Autodesk Inventor for Designers: Update Guide Release 7

Learning Objectives

After completing this update guide, you will be able to:

- *Use the Split tool to split surfaces using other surfaces.*
- *Insert Word documents or Excel Spreadsheets in the sketching environment.*
- *Define units for the Inventor presentation (.ipn) files.*
- *Use Communication Center.*
- *Insert AutoCAD 2004 files in the sketching environment.*
- *Open Mechanical Desktop 2004 files with parametric models.*
- *Use hybrid modeling to create solid models.*

SPLITTING SURFACES USING SURFACES

Toolbar:	Part Features > Split
Panel bar:	Part Features > Split

Autodesk Inventor 7 allows you to split surfaces using surfaces. This is done using the **Split** tool. For example, consider the two surfaces shown in Figure UG-1. This figure shows two intersecting surfaces.

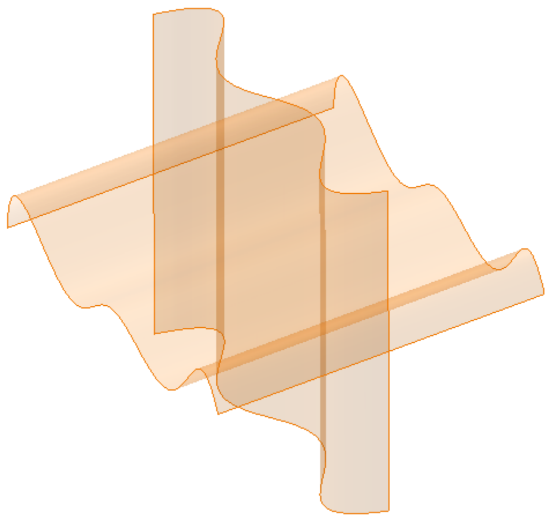


Figure UG-1 *Intersecting surfaces*

You can use the **Split** tool to split one of these surfaces into two using the other surfaces as the cutting edge, see Figure UG-2.

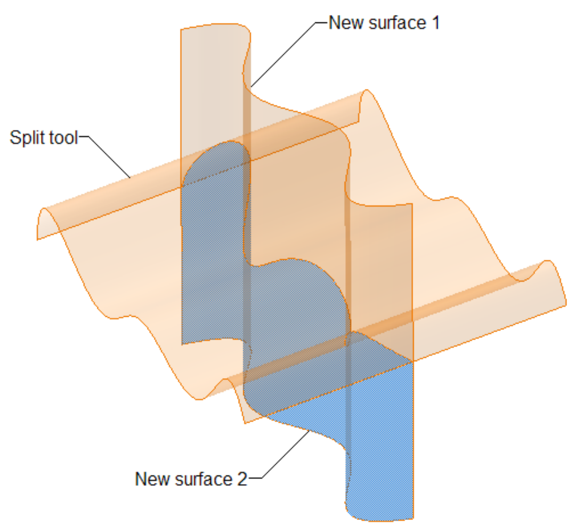


Figure UG-2 *Two split surfaces created using the selected split tool*

To do this, invoke the **Split** tool. The split dialog box is displayed and you are prompted to select a sketch profile, a work plane, or a surface as the split tool. Select the horizontal surface as the split tool. Next, select the vertical surface as the faces that will be split. Exit the dialog box. The vertical surface will be split in two using the horizontal surface. Similarly, you can split the horizontal surface using the vertical surface.

You can use the **Delete Face** tool to delete one ore more split surfaces, see Figure UG-3.

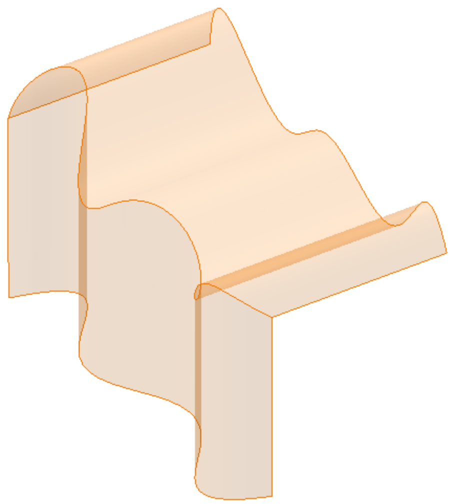
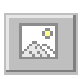


Figure UG-3 *After deleting the split surfaces*

INSERTING MS WORD OR MS EXCEL FILES IN THE SKETCHING ENVIRONMENT OF AUTODESK INVENTOR 7

Toolbar:	2D Sketch Panel > Insert Image
Panel bar:	2D Sketch Panel > Insert Image



Autodesk Inventor 7 allows you to insert MS Word or MS Excel files in the sketching environment. This is done using the **Insert Image** tool. Until the previous release of Autodesk Inventor, you were able to insert only the BMP images using this tool. However, Autodesk Inventor 7 allows you to insert MS Word (*.doc) or MS Excel (*.xls) files also in the sketching environment using this tool. Note that even the text files are inserted as images using this tool.

To insert the MS Word or Excel file, choose **Insert Image** from the **2D Sketch Panel** panel bar. The **Open** dialog box is displayed. Select **Excel Spreadsheets (*.xls)** or **Word Documents (*.doc)** from the **Files of type** drop-down list. Now, select the file that you want to insert.

Selecting the **LINK** check box in the **Open** dialog box allows you to link the original file with the image that will be inserted in the sketching environment of Autodesk Inventor 7. If the files are linked, any change made in the original file will be reflected in the Inventor image. **Note that the changes made in the original file will be reflected in the Inventor image**

only after you update it. To update the Inventor image, right-click on it and choose **Update** from the shortcut menu.

DEFINING UNITS FOR THE PRESENTATION FILES

Autodesk Inventor 7 allows you to define the units in the presentation (.ipn) file. By defining the units in the presentation file, you can control the distance and angle while specifying the tweak. To define the units in the presentation file, choose **Tools > Document Settings** from the menu bar. The **Document Settings** dialog box will be displayed as shown in Figure UG-4. The various options available in this dialog box are discussed next.

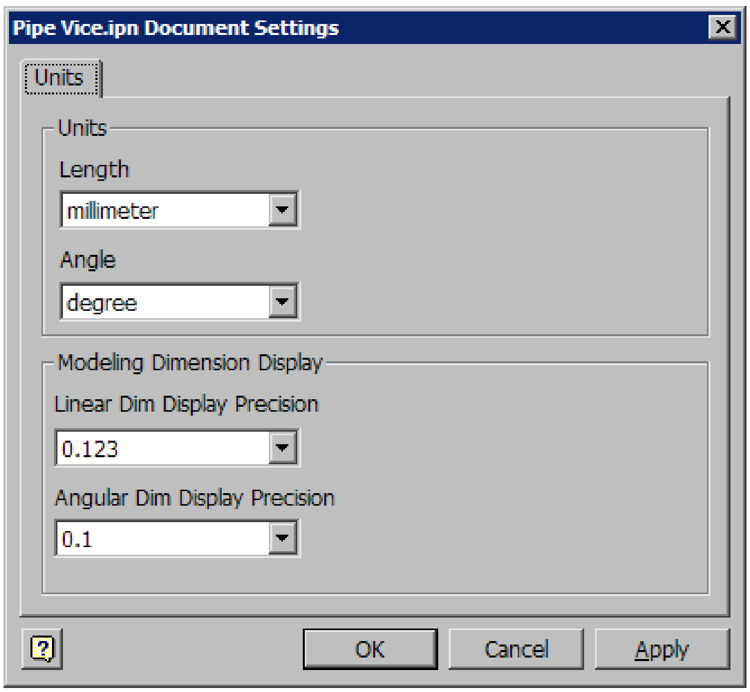


***Figure UG-4 Document Settings** dialog box*

Units Area

The options available in the **Units** area are used to specify the units for the linear and angular measurements. These options are discussed next.

