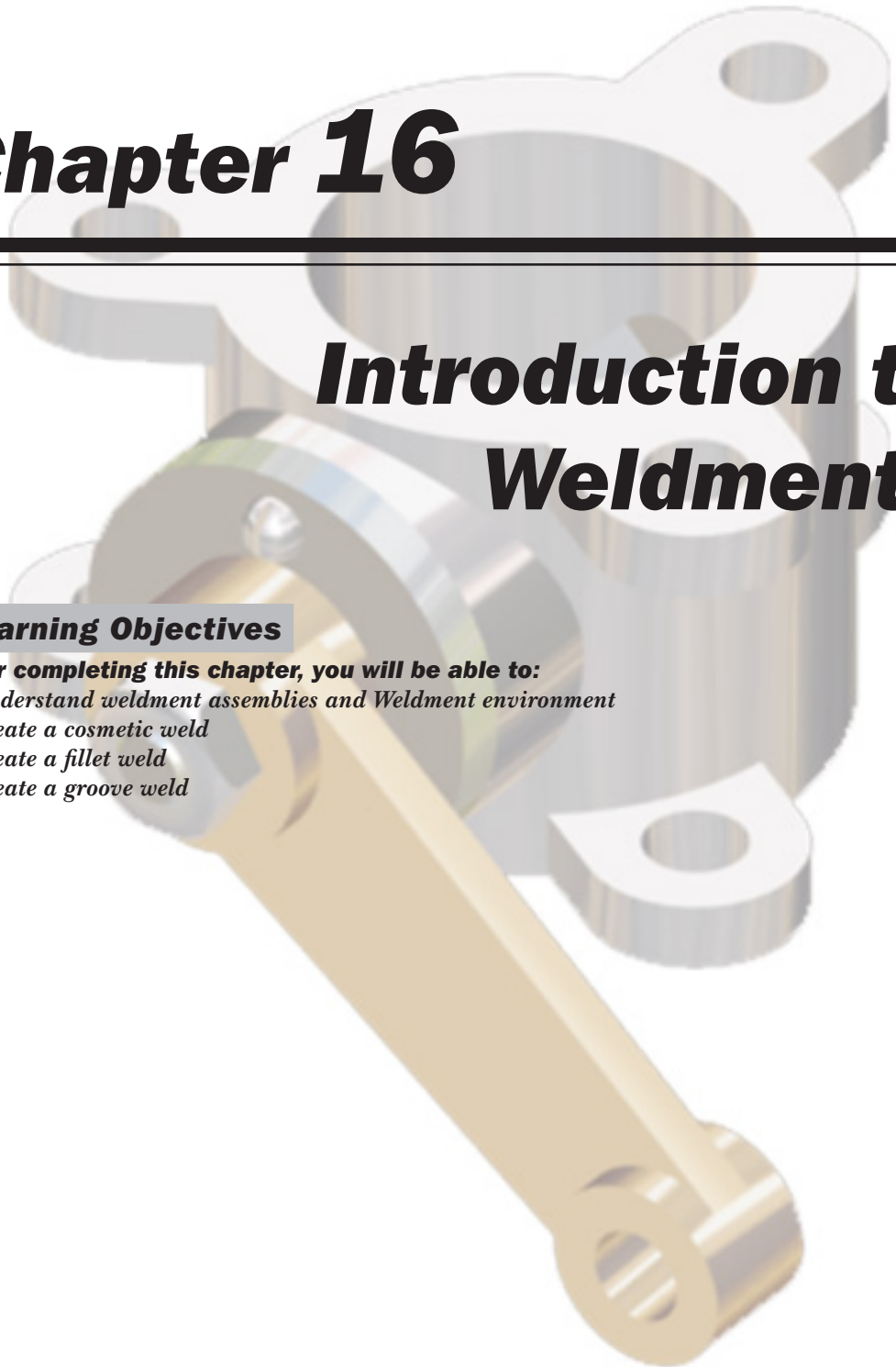
Chapter 16

Introduction to Weldments



Learning Objectives

After completing this chapter, you will be able to:

- *Understand weldment assemblies and Weldment environment*
- *Create a cosmetic weld*
- *Create a fillet weld*
- *Create a groove weld*

UNDERSTANDING WELDMENT ASSEMBLIES

Weldment assemblies are assemblies in which components are weld together. These assemblies are also called weldments. Autodesk Inventor professional provides you with a dedicated environment for creating weldments called the Weldment environment. This environment is similar to the assembly environment and provides tools to assemble components as well as to weld components. In this environment, you can also make some initial preparations to weld components. The initial preparations include creating cut features by using the tools available in the assembly environment.

Similar to various types of drawing templates, you are also provided with various weldment templates in Inventor. To invoke the Weldment assembly environment, double-click on any of the weldment assembly templates in the **Create New File** dialog box, see Figure 16-1.

