the **Assembly** mode.

1. Create a new SolidWorks document in the **Assembly** mode.

2. Place the Plastic cover coincident to the assembly origin.

3. Save the assembly file with the name **Mold base**.

Extracting the Cavity

After creating the interim assembly, you need to extract the cavity. For extracting the cavity, you will use the top-down approach of the assembly. For extracting the cavity, you need to create a new SolidWorks document in the **Part** mode.

1. Choose **Insert > Component > New Part** from the menu bar.

   The **Save As** dialog box is displayed. Enter the name of the new part as **Cavity** in the **File name** edit box and choose the **Save** button from the **Save As** dialog box.

2. Select the **Front** plane from the **FeatureManager Design Tree** to place the new component.

As you place the component, the Plastic Cover is displayed in transparent and the sketcher environment in invoked.

3. Exit the sketcher environment.

4. Using the **Radiate Surface** tool, create a radiated surface with the radiate distance of 9mm, refer to Figure 17-38.

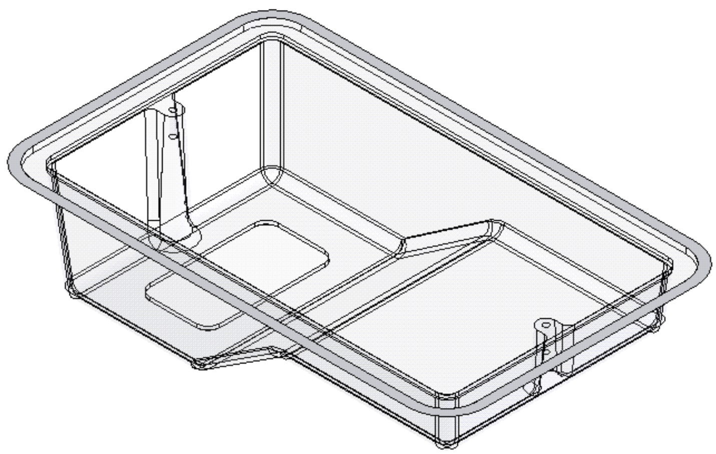


Figure 17-38 *After creating the radiated surface*

Remember that if you have created shutoff surfaces in the model, there will be no seed surface. You need to select all the other faces of the model and the radiated surface as the surface to knit.

5. Invoke the **Knit Surface** tool to knit the radiated surface with all the outer faces of the Plastic Cover. For knitting the radiated surface with the outer surfaces of the model, you need to select all the outer faces of the model as well as the radiated surface as the surfaces to knit. In this case, you will not select any surface as the seed surface. Also, select all the shutoff surfaces, including the shutoff surface on the bottom face of the plastic cover. After selecting the surfaces, choose the **OK** button from the **Knit Surface PropertyManager**.

6. Select the **Plastic cover** from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Hide components** option from the shortcut menu. The resultant surface body extracted is displayed in Figure 17-39.

Next, you need to create a plane at an offset distance from the radiated surface and then create the sketch of the cavity on the newly created plane. Then, extrude the sketch of the cavity up to the knitted surface.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

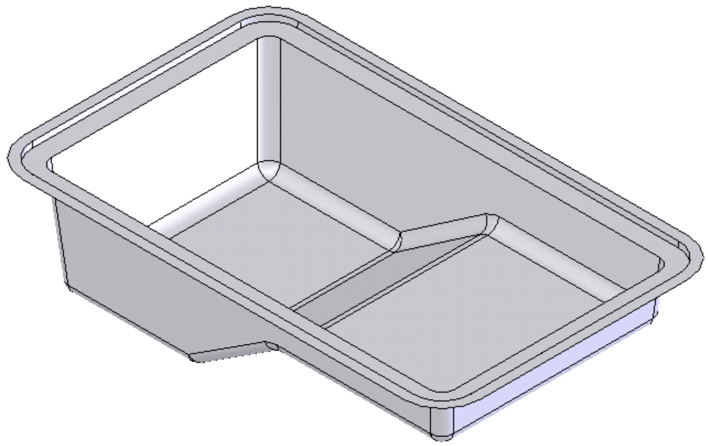


Figure 17-39 *Resultant knitted surface after hiding the Plastic Cover*

7. Create a new plane vertically downwards at an offset distance of 130 mm from the radiated surface.

8. Create the sketch for creating the solid cavity as shown in Figure 17-40.

9. Now, using the **Up To Surface** option, extrude the sketch up to the knitted surface.

10. Hide the knit surface.

    The resultant cavity is displayed in Figure 17-41.

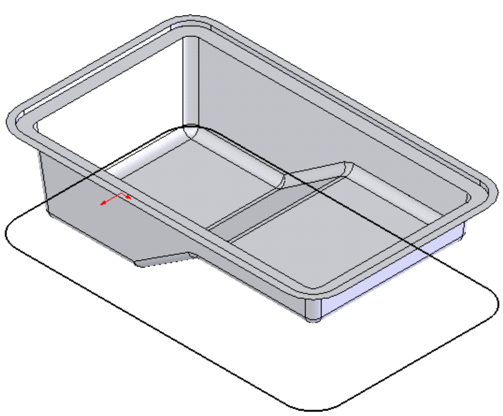


Figure 17-40 *Sketch for the Cavity Plate*

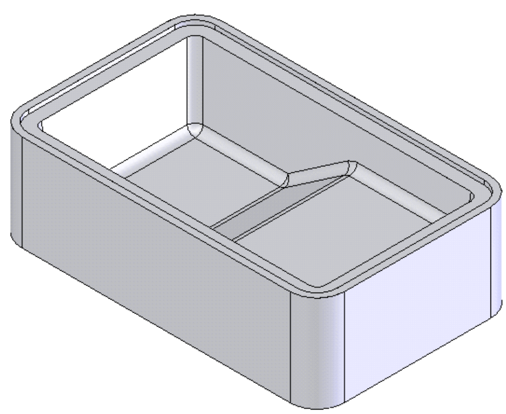


Figure 17-41 *Resultant Cavity*

Next, you need to create the cavity plate. The cavity plate will be created by extruding a rectangular sketch to a distance of 130mm.

11. Select the bottom face of the cavity as the sketching plane and invoke the sketching environment.

12. Create the sketch which consists of a rectangle of 525x350.

13. Extrude the sketch to a distance of 130mm.

14. Save the assembly.

Figure 17-42 shows the final Cavity Plate.