Length

The **Length** drop-down list displays a list of units that you can select for measuring linear distances in the presentation file. The units that are available in this drop-down list are inch, foot, centimeter, millimeter, meter, and micron.

Angle

The **Angle** drop-down list displays a list of units that you can select for measuring angular values in the presentation file. The units that are available in this drop-down list are degree and radian.

Modeling Dimension Display Area

The options available in the **Modeling Dimension Display** area are used to specify the precision for the linear and angular units. The precision defines the number of decimal places in a numeric value. You can select the precision for the linear dimensions using the **Linear Dim Display Precision** drop-down list and the precision for the angular dimensions using the **Angular Dim Display Precision** drop-down list.

USING THE COMMUNICATION CENTER

The **Communication Center** icon is displayed on the status bar close to the bottom right corner of the Autodesk Inventor 7 window. The **Communication Center** displays a message whenever Autodesk provides the latest information regarding Autodesk Inventor updates. You can configure the settings of the **Communication Center** by choosing the **Communication Center** button. The **Communication Center** dialog box is displayed. Choose the **Settings** button to display the **Settings** tab of the **Settings** dialog box as shown in Figure UG-5.

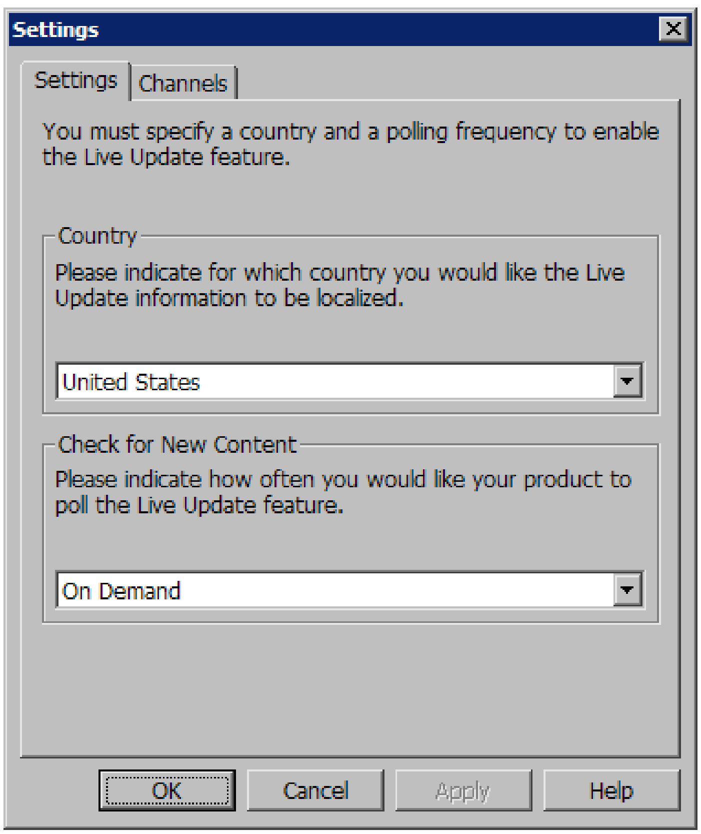


***Figure UG-5 Settings** tab of the **Settings** dialog box*

The drop-down list in the **Country** area allows you to select your country to localize the live update information. The **Check for New Content** area allows you to select the option that will force the **Communication Center** to check for the updates and the latest information provided by Autodesk daily, weekly, monthly, or on demand.

The **Channels** tab is used to select the information channels from which the **Communication Center** will download the updates and information. Note that this tab is available only after you download the updates or information at least once using the **Communication Center**.

After you have set the country settings, exit the **Settings** dialog box and make sure that you are connected to the internet. Choose the **Refresh Content** button. The **Communication Center** will search for all the latest updates and information about Autodesk Inventor 7. The **Communication Center** dialog box after downloading the information about latest updates is shown in Figure UG-6.

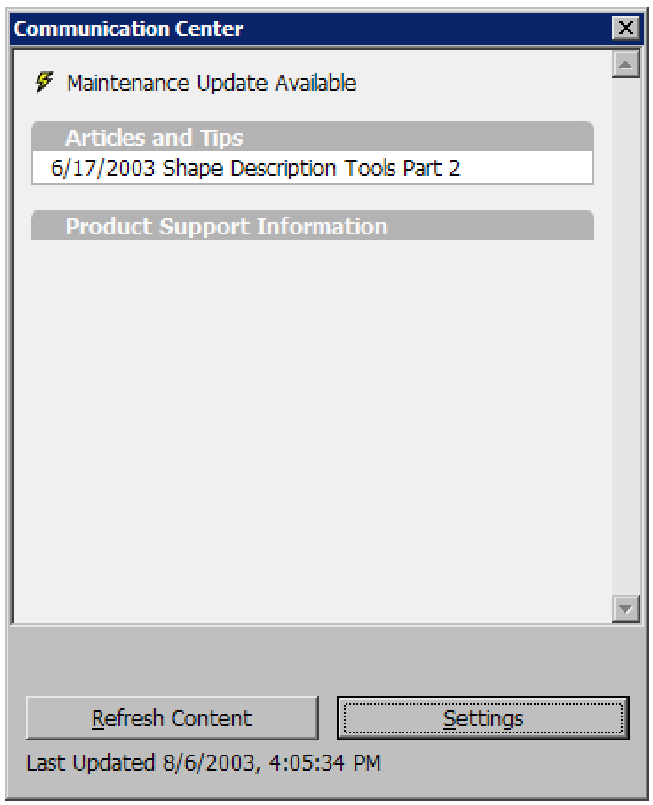
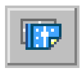


***Figure UG-7 Communication Center** dialog box after downloading the information about latest updates*

INSERTING AN AutoCAD 2004 FILE IN THE SKETCHING ENVIRONMENT OF AUTODESK INVENTOR 7

Toolbar:	2D Sketch Panel > Insert AutoCAD File
Panel bar:	2D Sketch Panel > Insert AutoCAD File



Autodesk Inventor 7 allows you to place AutoCAD 2004 files as 2D entities in the sketching environment. Remember that the drawing will be placed as a 2D sketch even if the AutoCAD 2004 file has 3D entities or solid models. To open an AutoCAD 2004 file in the sketching environment, choose **Insert AutoCAD File** from the **2D Sketch Panel** panel bar. The **Open** dialog box is displayed. You can use this dialog box to specify the name and the location of the AutoCAD 2004 file. Select the file that you want to insert in the sketching environment of Autodesk Inventor and choose **Open**. The **Layers and Objects Import Options** dialog box is displayed as shown in Figure UG-8.