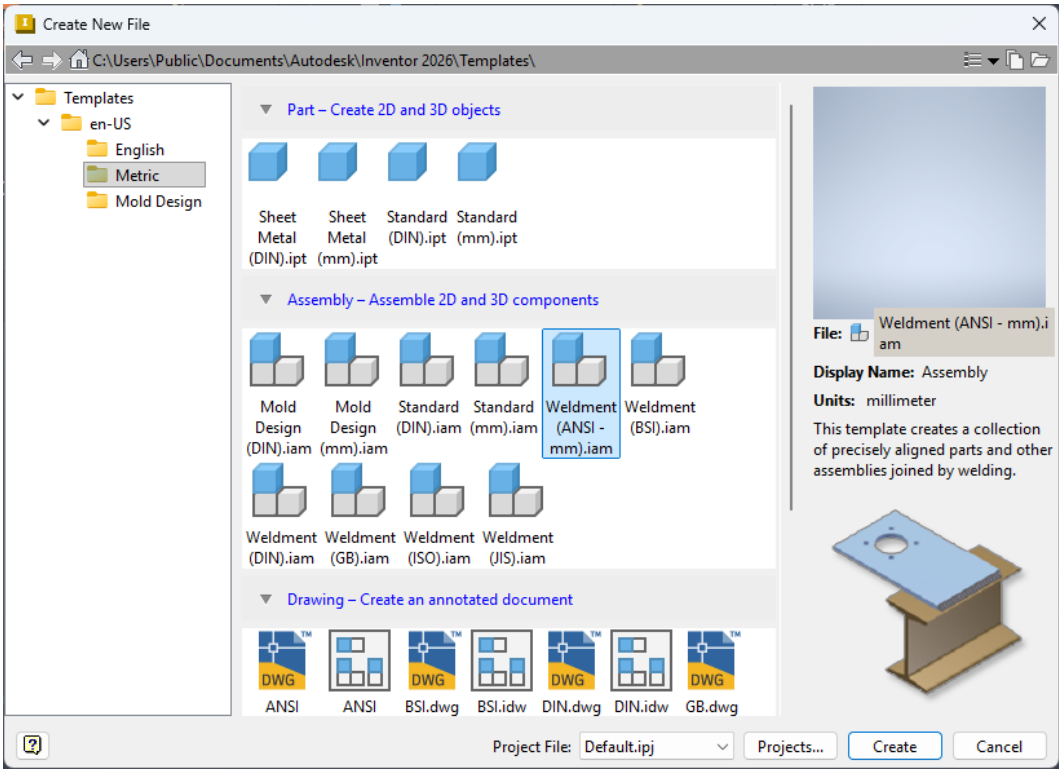


Figure 16-1 *Various types of weldment templates in the* ***Create New File*** *dialog box*

Figure 16-2 shows the Weldment assembly environment invoked using the **Weldment (ANSI - mm).iam** template.

Note

You can convert a simple assembly into a weldment assembly by using the ***Convert to Weldment*** *tool from the* ***Convert*** *panel of the* ***Environments*** *tab. However, remember that an assembly once converted into a weldment cannot be converted back into a simple assembly.*

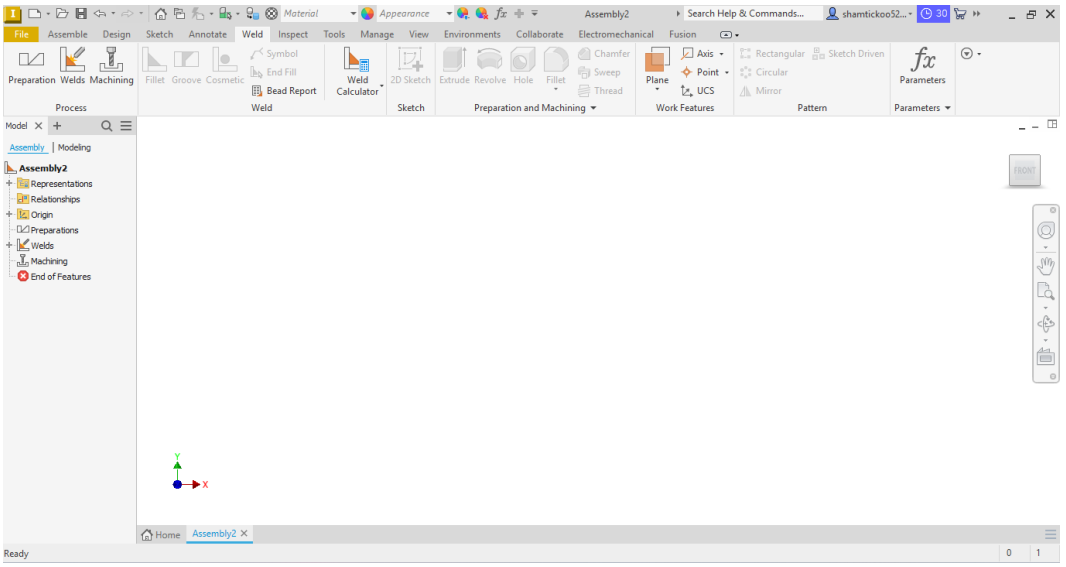


Figure 16-2 *Weldment assembly environment*

MAIN TYPES OF WELDS IN AUTODESK INVENTOR PROFESSIONAL

Autodesk Inventor allows you to create three main types of welds which are cosmetic, fillet, and groove. In addition to these, there are some other types of welds that can be used to create weldments. The three main types of welds are discussed next.

Cosmetic Welds

Cosmetic welds are artificial welds added to an edge. These types of welds are not actual welds and as a result, no weld bead is added to the model. To create a cosmetic weld, you just need to select the edge that requires welding.



Note

Adding cosmetic welds does not result in the modification of the physical properties of an assembly. This is because adding cosmetic welds does not physically add any material to a model. It only adds a convention that gives an impression of welding.

Figure 16-3 shows a part of the Shock assembly in which a cylinder is assembled with a bracket. Note that as there is no physical bonding between these two components, they cannot be held together. Therefore, you need to weld these two components together. Figure 16-4 shows the same assembly after creating a cosmetic weld. The cosmetic weld symbol is also shown in the assembly.

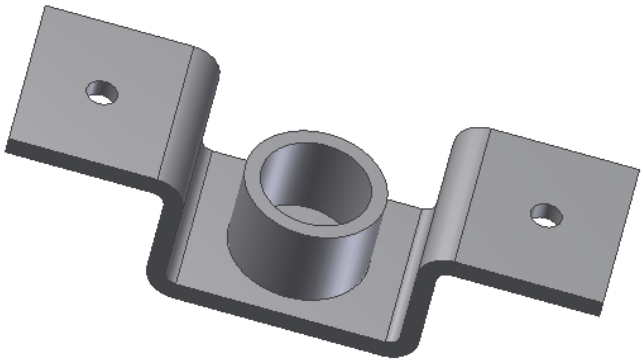


Figure 16-3 *Assembly before welding*

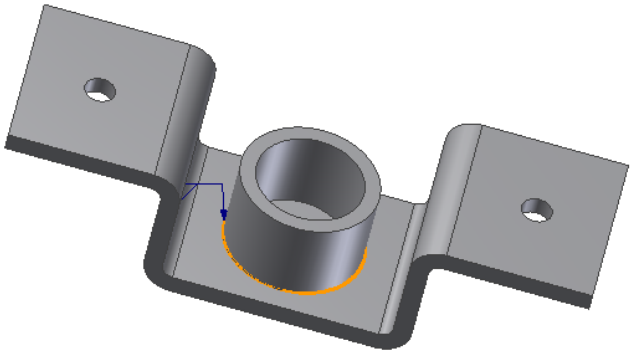


Figure 16-4 *Assembly after creating cosmetic weld*

Fillet Welds

Fillet welds are actual welds and are represented by a solid feature in an assembly. When you add fillet welds, a solid feature representing the weld bead is added in the assembly. Also, the physical properties of the assembly are modified. To create a fillet weld, you need two surfaces. Figure 16-5 shows the Shock assembly after adding a fillet weld to the bracket and cylinder. Figure 16-6 shows a butt-joint with a fillet weld.