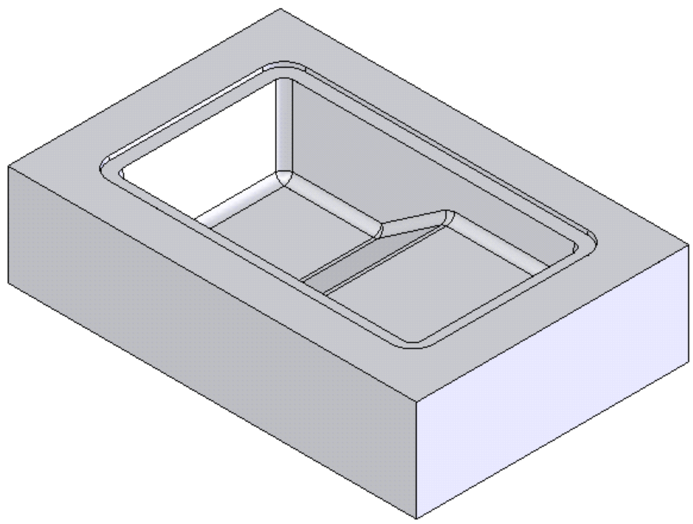


Figure 17-42 *Final Cavity Plate*

Extracting the Core

Next, you need to extract the core from the Plastic Cover. The core will also be extracted using the top-down approach of the assembly. Before proceeding further, you need to hide the cavity and unhide the Plastic Cover.

1. Hide the Cavity and unhide the Plastic Cover.

2. Choose **Insert** > **Component** > **New Part** from the menu bar. The **Save As** dialog box is displayed.

3. Enter the name of the new component as **Core** in the **File name** edit box and choose the **Save** button from the **Save As** dialog box.

4. Place the new part on the **Front** plane in the **FeatureManager Design Tree**.

5. Exit the sketcher environment.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

6. Create a radiated surface as done earlier.

7. Now, knit all the internal surfaces of the Plastic Cover with the radiated surface.

8. Hide Plastic Cover.

   The resultant extracted surface is displayed in Figure 17-43.

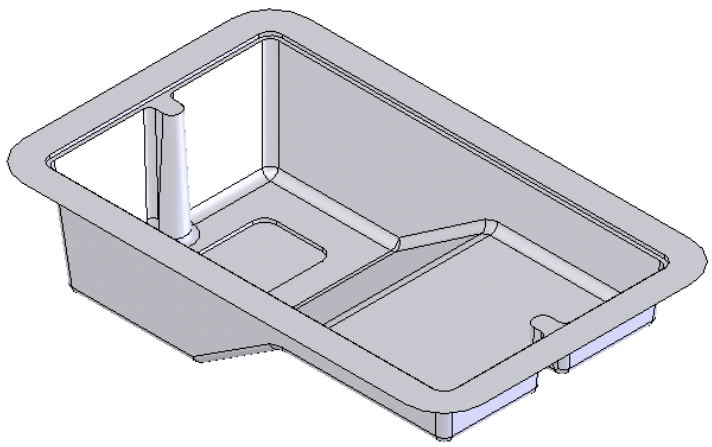


Figure 17-43 *Resultant extracted surface that will be used to create the core*

9. Now, create a plane at an offset distance of 30mm from the top face of the extracted surface.

10. Select the newly created plane as the sketching plane and create the sketch of the core as shown in Figure 17-44.

11. Extrude the sketch up to the knitted surface and hide the Plastic Cover. Figure 17-45 shows the resultant core extracted from the Plastic Cover.

12. Hide the surface body.

13. Now, create the core plate by extruding the sketch created on the top face of the core.

14. Save the assembly.

   The final core is displayed in Figure 17-46.

Creating the Exploded View of the Assembly

After extracting the core and cavity of the Plastic Cover, you need to create the exploded

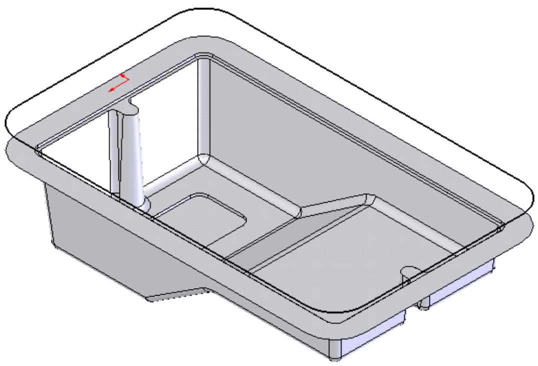


Figure 17-44 *Sketch for creating the core*

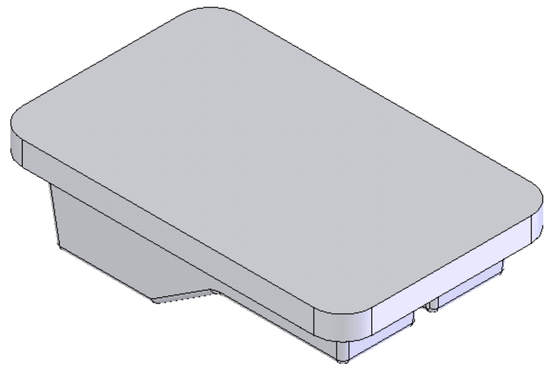


Figure 17-45 *Resultant core*

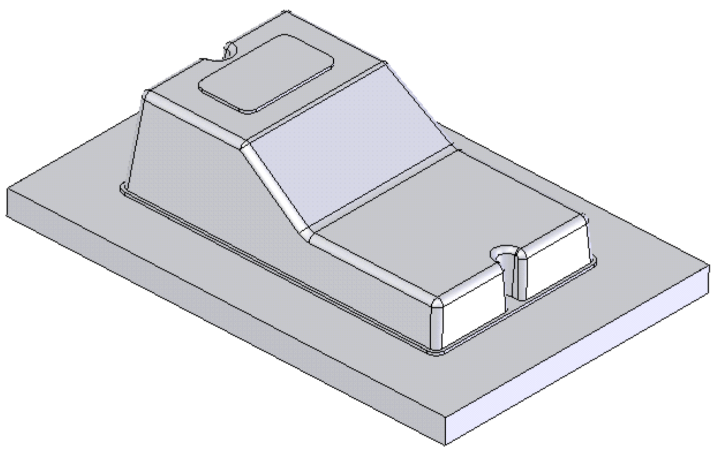


Figure 17-46 *Final core*

view of the Mold base assembly. For creating the exploded view of the mold base assembly, first you need to unhide the Cavity and the Plastic Cover.