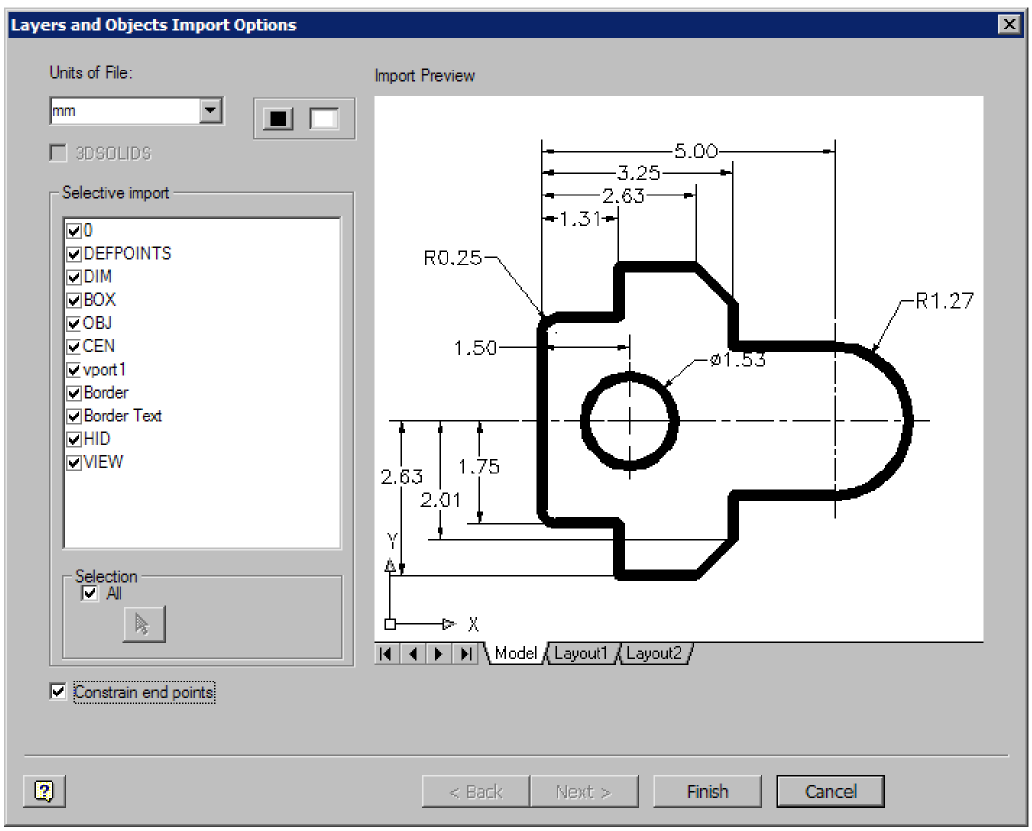


***Figure UG-8 Layers and Objects Import Options** dialog box*

The preview window of this dialog box is similar to the original AutoCAD 2004 screen. It has the model and the layout tabs below the drawing window. You can select these options to preview the items in the different layouts of the selected AutoCAD 2004 file. Remember that the data will be imported from the selected tab. You can right-click in the preview area to display a shortcut menu that has different drawing display commands of AutoCAD 2004.

You can set the units in which the drawing will be inserted in the sketching environment of Autodesk Inventor using the **Units of File** drop-down list. Using the buttons available on the right of this drop-down list, you can set the background color of the model and layout tabs.

The **Selective import** area is used to select the layers from which the data will be imported. The **Selection** area is used to select the entities in the drawing to import. By default, the **All** check box is selected. As a result, all the entities are selected to be imported. You can clear this check box and then manually select the entities to import from the preview window. You can use any object selection method like window or crossing to select the entities.

The **Constrain end points** check box is selected to constrain the endpoints of the imported entities.

After setting the parameters in this dialog box, choose **Finish**. The AutoCAD 2004 drawing will be imported in the sketching environment of Autodesk Inventor 7. Figure UG-9 shows the Autodesk Inventor 7 sketching environment after importing the AutoCAD 2004 file.

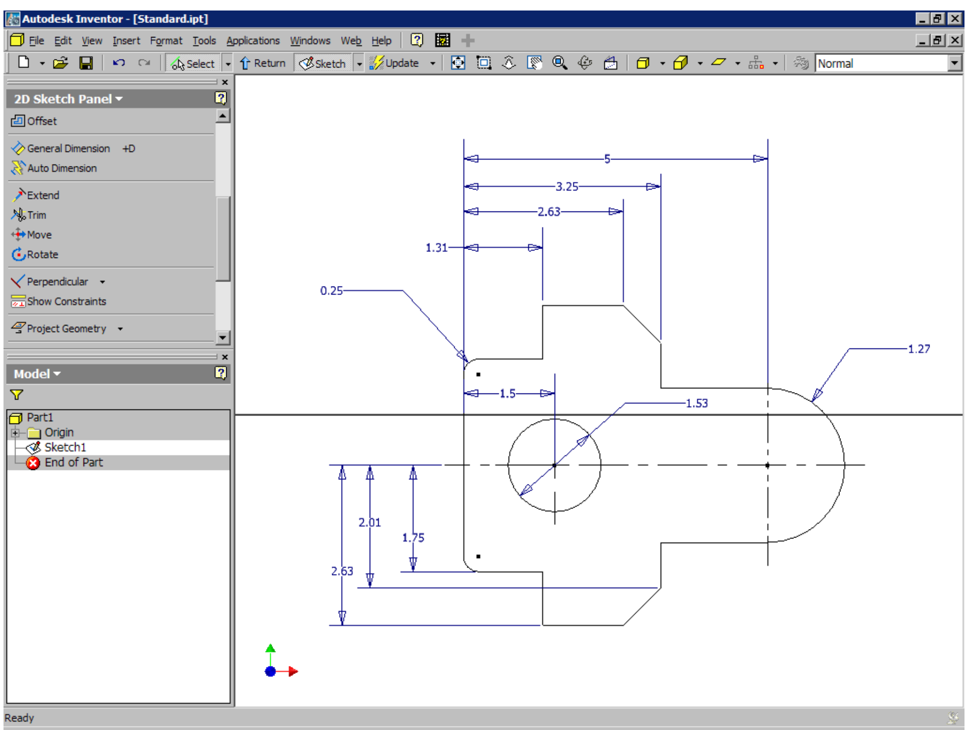


Figure UG-9 *Sketching environment after importing the AutoCAD 2004 drawing*

Note

Make sure that the file you are trying to insert in the sketching environment of Autodesk Inventor is not currently opened in AutoCAD. If it is, you need to first close the AutoCAD file and then import it in the Autodesk Inventor sketching environment.

OPENING A MECHANICAL DESKTOP 2004 FILE WITH PARAMETRIC MODEL IN AUTODESK INVENTOR 7

Autodesk Inventor 7 allows you to directly open a Mechanical Desktop 2004 file with a parametric solid model in Autodesk Inventor. The Mechanical Desktop 2004 model will be opened such that all the features of the model are imported, as they are, in Inventor 7. **However, make sure that if you want to open an MDT 2004 model with features, a session of Mechanical Desktop 2004 is currently opened**.

To open a Mechanical Desktop 2004 parametric model with features in Autodesk Inventor 7, start Mechanical Desktop 2004 and then start Autodesk Inventor 7. In the **Open** dialog box of Autodesk Inventor, choose **Open** from the **What To Do** area and then select **DWG Files (*.dwg)** from the **Files of type** drop-down list. Now, select the Mechanical Desktop 2004 file that you want to open. The **DWG File Import Options** dialog box is displayed, as shown in Figure UG-10.