Groove Welds

Groove welds are used to weld the parts that are placed at a certain offset from each other or have some grooves between them. These are also known as actual welds and are represented by a solid feature in an assembly. Figure 16-7 shows a partial three-quarter section view of the Shock assembly after adding a groove weld to the bracket and cylinder. Note that in this case, the cylinder is assembled at some offset from the bracket. As evident from this figure, the groove weld bead is filled in this offset space. Figure 16-8 shows a butt-joint with a groove weld. As is evident from this figure also, some offset is maintained between the two mating faces of the plates between which the groove weld bead is filled.

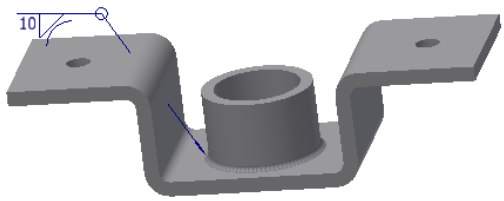


Figure 16-5 *Assembly with a fillet weld*

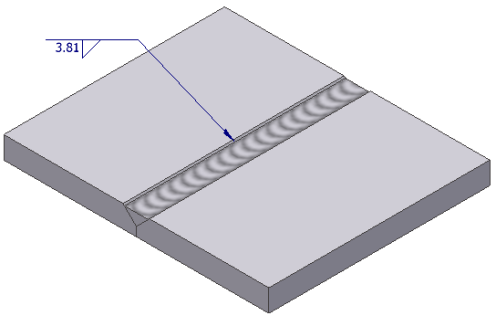


Figure 16-6 *Butt-joint with a fillet weld*

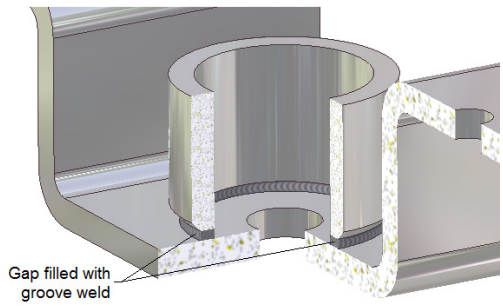


Figure 16-7 *Assembly with a groove weld*

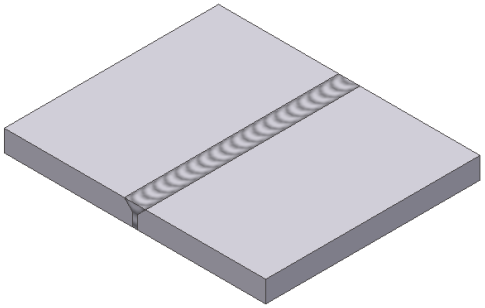


Figure 16-8 *Butt-joint with a groove weld*

ADDING WELDS TO ASSEMBLIES

The process of creating weldment assemblies is completed in three steps: assembling the components, preparing components for welding, and creating welds. These three steps are discussed next.

Assembling the Components of Weldment Assemblies

As mentioned earlier, you can create the weldment assemblies in the weldment environment. Alternatively, you can assemble the components in the assembly modeling environment and then switch to the weldment environment to add welds to the components. Remember that once you switch from the assembly modeling environment to the weldment environment, you cannot switch back.

Preparing Assemblies for Weldments

Once you have assembled components of the weldment assemblies, you need to prepare them for welding by removing material from the components to accommodate the weld beads. You can create cut features, holes, fillets, and chamfers to remove the material. For example, to create a butt-joint, you need to chamfer the two edges of the plates between which the weld bead will be added. This step would not be required if the components were chamfered during their creation. Note that similar to the assembly features, these features are also limited to the assembly and are not made on the individual part files.

To prepare weldments, choose the **Preparation** tool from the **Process** panel of the **Weld** tab. Alternatively, double-click on the **Preparations** node in the **Browser Bar**; various material removal tools will get activated in the **Weld** tab of the **Ribbon**. Next, use the required preparation tool from the **Preparation and Machining** panel of the **Weld** tab and then choose the **Return** tool from the **Return** panel to return to the Weldment environment.

Note
*While preparing weldments, the **Extrude** and **Revolve** tools provide only the **Cut** option.*

Adding Welds

The final step in creating weldments is adding welds. To do so, choose the **Welds** tool from the **Process** panel of the **Weld** tab. Alternatively, double-click on the **Welds** node in the **Browser Bar**; various weldment tools will be activated in the **Weld** panel of the **Weld** tab. You can also invoke the **Welds** tool from the Marking menu, which is displayed on right-clicking in the graphics window. You can add the fillet symbol while adding welds or later on by using the **Symbol** tool.

CREATING FILLET WELDS

Ribbon: Weld > Weld > Fillet



To create a fillet weld between components, choose the **Fillet** tool from the **Weld** panel of the **Weld** tab; the **Fillet Weld** dialog box will be displayed, as shown in Figure 16-9. The **Fillet Weld** tool can also be invoked from the Marking menu.