1. Unhide the Cavity and the Plastic Cover.

2. Invoke the **ConfigurationManager** and select the **Default** configuration from the **ConfigurationManager**. Invoke the shortcut menu and select the **New Exploded View** option from the shortcut menu.

3. Using the **Assembly Exploder** dialog box, explode the Mold base assembly, refer to Figure 17-47.

4. Save the assembly.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

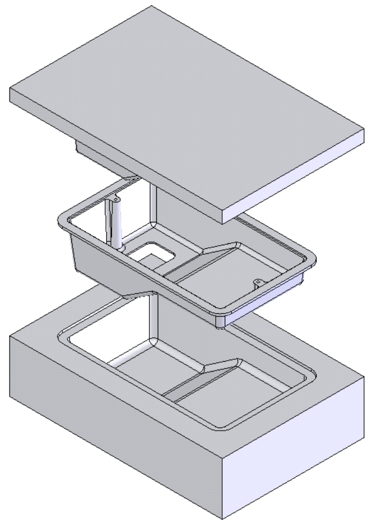


Figure 17-47 *Exploded view of the Mold Base assembly*

Tutorial 2

In this tutorial, you will extract the core and cavity of the Cover shown in Figure 17-48. The other side of the Cover is displayed in Figure 17-49. The Core of the Cover is displayed in Figure 17-50 and the Cavity in Figure 17-51. The Insert1 to be inserted in the mold is displayed in Figure 17-52. The second insert is also the same as displayed in Figure 17-51. Figure 17-53 shows the exploded view of the Mold base assembly. Figure 17-54 shows the views and dimensions of the cover. Figure 17-55 shows the views and dimensions of the Core. Figure 17-56 shows the views and dimensions of the Cavity. Figure 17-57 shows the views and dimensions of the Insert1. After creating the Mold base assembly, you will also create the exploded view of the assembly. **(Expected time: 2hrs)**

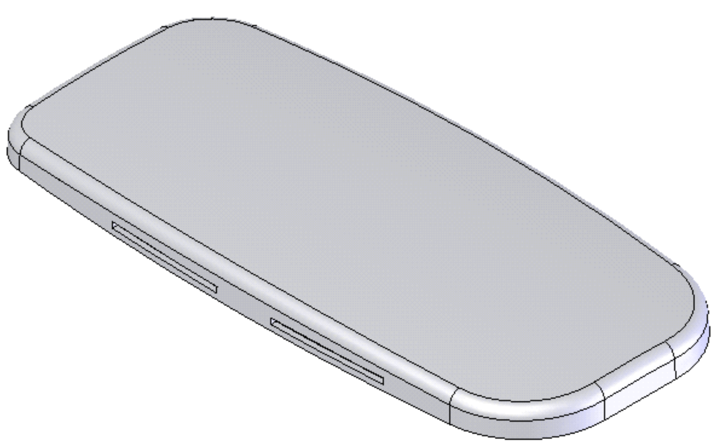


Figure 17-48 *Isometric view of the Cover*

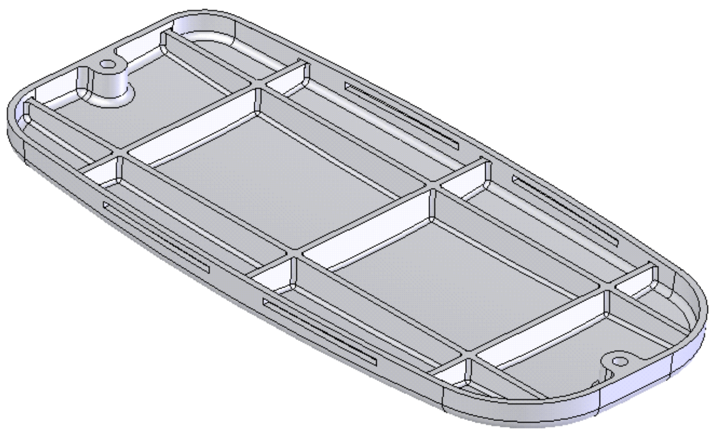


Figure 17-49 *Rotated view of the Cover*

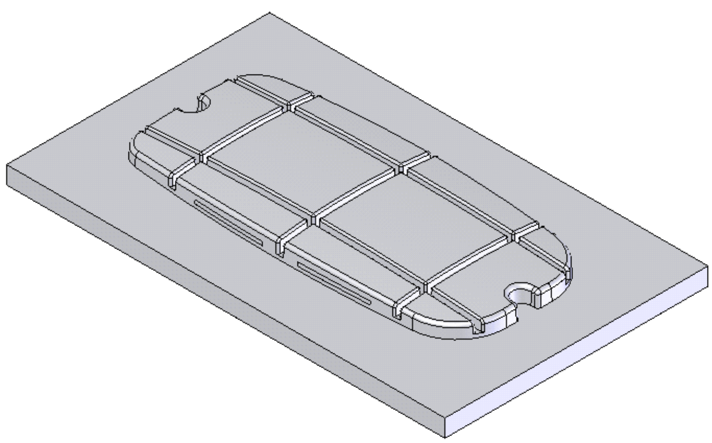


Figure 17-50 *Isometric view of the Core*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