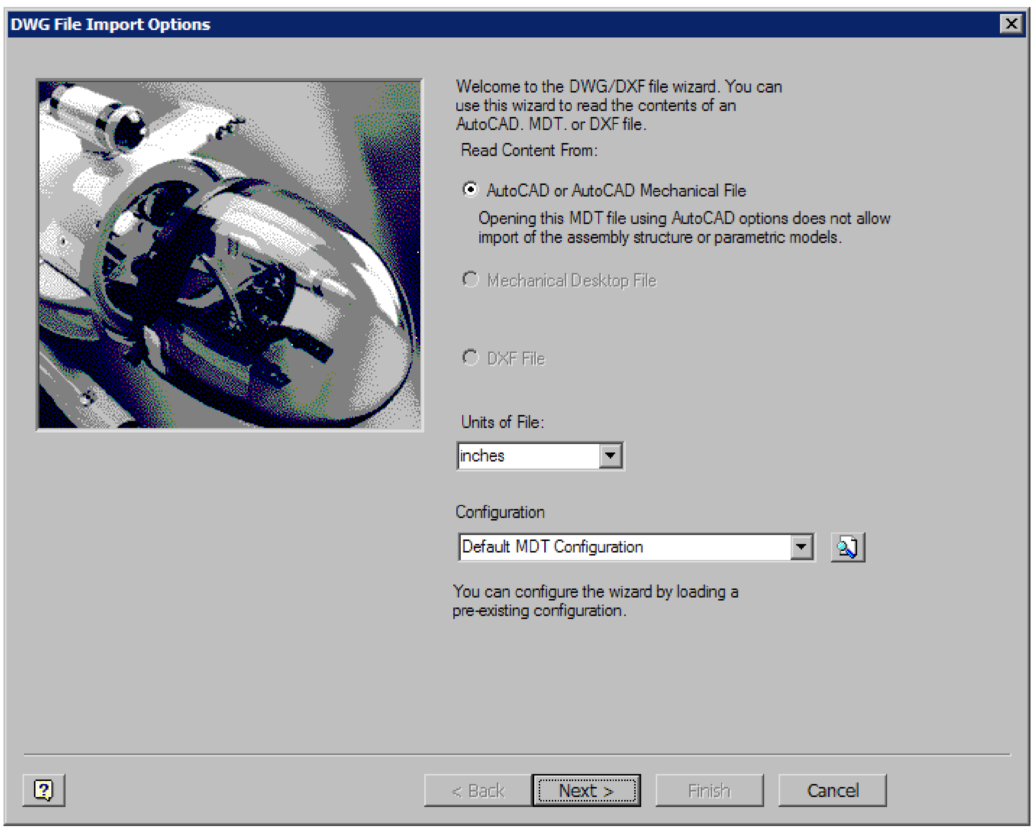


*Figure UG-10 **DWG File Import Options** dialog box*

The **Mechanical Desktop File** radio button is selected in the **Read Content From** area. Choose **Next** to display the **MDT Model/Layout Import options** dialog box, see Figure UG-11. The parametric model of Mechanical Desktop 2004 is displayed in the preview window. Choose

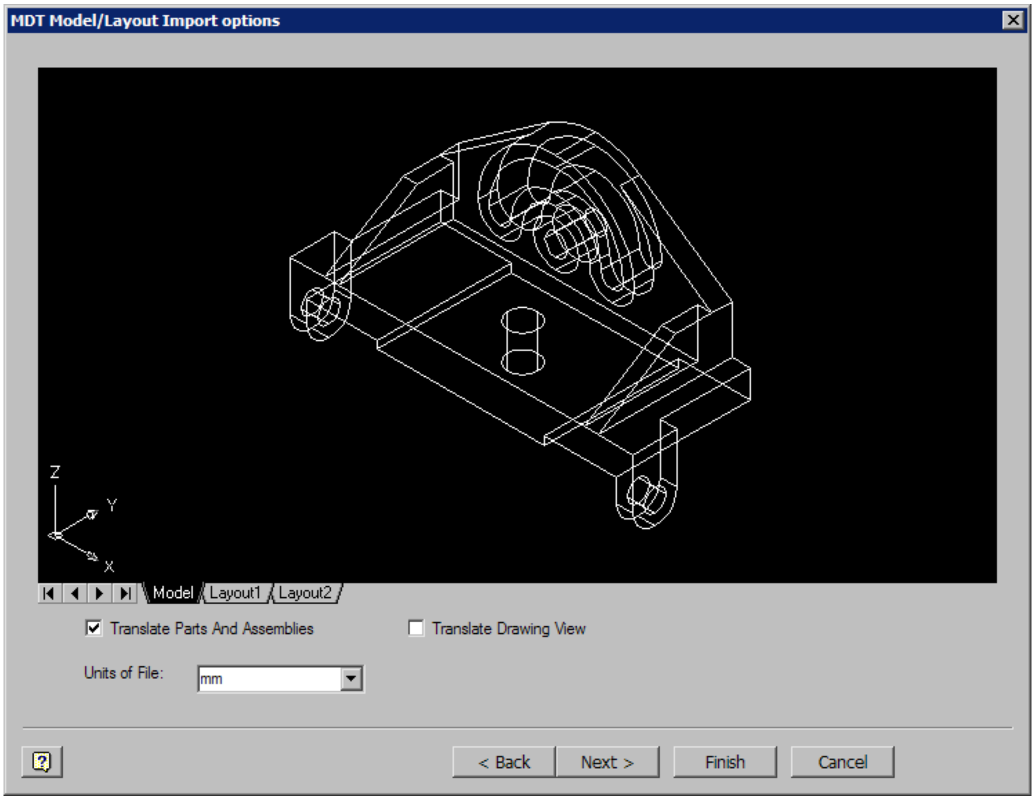


***Figure UG-11 MDT Model/Layout Import options** dialog box*

the **Model** tab if it is not chosen by default. You can right-click in the preview window to invoke the drawing display tools to view the model. Select the **Translate Parts And Assemblies** check box and then select the units for the model from the **Units of File** drop-down list.

Choose the **Next** button, the **MDT Import Destination Options** dialog box is displayed. Set the parameters in this dialog box, as shown in Figure UG-12. Choose the button available in the **Destination Folder** area and select the destination to save the Autodesk Inventor 7 part file.

Choose **Finish**, The **Multiple Part Definitions** dialog box will be displayed. Select the name of the Mechanical Desktop 2004 file that you selected to open and then choose **OK**. Autodesk Inventor 7 will start converting the Mechanical Desktop 2004 data into Autodesk Inventor 7 data.

Note

*If the **Server Busy** dialog box is displayed when you are opening the Mechanical Desktop 2004 file, choose **Switch To** and then invoke the Mechanical Desktop 2004 window. In this window, press ESC. Autodesk Inventor continues to load the file*

After the translation is complete, Autodesk Inventor opens the Mechanical Desktop 2004