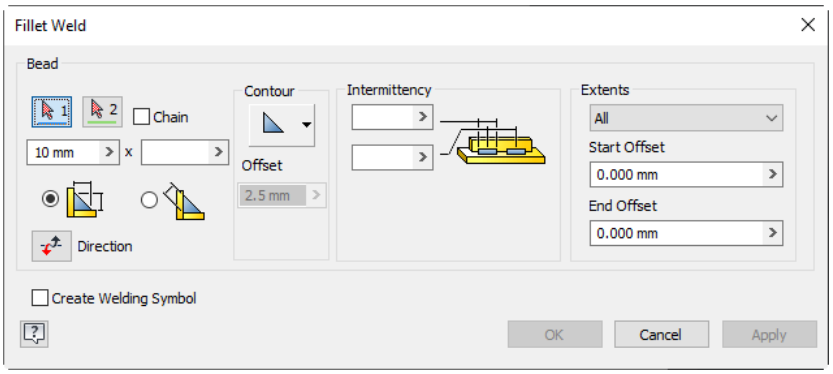


***Figure 16-9** The **Fillet Weld** dialog box*

Bead Area

To create a fillet weld, you need to select the two faces that are to be welded together. This is the reason, the **1** button in the **Bead** area will automatically be chosen in the **Fillet Weld** dialog box. Also, you will be prompted to select a face to be welded. After selecting the first face to weld, choose the **2** button; you will again be prompted to select the second face to weld. Select the second face; the second face will be highlighted and the preview of the fillet weld will appear. Selecting the **Chain** check box ensures that all the faces tangent to the selected face are also selected.

You can specify the dimension of the weld in terms of leg length or in terms of throat measurement by selecting their respective radio buttons from this area. The value of the weld leg or the throat measurement can be entered in the edit boxes below the **1** and **2** buttons. There are two radio buttons present in this area: **Leg Length Measurement** and **Throat Measurement**. If you select the **Leg Length Measurement** radio button, then you need to specify the **Leg 1** and **Leg 2** parameters of the Fillet weld in their respective edit boxes. If you select the **Throat Measurement** radio button, then you need to specify the value of the throat in the **Throat Measurement** edit box. Further you can reverse the direction of the weld by clicking on the **Direction** button.

Contour Area

The options in this area are used to specify the contour of the resulting weld bead. By default, the **Flat** button is chosen. To use any other button, click on the down arrow on the right of the **Flat** button in this area; a flyout will be displayed. Choose the **Convex** or **Concave** button to specify the required contour of weld bead. Specify the offset value of the convex or concave surface in the **Offset** edit box in this area. Figure 16-10 shows a fillet weld with convex contour.

Intermittency Area

The options in this area are used to create an intermittent fillet weld. You can specify the length and pitch of the intermittent fillet in the **Length** and **Pitch** edit boxes, respectively. Figure 16-11 shows an intermittent fillet weld in a butt-joint. In this case, the length value is 10 mm and the pitch value is 20 mm.

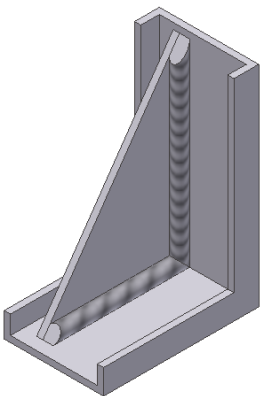


Figure 16-10 *Fillet weld with convex contour*

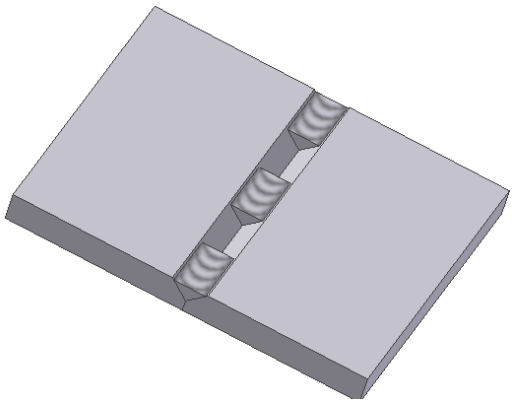


Figure 16-11 *Intermittent fillet weld in a butt-joint*

Extents Area

The options in this area are used to specify the extents of a fillet weld. The **All** option is used to create the fillet weld throughout the selected faces. You can also specify the start and end offsets of the fillet weld. The **From-To** option is used to define the extent of the intermittent weld. You can create work planes to specify the “from” and “to” faces. The **Start-Length** option is used to create a weld that starts from a selected face and ends at a specified length.

Start Offset/End Offset

The **Start Offset** and **End Offset** edit boxes will be available only when you select the **All** option from the drop-down list in the **Extents** area. These edit boxes are used to specify the distance value of a weld from the start/end edge of the components on which you want to create a fillet weld, as shown in Figure 16-12. If you select the **From-To** option from the drop-down list in the **Extents** area, the **From** and **To** buttons will be available. You can select faces or planes for the start and end of the fillet. If you select the **Start-Length** option from the drop-down list, the **Start Offset** and **Length** edit boxes will be available. You can specify the start offset and the length of the fillet weld in these edit boxes.

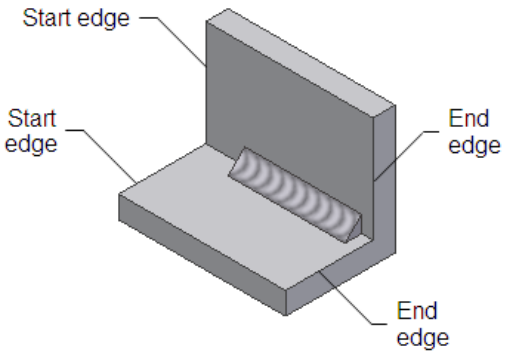


Figure 16-12 *Fillet weld offset from the start and end edges*

Create Welding Symbol

When you select this check box, the **Fillet Weld** dialog box expands and provides option to add weld symbols.

CREATING COSMETIC WELDS

Ribbon: Weld > Weld > Cosmetic



To create cosmetic welds between components, choose the **Cosmetic** tool from the **Weld** panel of the **Weld** tab; the **Cosmetic Weld** dialog box will be displayed, as shown in Figure 16-13. You can also invoke this tool from the Marking menu.