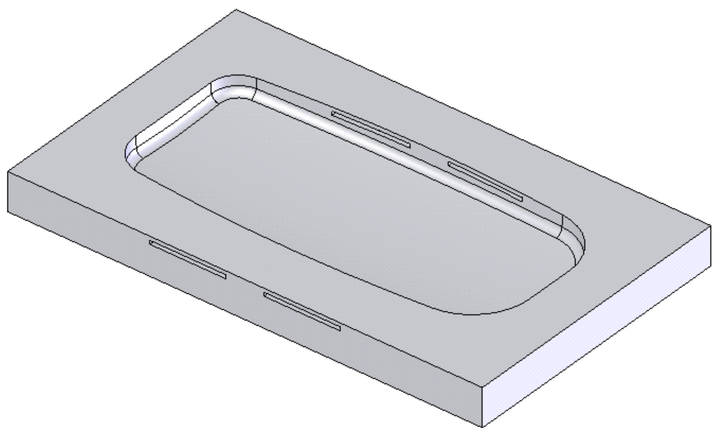


Figure 17-51 *3D view of the Cavity*

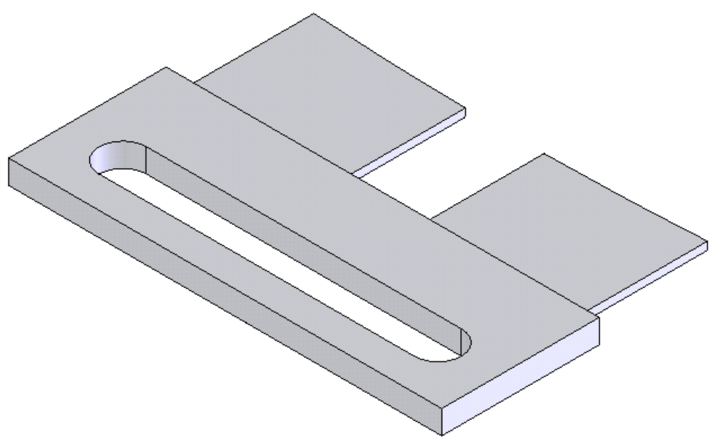


Figure 17-52 *Isometric view of the Insert*

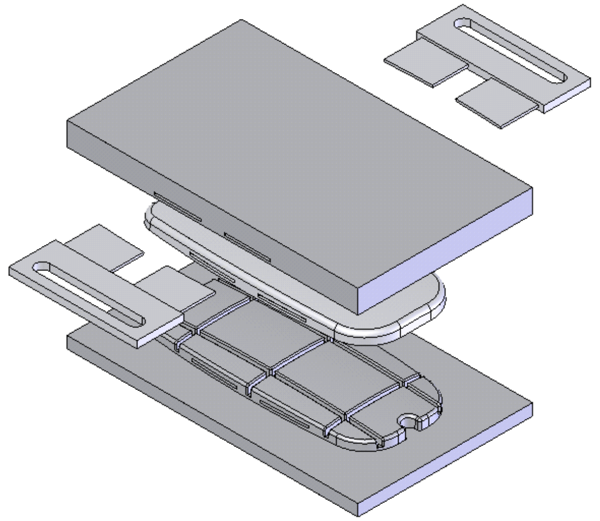


Figure 17-53 *Exploded view of the Mold Base assembly*

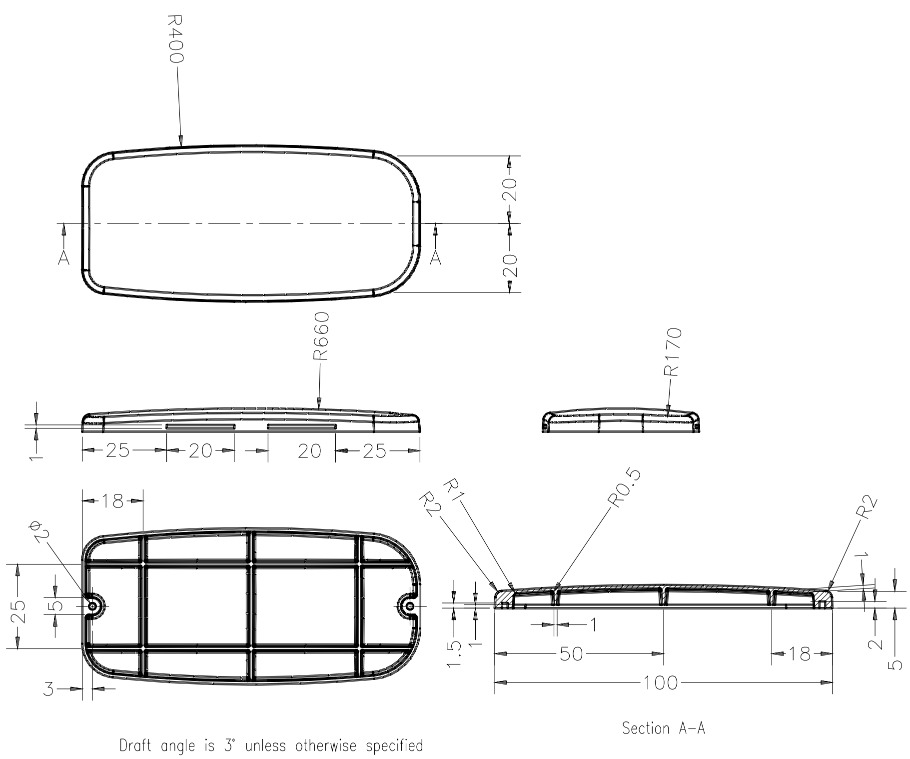


Figure 17-54 *Views and dimensions of the Cover*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