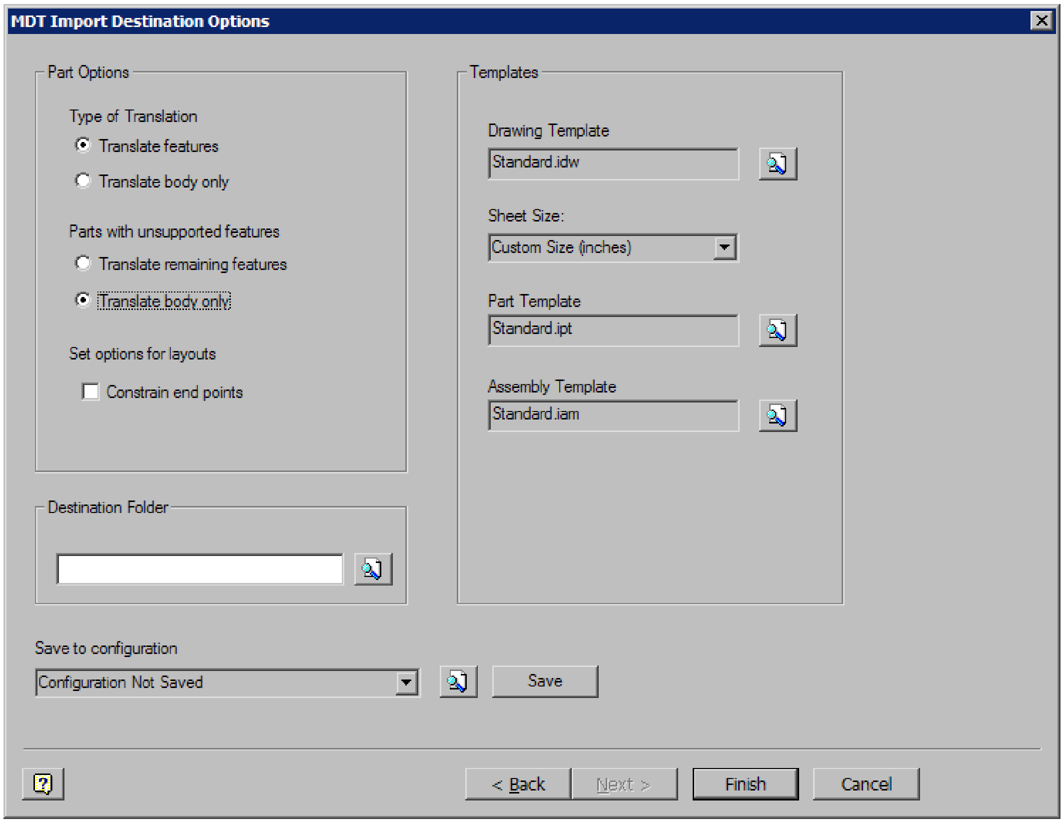
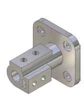


***Figure UG-12 MDT Import Destination Options** dialog box*

file in the assembly modeling environment. When you import the Mechanical Desktop file in Autodesk Inventor, a part file (.ipt) of the selected file is created. The name of the part file will have the name of the Mechanical Desktop 2004 file followed by PART1 as suffix. But it will be opened in an assembly file (.iam) of Autodesk Inventor. The entire data related to the parametric features of the Mechanical Desktop solid model will be retained in Autodesk Inventor also. To view these features, activate the imported part file by using the browser. The features and their sequence in the model will be displayed in the browser. Notice the features in the browser. The features are in the same order in which they were created in Mechanical Desktop 2004. If you double-click on any of the feature, its dimensions will be displayed on the graphics screen. This suggests that each and every dimension related to the solid model in Mechanical Desktop will be imported in Autodesk Inventor. If you want, you can save the assembly file also.



Note

*The default material that is applied to the MDT 2004 model in Autodesk Inventor 7 is ACAD-Default. You can apply any other material by selecting the model and then specifying the material using the drop-down list provided in the **Inventor Standard** toolbar.*

USING HYBRID SURFACE-SOLID MODELING TO CREATE MODELS

With the enhancements in the Shape Manager toolset, Autodesk Inventor 7 allows you to use hybrid surface-solid modeling to create complex solid models. The following tutorial explains

Tutorial 1

the use of hybrid surface-solid modeling.

In this tutorial, you will use the hybrid surface-solid modeling to create the model shown in

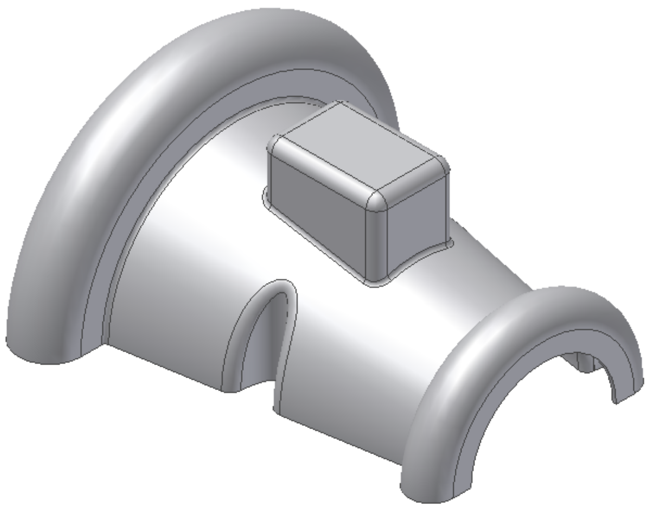


Figure UG-13 *Model for Tutorial 1*

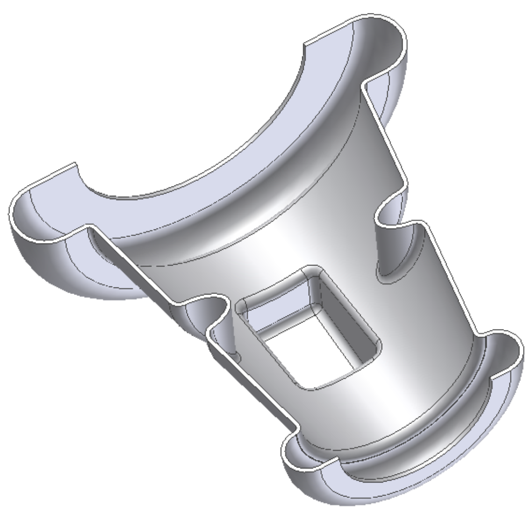


Figure UG-14 *Alternate view of the model for Tutorial 1*

Figures UG-13 and UG-14. The dimensions of this model are provided in the tutorial steps. The following steps outline the procedure of creating this model.

a. Open a new part file and then create the sketch shown in Figure UG-15 on the default XY plane.
b. Revolve this sketch through an angle of 180-degree such that the final output is a surface, as shown in Figure UG-16.
c. Create two extruded surfaces as shown in Figure UG-18.
d. Using the **Face Split** option, split the surfaces as shown in Figure UG-19.
e. Using the **Delete Face** tool, delete the faces shown in Figure UG-22.
f. Define a new work plane at an offset distance of 45 mm and then create the sketch shown in Figure UG-23.
g. Using the last sketch, split the base surface and then delete the face as shown in Figure UG-24.
h. Create a 3D sketch on the deleted face and then share the sketch used in splitting the base surface. Using the shared sketch and the 3D sketch, create a lofted surface as shown in Figure UG-26.
i. Create another extruded surface to close the top face of the lofted surface, as shown in Figure UG-27.
j. Stitch all the surface together and fillets all the sharp edges as shown in Figure UG-29.
k. Thicken the surface using the **Thicken/Offset** tool as shown in Figure UG-30.

Creating the Base Surface

The base surface will be created using a sketch drawn on the XY plane. This sketch will be revolved through an angle of 180-degree.

1. Open a new metric part file and then draw the sketch shown in Figure UG-15 on the XY plane. Note that a sketch point is placed at the origin and is fixed using the **Fix** constraint. The sketch is dimensioned using this point also to make it a fully constrained sketch. Remember that a sketch is fully constrained only when all its entities turn blue in color.