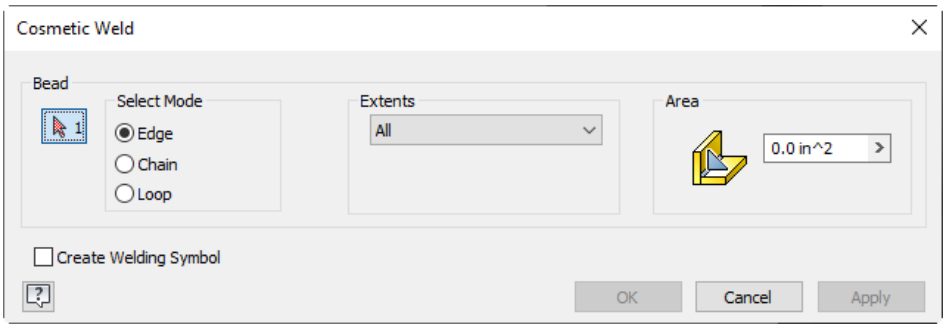


Figure 16-13 *The **Cosmetic Weld** dialog box*

Bead Area

When you invoke the **Cosmetic Weld** dialog box, the **1** button is chosen and you are prompted to select an edge or loop for the weld. You can set the selection mode using the options in the **Select Mode** area on the right of the **1** button. The selected edge turns blue.

Extents

The options in this area are used to create a cosmetic weld up to a specified extent. By default, the **All** option is selected in this area. As a result, the weld is created across the whole length of the chosen edge. The extent of the weld can also be specified by selecting the **From-To** option from the drop-down list in this area. You can create work planes to specify the "from" and "to" faces.

Area

The edit box in this area is used to specify the cross-sectional area of the fillet weld. Note that even when you increase this value, there will be no change in the display of the fillet weld in the model. This is because this value is used only for calculating the physical properties of the model after welding.

Figure 16-14 shows two components welded together using the cosmetic weld.

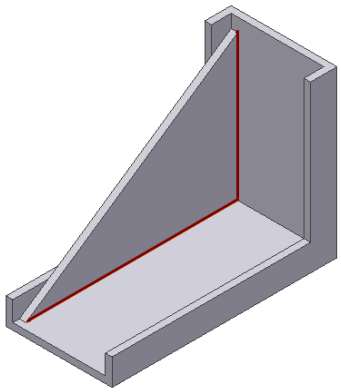


Figure 16-14 *Two components welded using the cosmetic weld*

CREATING GROOVE WELDS

Ribbon: Weld > Weld > Groove



As mentioned earlier, groove welds are created between the components that are either assembled at some offset or have some grooves between them. To create groove weld, invoke the Welding environment and choose the **Groove** tool from the **Weld** panel of the **Weld** tab; the **Groove Weld** dialog box will be displayed, as shown in Figure 16-15. The options in this dialog box are discussed next.

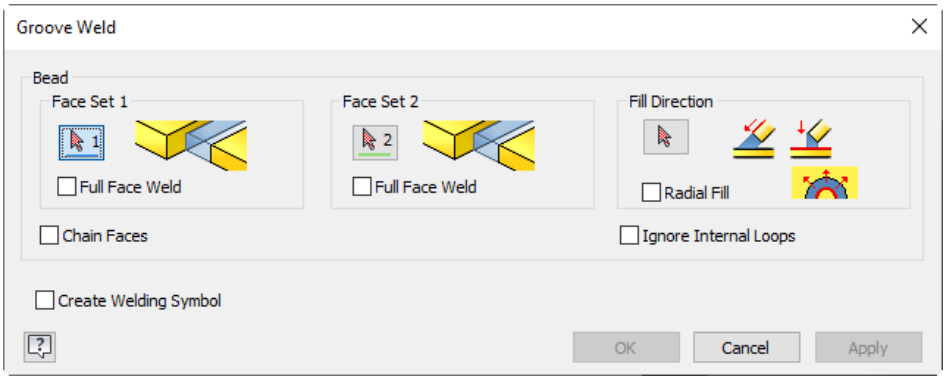


Figure 16-15 *The* ***Groove Weld*** *dialog box*

Bead Area

The options in this area are used to specify the faces to be groove welded. You can also specify the direction of the groove weld. These options are discussed next.

Face Set 1 Area

The options in this area are used to select the first face for applying the groove weld. When you invoke the **Groove Weld** dialog box, the **1** button in the **Face Set 1** area will be chosen and you will be prompted to the select the face to be welded. The face selected as face set 1 turns blue. You can select the **Full Face Weld** check box to add the weld bead to the entire face.

Face Set 2 Area

The options in this area are used to select the second face for applying the groove weld. The face selected as face set 2 is highlighted. You can select the **Full Face Weld** check box to add the weld bead to the entire face.

Fill Direction Area

If you are not creating a full face weld, you need to specify the direction of a groove weld. You can specify the direction by using a linear edge, cylindrical face, or planar face, or by using two vertices.

Figure 16-16 shows the quarter section view of a full face groove weld. Note that in this case, the cylinder is assembled at an offset from the bracket. Select the **Radial Fill** check box to project the groove weld for the cylindrical or hole feature, as no fill direction is required for welding these features.

Ignore Internal Loops

Select this check box to create a groove weld by ignoring the internal loop. Figure 16-17 shows the quarter section view of a groove weld created by ignoring the internal loop which is the hole in the bracket.