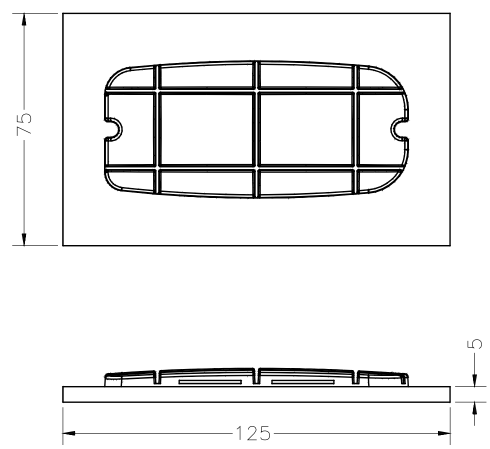


Figure 17-55 Views and dimensions of the Core

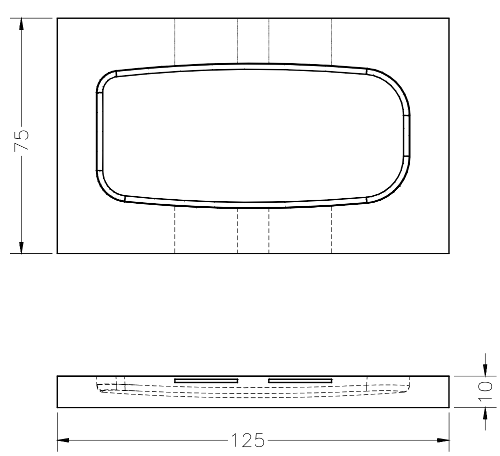


Figure 17-56 Views and dimensions of the Cavity

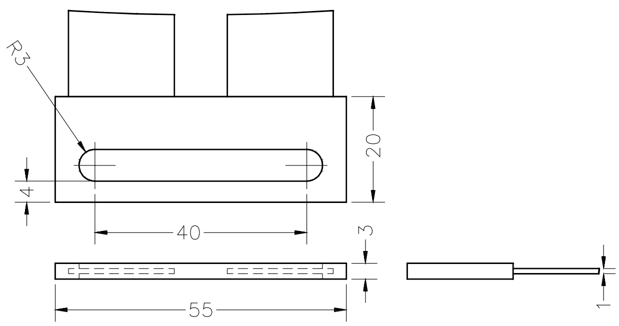


Figure 17-57 *Views and dimensions of the Insert*

The steps to be followed to complete this tutorial are discussed next.

a. Create a new SolidWorks document in the **Part** mode and using the standard modeling tools, create the model of the Cover, refer to Figure 17-58.
b. Analyze the model for the draft angles.
c. Scale the model.
d. Create the shutoff surfaces.
e. Create the interim assembly.
f. Extract the Core, refer to Figure 17-59.
g. Extract the Cavity, refer to Figure 17-60.
h. Create the slots in the cavity for the insertion of inserts, refer to Figure 17-61.
i. Using the top-down approach, create the inserts, refer to Figures 17-62 through 17-64.
j. Create the exploded view of the assembly, refer to Figure 17-65.

Creating the Cover Part

First, you need to create the model of the cover plate. For creating the model, create a new SolidWorks document in the **Part** mode.

1. Create a new SolidWorks document in the **Part** mode.

2. Using the standard modeling tools, create the model of the Cover. The completed model of the Cover is displayed in Figure 17-58.

Analyzing the Draft Angles

After creating the model, you need to analyze the model for draft angles using the **Draft Analysis** tool.

1. Invoke the **Draft Analysis** tool from the **Mold Tools** toolbar to display the **Draft Analysis** dialog box. Using this tool, analyze the draft angles in the model. If some faces of the

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

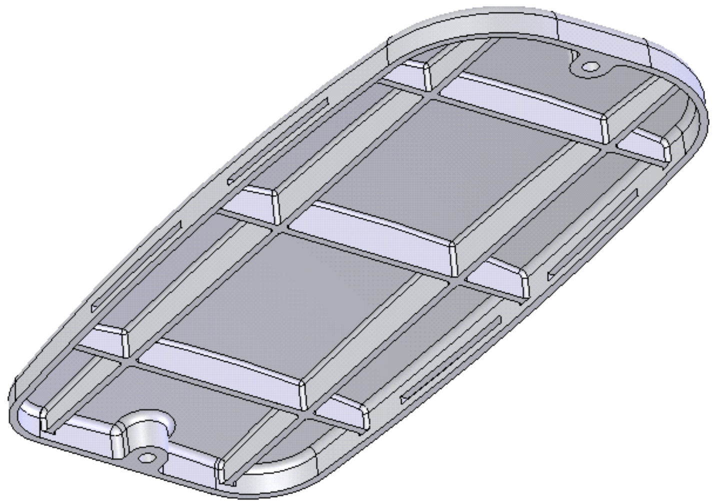


Figure 17-58 Model of the Cover

model are displayed in yellow, then you need to add the draft angles on those faces. You can ignore the circular faces of the holes placed at the bottom face of the Cover if they are displayed in yellow color.

Note

If some part of the middle portion of the curved face is displayed in yellow, you can ignore this.

Scaling the Model

After analyzing the model for draft angles and adding the drafts as required, you need to scale the model.

1. Invoke the **Scale PropertyManager**.

2. Set the value of the scale factor in the **Scale Factor** spinner to 1.1.

3. Choose the **OK** button from the **Scale PropertyManager**.

Creating the Shutoff Surfaces

After scaling the model, you need to create the shutoff surfaces.

1. Using the **Planar Surface** tool, create a planar surface using the edges of the holes placed on the bottom face of the Cover part.

   You also need to create the surfaces to create the shutoff surfaces on the slots created on the front and the back face of the model.

2. Using the **Fill Surface** tool, shut the openings on the outer front face and on the inner front face.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Similarly, shut the openings on the outer back face and on the inner back face using the **Fill Surface** tool.

3. Save the model.

Creating the Interim Assembly

After creating and saving the part model, you need to create an interim assembly. In this interim assembly, you will extract the core and cavity using the top-down approach.

1. Create a new SolidWorks document in the **Assembly** mode.

2. Save the assembly the name **Mold base**.

3. Place the Cover part coincident to the assembly origin.

Extracting the Core

For extracting the core, first you need to place a new SolidWorks part document in the assembly.

1. Choose **Insert** > **Component** > **New Part** from the menu bar. The **Save As** dialog box is displayed.

2. Enter the name of the new component as **Core** in the **File name** display area and choose the **Save** button from the **Save As** dialog box.

3. Select the **Front** plane from the **FeatureManager Design Tree** to place the new component.

   The sketcher environment is invoked.

4. Exit the sketcher environment.

5. Using the **Radiate Surface** tool, create the radiated surface using the outer edges of the bottom face of the Cover.

6. Now, using the **Knit Surface** tool, knit all the inner surfaces of the Cover with the radiated surface to create a single knitted surface.

7. Hide the Cover and create a new plane offsetted vertically downwards at a distance from the radiated surface.

8. Select the newly created plane as the sketching plane and invoke the sketching environment.

9. Create the sketch of the core and extrude it up to the knitted surface.

10. Select the bottom face of the core and invoke the sketching environment.

11. Create the sketch of the core plate and extrude it to the required depth.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

12. Now, return to the assembly environment and save the assembly.

    The final core is shown in Figure 17-59.