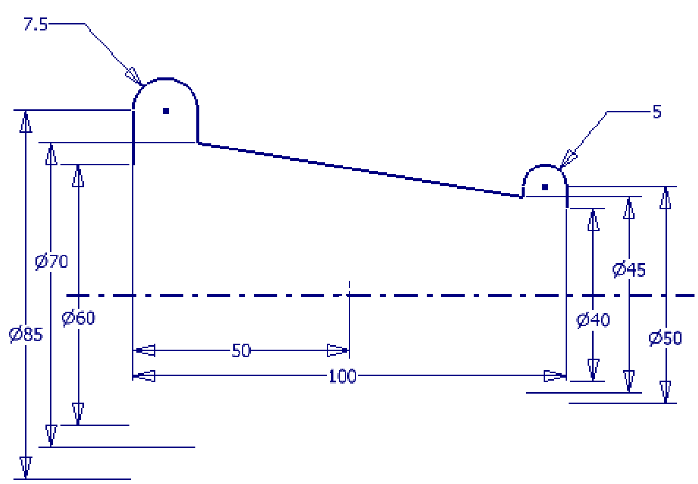


Figure UG-15 *Fully constrained sketch for the base surface*

2. Exit the sketching environment and then invoke the **Revolve** tool. The **Surface** button is automatically chosen in the **Output** area and you are prompted to select the profile.

3. Select the original sketch and then select the centerline as the axis of revolution. The preview of the resultant surface is displayed.

4. Select **Angle** from the drop-down list in the **Extents** area and set the value of the angle to 180-degree. Choose **OK**, the base surface is created as shown in Figure UG-16.

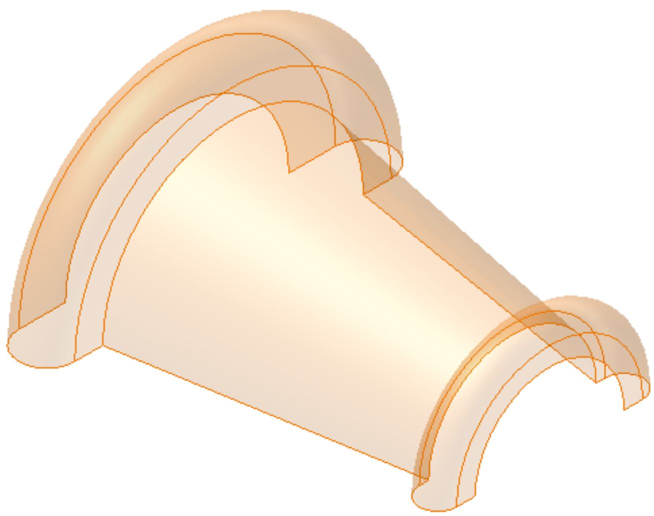


Figure UG-16 *Base surface*

Creating the Side Surfaces

The two side cuts in the base surface are created by splitting the base surface using the two surfaces at the sides. Therefore, you first need to create the side surfaces.

1. Define a new sketch plane on the XY plane and then create two ellipses as shown in Figure UG-17. Note that the center points of the two ellipses are made coincident with the edges of the base surface. Also, as the bottom ellipse is a mirror image of the upper ellipse, you need to dimension only one of the ellipses.

   Next, you need to extrude the two ellipses to create surfaces. However, when you are extruding the sketches as surfaces, you cannot select more than one sketch. In this case, there are two separate ellipses, resulting in two sketches. Therefore, you need create one surface and then share the sketch to create the other.

2. Exit the sketching environment and then extrude the lower ellipse as surface through a distance of 30 mm.

3. Share the previous sketch using the browser and then extrude the upper ellipse as surface through a distance of 30 mm.

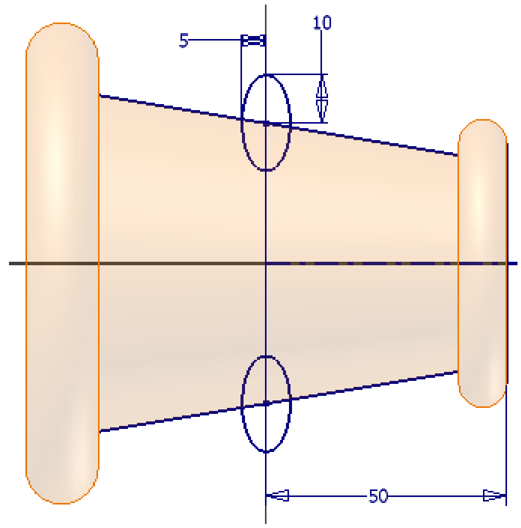


Figure UG-17 *Fully dimensioned sketch for the side surfaces*

4. Turn off the visibility of the shared sketch using the browser. Figure UG-18 shows the model after creating the two side surfaces.

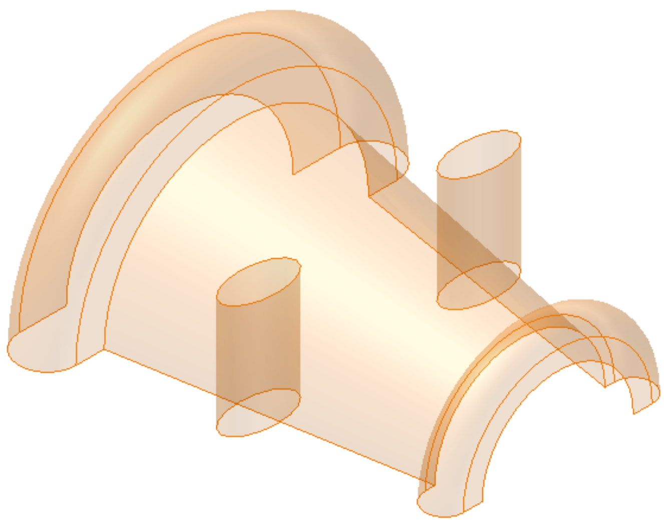


Figure UG-18 *Model after creating the side surfaces*

Creating Side Cuts Using the Side Surfaces

To create the side cuts, the two side surfaces will first be used to split the base surface. Next, the base surface will be used to split the side surfaces. Finally, all the unwanted surfaces will be deleted using the **Delete Face** tool.

1. Invoke the **Split** tool and select the lower extruded surface as the split tool. Select the base surface as the surface that will be split. Choose **OK**.

2. Similarly, using the upper extruded surface as the split tool, split the base revolved surface.

3. Now, using the base surface, split the two side surfaces one by one. The model after splitting the surfaces is shown in Figure UG-19.

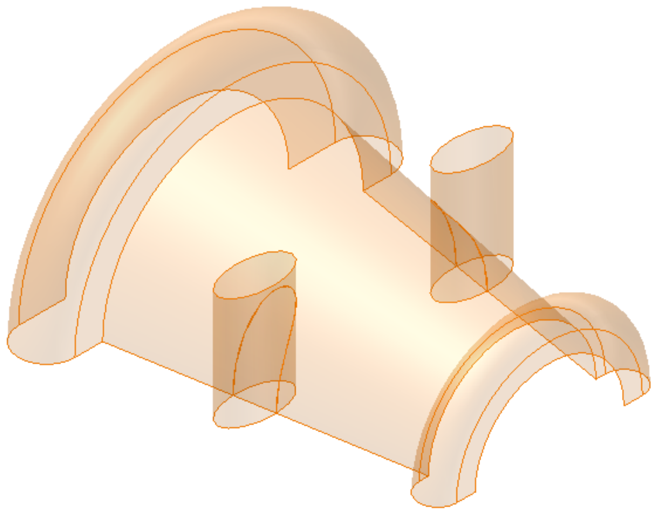


Figure UG-19 *Model after splitting the surfaces*

Next, you need delete the unwanted surfaces using the **Delete Face** tool.