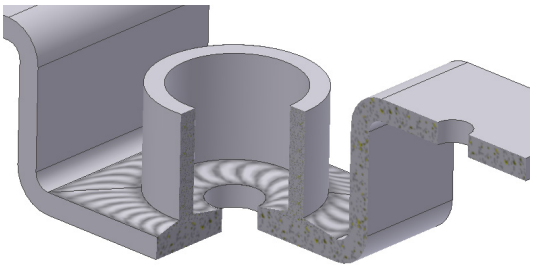


Figure 16-16 Full face groove weld

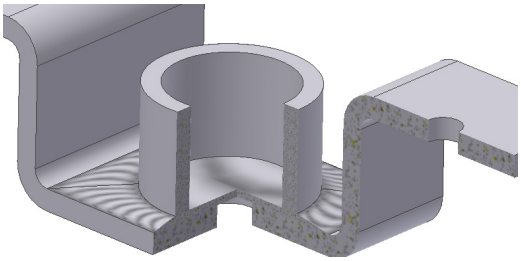


Figure 16-17 Groove weld created by ignoring the internal loop

CREATING SYMBOLS

Ribbon: Weld > Weld > Symbol



You can also add welding symbols to a weld bead by using the **Symbol** tool. To do so, choose the **Symbol** tool from the **Weld** panel of the **Weld** tab; the **Welding Symbol** dialog box will be displayed, refer to Figure 16-18. Now, select a weld bead from the model; a weld symbol will be displayed attached to the selected bead. Next, specify the weld properties in the respective edit boxes available in the **Welding Symbol** dialog box. After specifying the properties, choose the **OK** button; the weld symbol along with the specified properties will be displayed on the weld bead.

Note that before invoking this tool you need to make sure that there are unconsumed weld beads in the assembly otherwise the **Autodesk Inventor Professional 2026** dialog box will be displayed, informing that there are no unconsumed weld beads in the assembly.

Figure 16-19 shows a dialog box with various symbols of a Fillet weld.

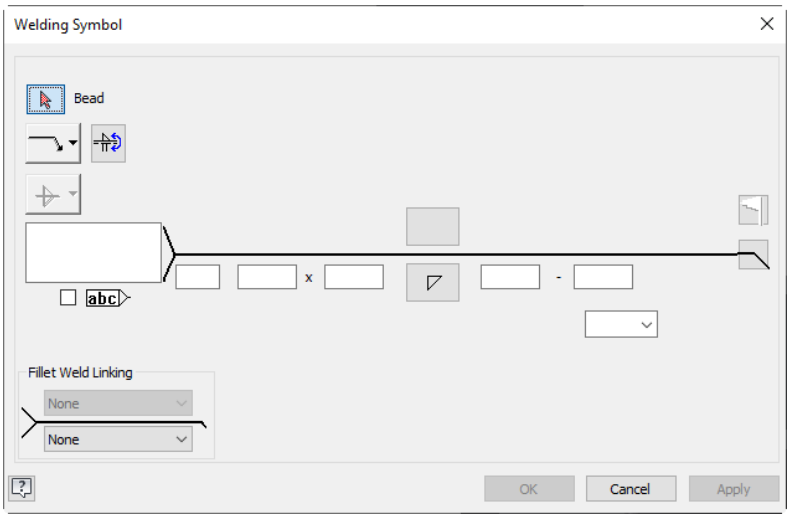


*Figure 16-18 The **Welding Symbol** dialog box*

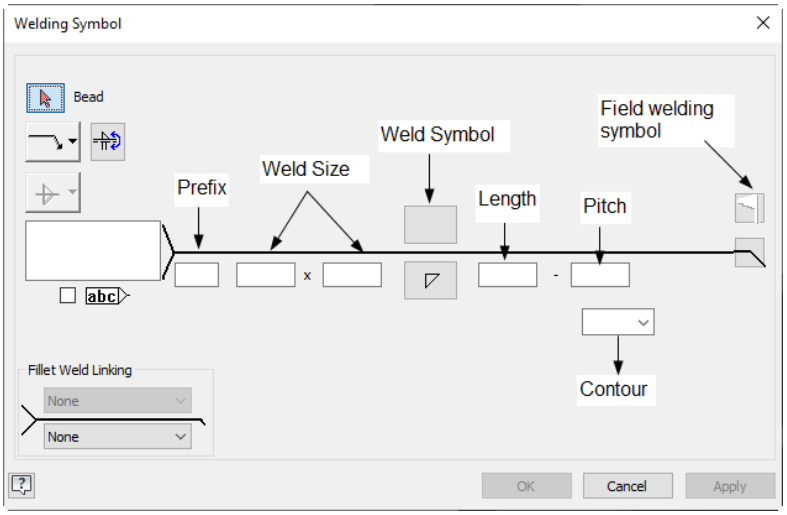


Figure 16-19 *The* ***Welding Symbol*** *dialog box with symbols*

Figure 16-20 shows the Welding symbol applied to a fillet weld. The Specification Process indicates the type of welding process to be used. The Welding symbol indicates a Fillet weld of 10 x 10 mm with convex contour symbol as weld shape symbol. The flag symbol indicates a field or site weld to the selected reference line, refer to Figure 16-20. The All Around Symbol specifies whether to use an all around symbol on the selected reference line. This sign indicates the welding to be done through all the part or not.

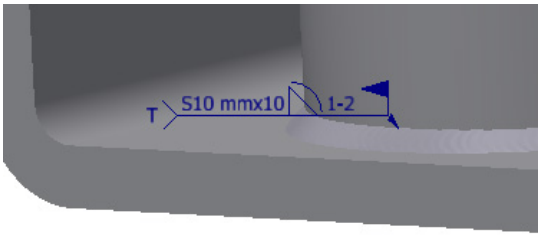


Figure 16-20 *Welding symbols applied on a weld*

GENERATING REPORT

Ribbon: Weld > Weld > Bead Report

In Autodesk Inventor, you can generate bead report. This report will be in form of Excel spreadsheet and provides information about different properties like weld ID, type, length, mass, area, and volume. To generate a bead report, choose the **Bead Report** tool from the **Weld** panel of the **Weld** tab; the **Weld Bead Report** dialog box will be displayed. Now, choose the **Next** button from this dialog box; the **Report Location** dialog box will be displayed. In this dialog box, specify the file name and file location. Next, choose the **Save** button; bead report is generated in Excel format at the specified location.