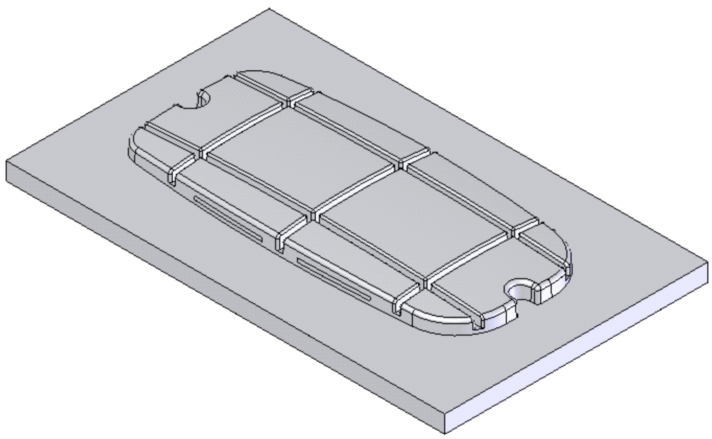


Figure 17-59 *Final core*

Extracting the Cavity

After extracting the core, you need to the extract the cavity. The cavity will also be extracted using the same procedure that was used to extract the core. For this, you first need to create a new component in the assembly.

1. Hide the Core and unhide the Cover.
2. Choose **Insert** > **Component** > **New Part** from the menu bar. The **Save As** dialog box is displayed.
3. Enter the name of the new component as **Cavity** in the **File name** area and choose the **Save** button from the **Save As** dialog box.
4. Place the new component on the **Front** plane.
5. Exit the sketcher environment.
6. Create the radiated surface and knit the radiated surface with the outer surfaces of the part.
7. Now, create a plane vertically upwards at an offset distance of 10mm from the radiated surface.
8. Select the newly created plane and invoke the sketching environment.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

9. Create the sketch of the cavity and then extrude the sketch up to the knitted surface.

10. Add the remaining features to create the final Cavity Plate, see Figure 17-60.

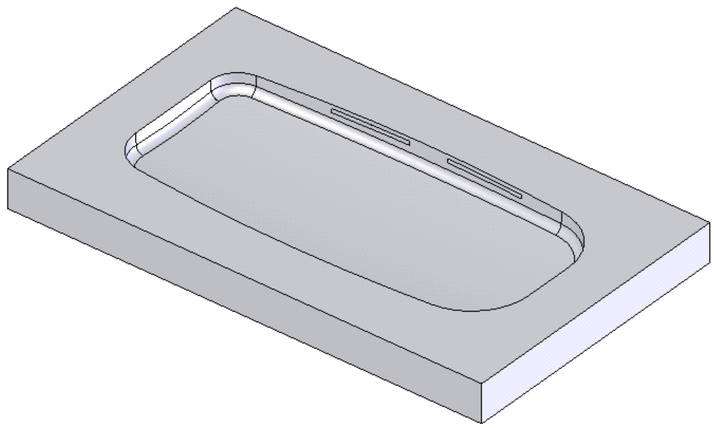


Figure 17-60 *Cavity displayed after hiding the Cover*

Next, you need to create the slots in the cavity that will be used to place the inserts. The slots will be created by extruding a sketch created on the front face of the cavity using the cut option. The sketch of the slots will be created by using the **Convert Entities** tool on the edges of the slots of the Cover.

10. Select the front face of the cavity as the sketching plane and invoke the sketching environment.

11. Choose the **Convert Entities** tool and select the edges of two slots displayed on the front face of the Cover.

12. Invoke the **Cut-Extrude PropertyManager** and extrude the sketch using the **Through All** option.

13. Return to the assembly environment and save the assembly.

The final cavity after creating the slots is displayed in Figure 17-61.

Creating Insert1

After extracting the core and cavity, you need to create the insert that will be used to restrict the flow of molten material in the portion where the insert is placed. This in turn will create the slot in the molded plastic component. The insert will also be created as a separate part using the top-down approach.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

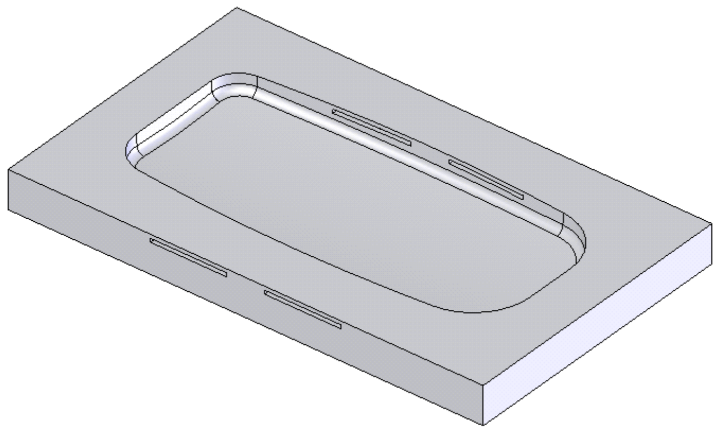


Figure 17-61 *Cavity after creating the slots*

1. Choose **Insert > Component > New Part** from the menu bar. The **Save As** dialog box is displayed.

2. Enter the name of the new component as **Insert1** in the **File name** edit box and choose the **Save** button from the **Save As** dialog box.

3. Place the new component on the **Front** plane.

4. Exit the sketching environment.

5. Select the front face of the cavity as the sketching plane and invoke the sketching environment.

6. Using the **Convert Entities** tool, select the edges of the slots placed on the front face of the cavity.

7. Extrude the sketch up to the inner front curved face of the Cover. The assembly after creating the base feature of the Insert1 is shown in Figure 17-62.

8. Using the standard modeling tools, create the back plate of the insert.

Figure 17-63 shows the assembly after creating the insert.