4. Invoke the **Delete Face** tool and then select the outer part of the lower extruded surface to be deleted, see Figure UG-20. Note that you can delete only one face at a time using this tool.

5. Invoke the **Delete Face** tool again and delete the split surface created on the base surface, see Figure UG-21.

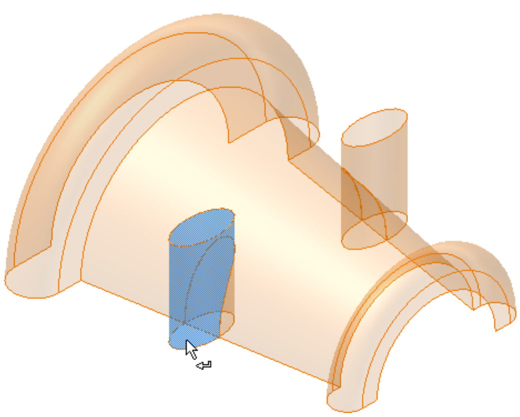


Figure UG-20 *Deleting the split surface created on the extruded surface*

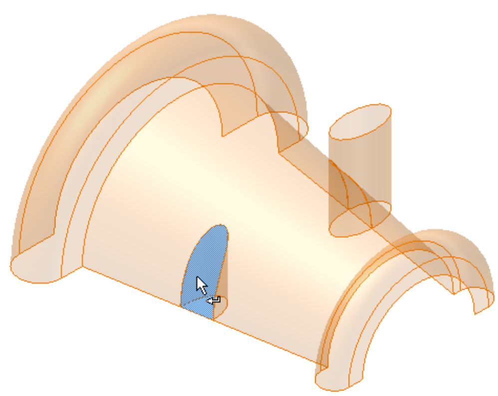


Figure UG-21 *Deleting the split surface created on the base surface*

6. Similarly, delete the unwanted surface on the other side. The model after deleting all the unwanted surface is shown in Figure UG-22.

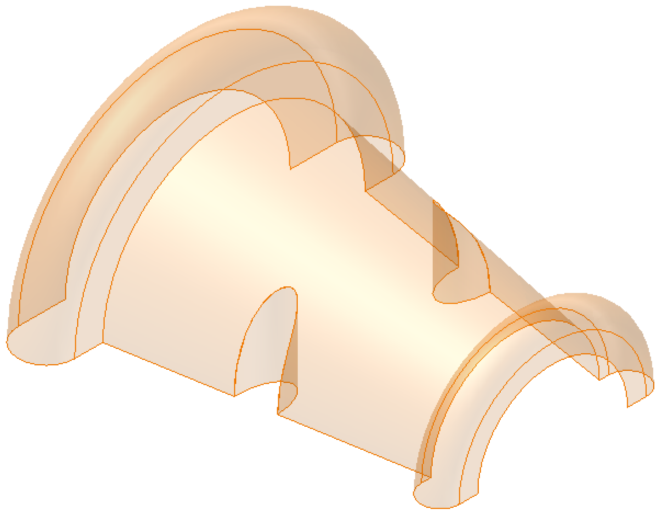


Figure UG-22 *Model after deleting the unwanted surfaces*

Creating the Lofted Surface

Next, you need to create the lofted surface. To create this surface, you first need to create a work plane at an offset distance of 45 mm from the XY plane.

1. Create a new work plane at an offset distance of 45 mm in the upward direction from the XY plane.

2. Select this plane as the sketching plane and draw the sketch as shown in Figure UG-23.

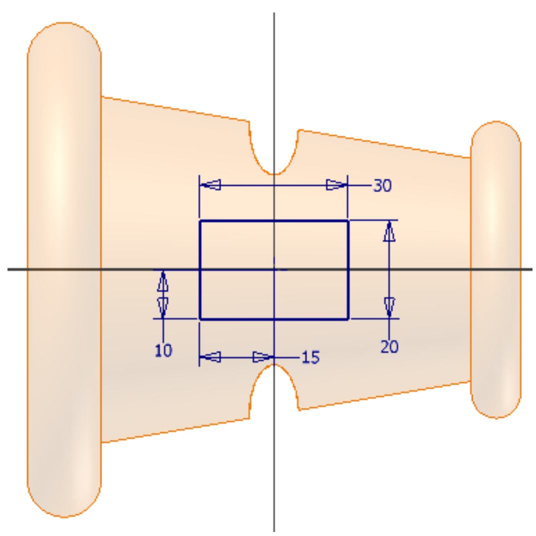


Figure UG-23 *Sketch drawn on the offset plane*

3. Exit the sketching environment and invoke the **Split** tool. Select the sketch as the split tool and select the base surface and the faces that will be split. Choose **OK**, the base surface will be split using the sketch.

4. Delete the split surface on the base surface using the **Delete Face** tool. The model after deleting the split surface is shown in Figure UG-24.

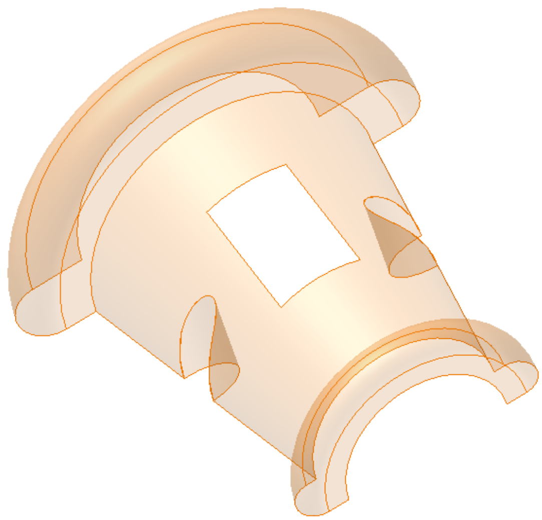


Figure UG-24 *Model after deleting the split surface*

Next, you will create a 3D sketch using the edges of the surface that is removed from the base surface.

5. Invoke the 3D sketching environment and then choose **Include Geometry** from the **3D Sketch** panel bar.

6. Select the four edges of the split surface that was removed from the base surface, see Figure UG-25.

7. Exit the 3D sketching environment and then share the rectangular sketch (refer to Figure UG-23) that was used to split the base surface.