TUTORIALS

Tutorial 1

In this tutorial, you will create the welded butt-joint shown in Figure 16-21. To create this weldment, you will use the **Weldment (ANSI - mm).iam** template and the top-down approach for assembling plates. Next, you will prepare plates for welding in the Weldment environment by chamfering their edges. The plate to be used for creating the butt-joint is 30 mm long and 50 mm wide. The thickness of the plate is 5 mm and the chamfer is a 3 mm equidistant chamfer. **(Expected time: 30 min)**

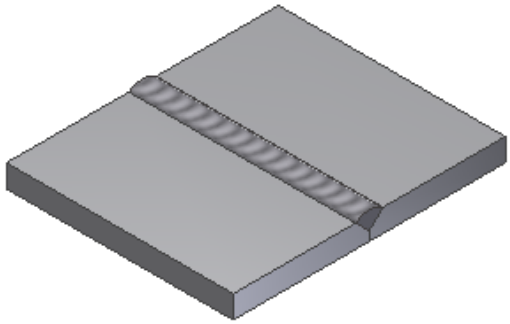


Figure 16-21 *Welded butt-joint for Tutorial 1*

The following steps are required to complete this tutorial:

a. Open a new **Weldment (ANSI - mm).iam** template and then create a plate using the top-down assembly approach.
b. Exit the part modeling environment and insert another instance of the plate into the Weldment environment.
c. Prepare plates for welding by creating chamfers.
d. Assemble two plates by using assembly constraints.
e. Invoke the welding options and create the butt-joint using the fillet weld.

Opening a New Weldment File and Creating a Plate

As mentioned in the tutorial description, you need to use the **Weldment (ANSI - mm).iam** file for creating the butt-joint. Therefore, you need to select this file from the **Create New File** dialog box.

1. Start Autodesk Inventor and invoke the **Create New File** dialog box. Next, double-click on the **Weldment (ANSI - mm).iam** template from the **Metric** tab to invoke the Weldment environment.

2. Choose the **Create** tool from the **Assemble** tab; the **Create In-Place Component** dialog box is displayed. Enter **Plate** in the **New Component Name** edit box.

3. Choose the **Browse Templates** button; the **Open Template** dialog box is displayed. Select **Standard(mm).ipt** from the **Metric** tab. Choose the **OK** button from the **Open Template** dialog box.

4. Next, choose the **OK** button from the **Create In-Place Component** dialog box and then create a plate of dimensions 50 x 30 x 5 mm. Save the file and then choose the **Return** tool from the **Return** panel to return to the Assembly environment. The Weldment assembly after creating the plate is shown in Figure 16-22.

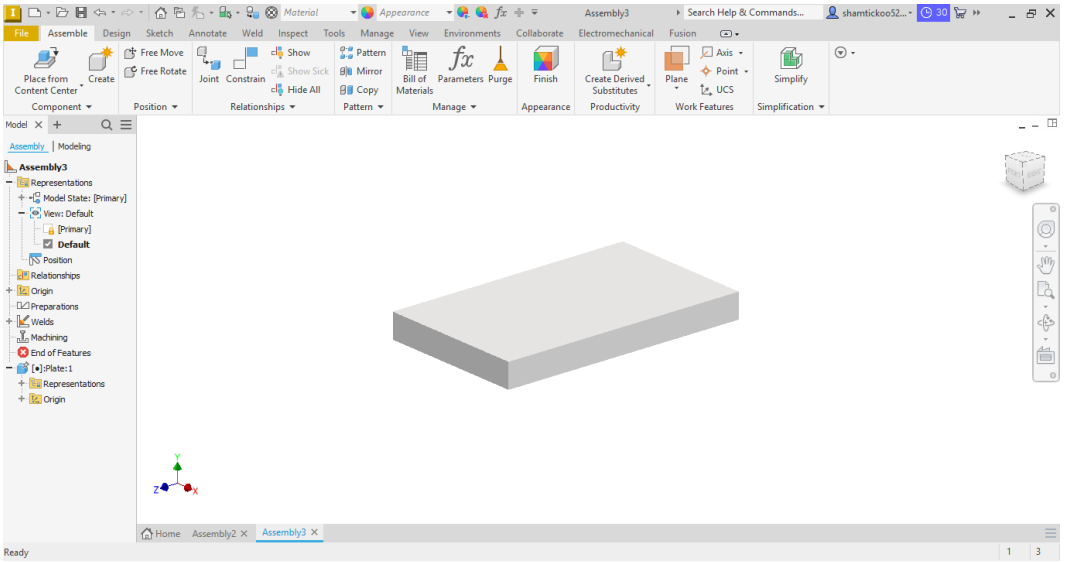


Figure 16-22 *Weldment after creating the plate*

5. Choose the **Save** tool from the **Quick Access Toolbar**; the **Save As** dialog box is displayed.

6. Specify **Tutorial 1** in the **File name** edit box of this dialog box and then choose the **Save** button; the **Save** message box is displayed.

7. Choose **Yes to All** and then **OK** from the **Save** message box to save both part and assembly files.

Placing another Instance of the Plate

Next, you need to place another instance of the Plate in the Weldment (assembly) environment. Note that you will not assemble the two plates at this stage. First, you need to prepare them for welding by chamfering their edges.

1. Choose the **Place** tool from the **Component** panel of the **Assemble** tab to invoke the **Place Component** dialog box.

2. Double-click on *Plate.ipt* file; the file is selected and the **Place Component** dialog box is closed. Also, you are prompted to place the component.

3. Specify a point on the screen at a location where the second instance will not interfere with the previous instance, refer to Figure 16-23. Right-click and then choose **OK** from the shortcut menu displayed.

4. Choose **Zoom All** from the **Navigation Bar** to modify the drawing display area.

Preparing the Two Plates for Welding

Next, you need to prepare the two plates for welding by chamfering them. To chamfer the edges, you need to activate the **Weld** tab.

1. Choose the **Preparation** tool from the **Weld** tab or double-click on **Preparations** in the **Browser Bar** to activate various preparation and machining tools in the **Ribbon**. You can also invoke this tool from the Marking menu.

2. Choose the **Chamfer** tool from the **Preparation and Machining** panel of the **Weld** tab to invoke the **Chamfer** dialog box.

3. Enter **3** in the **Distance** edit box and then select one of the edges (of 50 mm length) of one of the plates. Choose **OK** to create the chamfer and exit the dialog box.