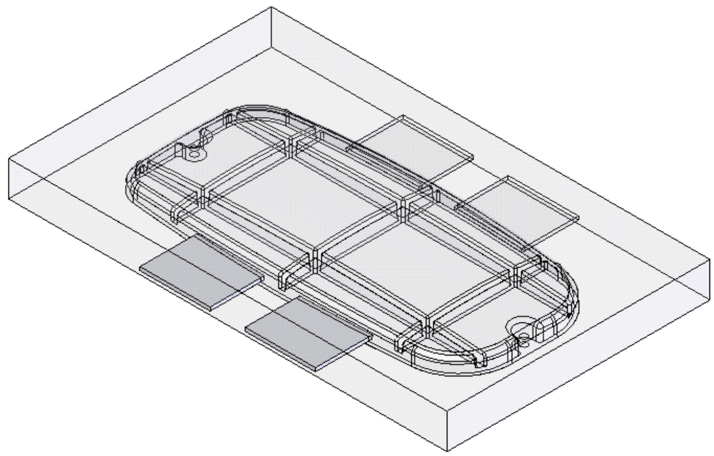


Figure 17-62 *Base feature of the Insert1*

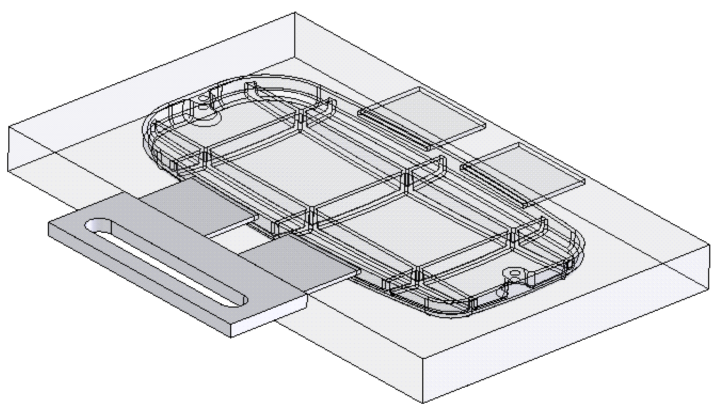


Figure 17-63 *Assembly after completing the Insert1*

Similarly, create the Insert2. The final Mold Base assembly is shown in Figure 17-64.

Next, you need to create the exploded view of the assembly, refer to Figure 17-65.

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

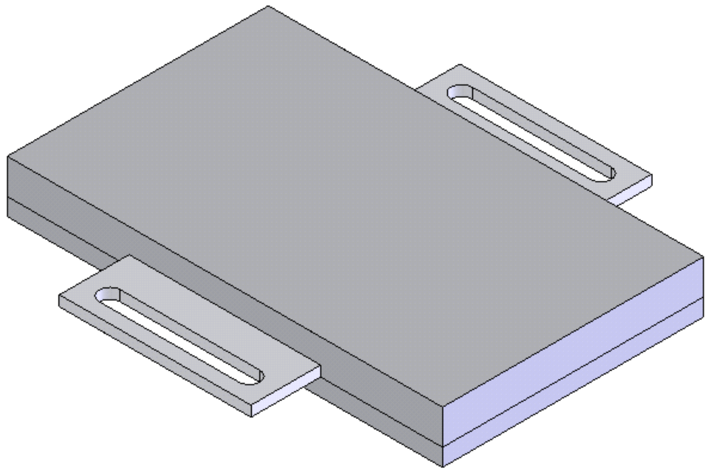


Figure 17-64 *Final Mold Base assembly*

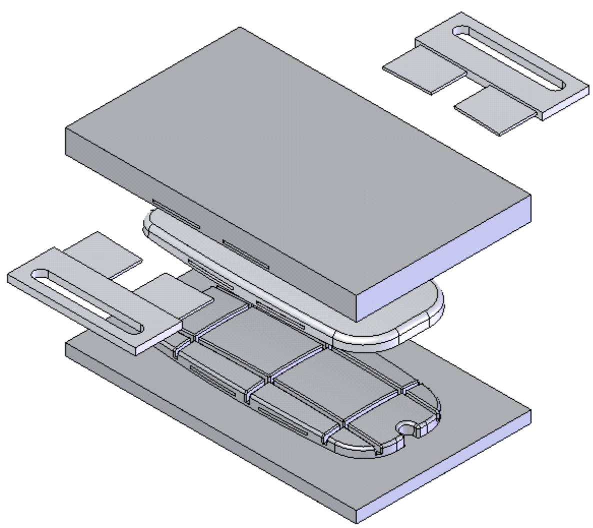


Figure 17-65 *Exploded view of the Mold Base assembly*

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with the answers given at the end of this chapter.

1. The __________ is the female part of the mold, it is hollowed from inside and the molten metal or the model plastic is poured so that it can take the shape of the cavity to create the component.

2. The __________ is the male part of the mold and it defines the area where the molten metal should not reach.

3. The cavity is created using the __________ tool.

4. The options available in the __________ drop-down list provided in the **Type** area of the **Cavity** dialog box are used to define the origin about which you need to scale the cavity.

5. For creating the derived components, choose the **File** > _________ from the menu bar.

6. If the **Mold Tools** toolbar is not available by default, then choose **View** > **Toolbars** > **Mold Tools** from the menu bar to display this toolbar. (T/F)

7. Choose the **Scale** button from the **Mold Tools** toolbar to invoke the **Scale** tool. (T/F)

8. The **Uniform scaling** check box is not selected by default and the model is uniformly scaled. (T/F)

9. The **Find steep faces** check box is selected to analyze the positive steep faces and the negative steep faces. (T/F)

10. Choose the **Calculate** button available in the **Analysis Parameters** rollout to analyze the draft angle using the **Draft Analysis PropertyManager**. (T/F)

REVIEW QUESTIONS

Answer the following questions:

1. The __________ tool is used to analyze the draft angles of the model.

2. For scaling the model, you need to invoke the __________ tool.

3. Using the options available in the __________ drop-down list, you can specify the direction along which you can scale the model.

4. The __________ is invoked to analyze the draft angles.

5. The __________ check box is selected to display color variations specifying the draft angles on the model in the **Draft Analysis PropertyManager**.

6. The **Gradual transition** check box available in the **Color Settings** rollout of **Draft Analysis PropertyManager** is selected to display color variations specifying the draft angles on the model. (T/F)

7. Using the **Scale Factor** spinner, you cannot specify the scale factor for scaling the model. (T/F)

8. The **Face classification** check box available in the **Analysis Parameters** rollout is used to refine the draft analysis depending on the type of draft. (T/F)

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

9. As you invoke the **Cavity** tool, the **Cavity** dialog box is displayed. (T/F)

10. The **Find steep faces** check box is selected to analyze the positive steep faces and the negative steep faces. (T/F)

EXERCISE

Exercise 1

In this exercise, you will extract the core and cavity of the model created in Tutorial 2 of Chapter 6. Before proceeding further, you need to modify the model by deleting the lip created as an extruded feature and then add the required draft angles to the model. Figure 17-66 shows the modified model. Figure 17-67 shows the model of the core and Figure 17-68 shows the model of the cavity. Figure 17-69 shows the exploded view of the Mold base assembly. Figure 17-70 shows the drawing views of the core and Figure 17-71 shows the drawing views of the cavity. **(Expected time: 2hrs)**