8. Create a lofted surface between the two sketches.

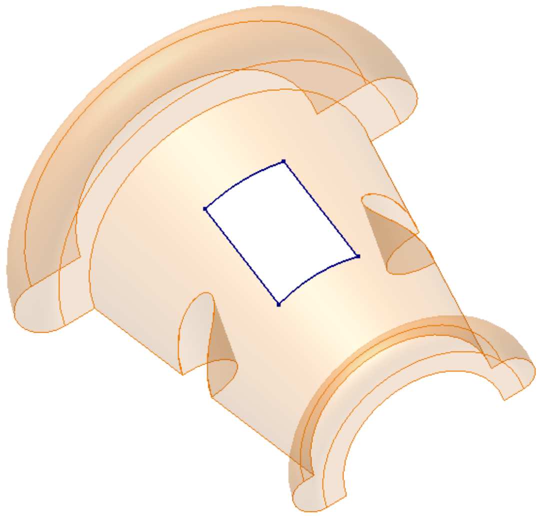


Figure UG-25 *3D sketch created using the edges*

9. Turn off the visibility of the two sketches. The model after creating the lofted surface is shown in Figure UG-26.

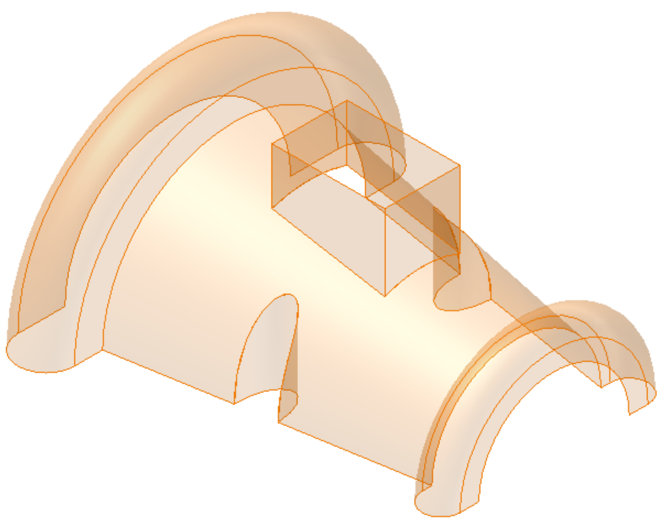


Figure UG-26 *Model after creating the lofted surface*

Next, you need to cover the top face of the lofted surface.

10. Select one of the vertical faces of the lofted surface as the sketching plane. Retain the top edge of the reference sketch and delete the remaining three edges.

11. Exit the sketching environment and then extrude the line up to the next surface. This covers the top of the lofted surface, see Figure UG-27.

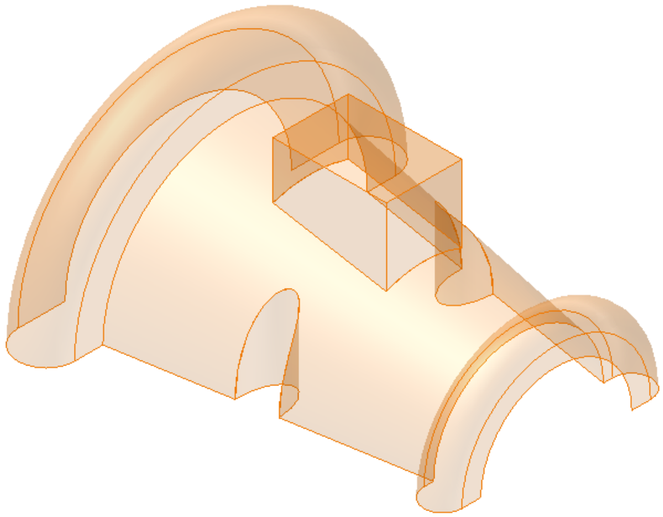


Figure UG-27 *Model after covering the top face of the lofted surface*

Stitching the Surfaces and Hiding the Original Surfaces

After creating all the surfaces, you need to stitch them together. The surfaces are stitched using the **Stitch Surface** tool.

1. Invoke the **Stitch Surface** tool and select all the surfaces so that they are stitched together. Figure UG-28 shows the model after stitching the surfaces.

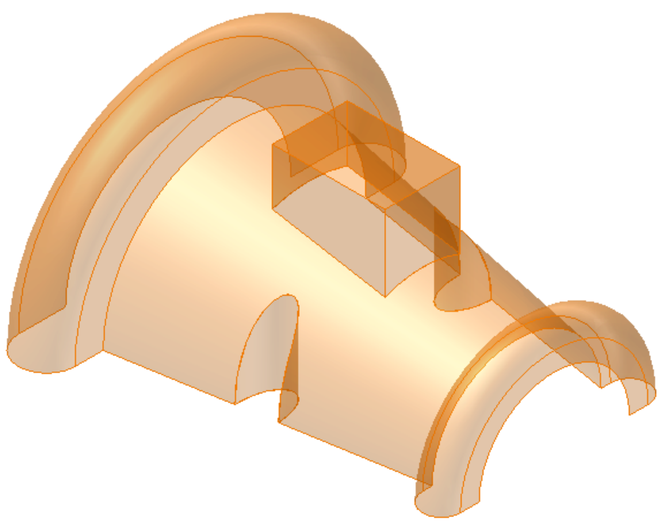


Figure UG-28 *Model after stitching the surfaces*

When you stitch the surfaces, the stitched surface is created above the original surfaces. As a result, you need to turn off the visibility of the original surfaces.

2. Using the browser, turn off the visibility of **RevolutionSrf1**, **ExtrusionSrf1**, **ExtrusionSrf2**, **LoftSrf1**, and **ExtrusionSrf3**. The stitch surface is now displayed.

3. Fillet all the sharp edges of the stitched surface with a radius of 2 mm. The surface after creating fillets is shown in Figure UG-29.

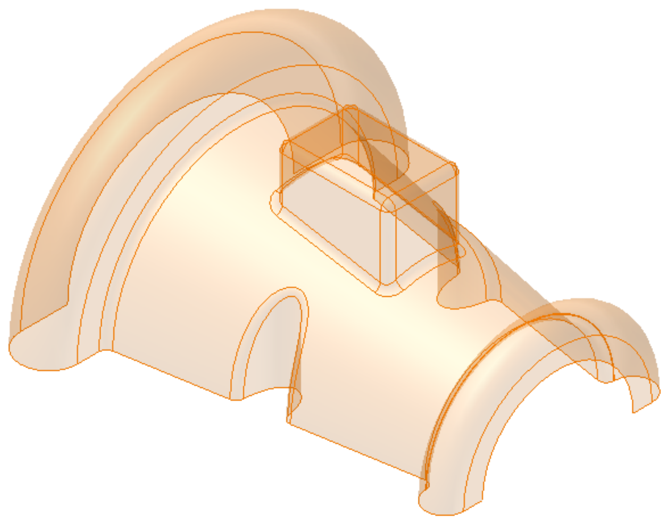


Figure UG-29 *After filleting the surfaces*

Thickening the Surface

The final step in creating the hybrid surface-solid model is to thicken the complex surface created in the previous steps. This is done using the **Thicken/Offset** tool.

1. Invoke the **Thicken/Offset** dialog box. Select the **Quilt** radio button.

2. Set the value of the **Distance** spinner to **1** and then select the stitched surface. You will notice that because you have selected the **Quilt**, the entire stitched surface is selected in a single click.

3. Turn off the visibility of the stitched surface using the browser. The final model for Tutorial 1 is shown in Figure UG-30.

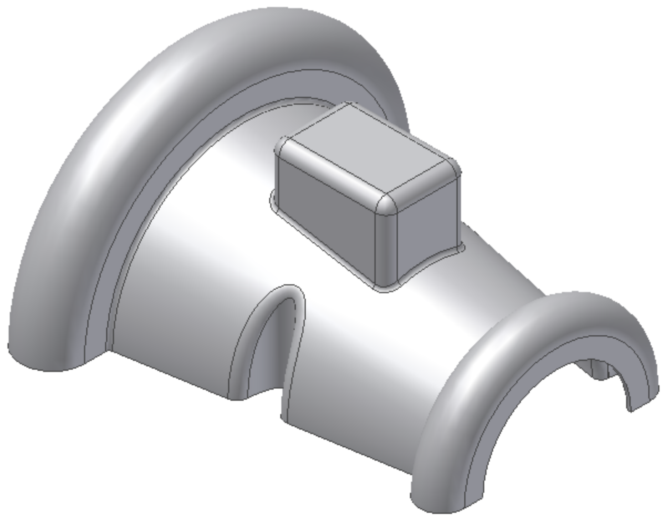


Figure UG-30 *Final hybrid model*