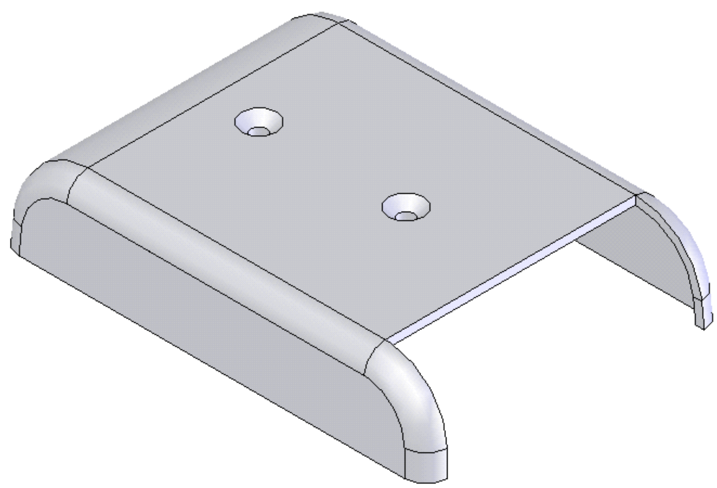


Figure 17-66 *Modified model*

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

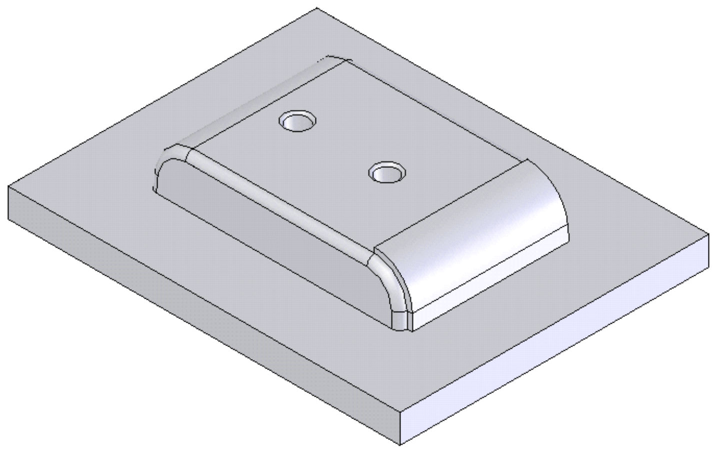


Figure 17-67 Model of core

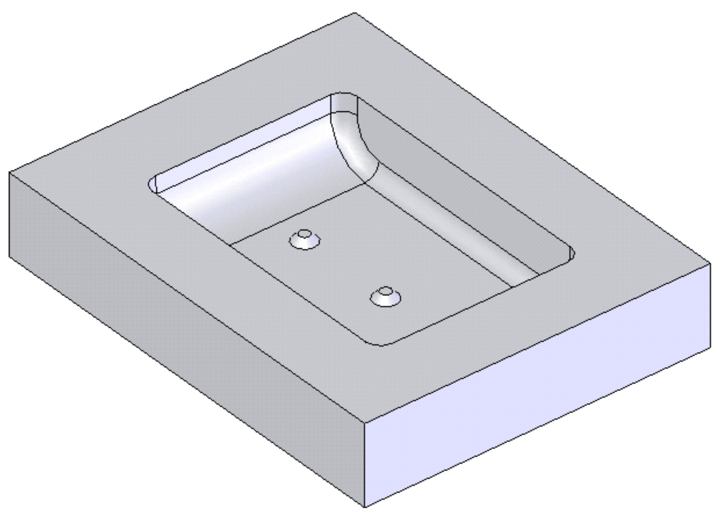


Figure 17-68 Model of cavity

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

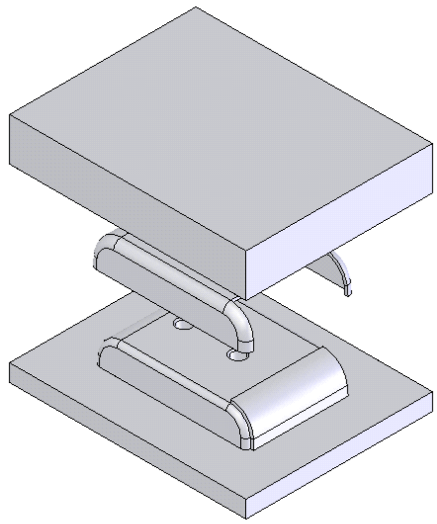


Figure 17-69 *Exploded view of the Mold Base assembly*

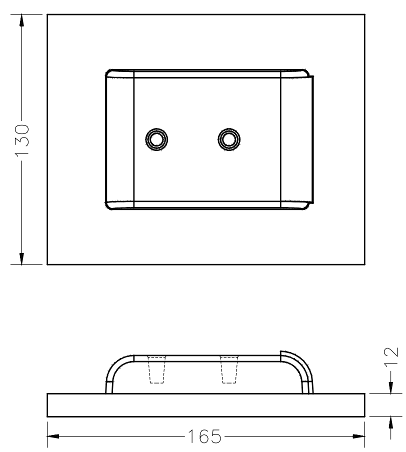


Figure 17-70 *Views and dimensions of core*

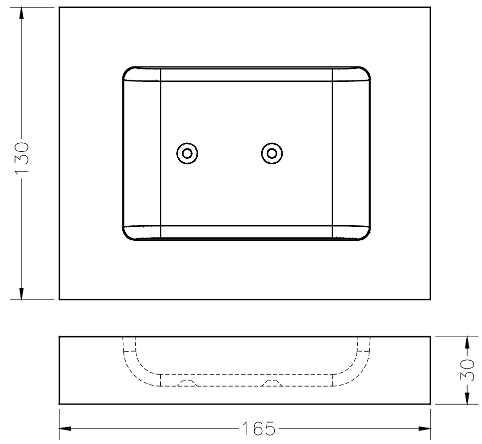


Figure 17-71 Views and dimensions of cavity

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

© CADCIM Technologies, USA. For engineering services, contact sales@cadcim.com

Answer to Self-Evaluation Test

1. cavity, **2.** core, **3. Cavity**, **4. About**, **5. Derive Component Part**, **6.** T, **7.** T, **8.** F, **9.** T, **10.** T