4. Similarly, chamfer the edge on the top face of the other plate.

   You will notice that **Chamfer 1** and **Chamfer 2** features are added under **Preparations** in the **Browser Bar**. This is because you have created both the chamfers in the two plates as two different features.

5. Choose the **Return** tool from the **Return** panel of the **Weld** tab to finish the preparation (chamfering) of the component for weldment. The weldment assembly after chamfering the edges of the two plates is shown in Figure 16-23.

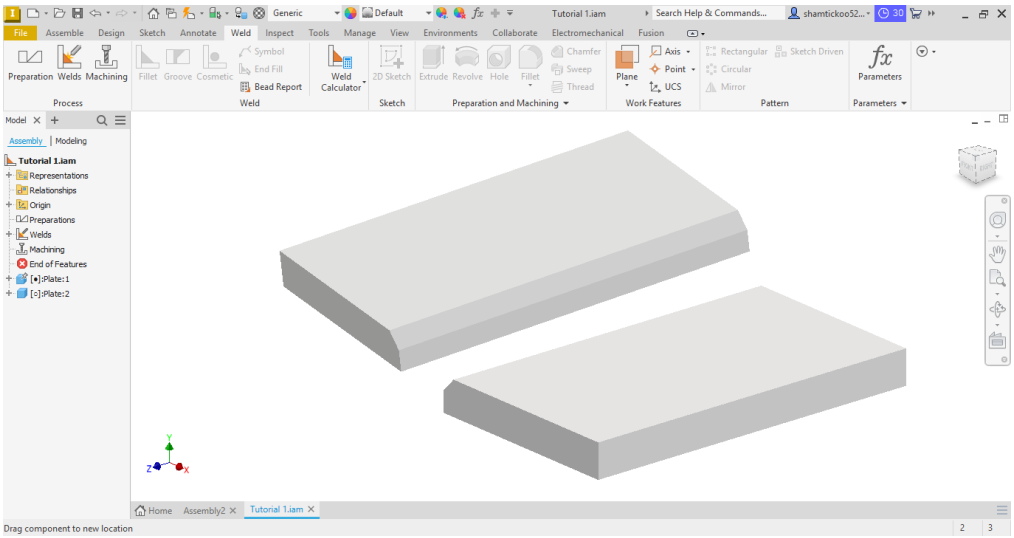


Figure 16-23 *Weldment after chamfering the two plates*

Assembling the Two Plates

1. Apply multiple instances of the **Mate** constraint to the two instances of the plate and assemble them. The assembly of the two plates is shown in Figure 16-24.

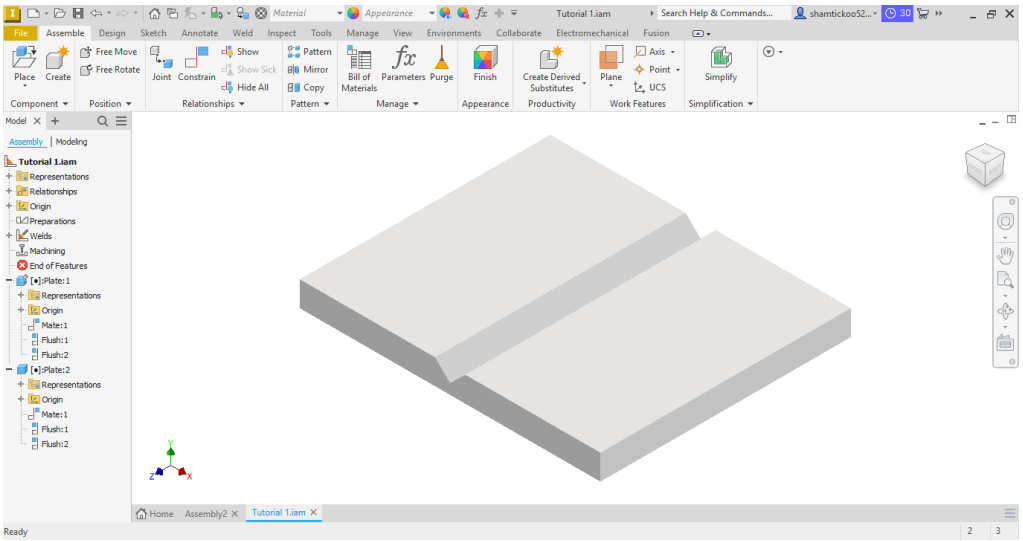


Figure 16-24 *Weldment after assembling the two plates*

Creating the Fillet Weld

Next, you need to create the fillet weld. To weld components, you need to activate the welding tools.

1. Choose the **Welds** tool from the **Process** panel of the **Weld** tab or double-click on **Welds** in the **Browser Bar**; various welding tools are activated. You can also invoke this tool from the Marking menu.

   Remember that when you double-click on **Preparations** in the **Browser Bar**, the welding tools in this panel do not get activated. But, when you double-click on **Welds** in the **Browser Bar**, these tools get activated.

2. Choose the **Fillet** tool from the **Weld** panel of the **Weld** tab; the **Fillet Weld** dialog box is displayed and you are prompted to select the face to weld. You can also invoke this tool from the Marking menu.

3. Select the chamfered face on one of the plates; the selected face turns blue.

4. Choose the **2** button from the **Bead** area of the **Fillet Weld** dialog box; you are prompted again to select the face to weld.

5. Rotate the model using the **Rotate** tool and select the chamfered face on the other plate; second selected face gets highlighted.

   If you apply weld at this stage, a warning box will appear informing that the face selected for the leg of the bead will be recomputed which will be smaller than the specified leg size. Therefore, you need to reduce the leg size of the bead.

6. Enter **4** in the **Leg 1** edit box below the **1** button. Make sure that the **Leg Length Measurement** radio button is selected in the dialog box.

   The welding shown in the preview will not extend beyond the V groove. As is evident from Figure 16-21, the fillet weld has a convex contour. Therefore, you need to select the **Convex** option from the **Fillet Weld** dialog box.

7. Click on the down arrow on the right of the **Flat** button in the **Contour** area; a flyout is displayed. Next, choose the **Convex** button from the Contour flyout. On doing so, the **Offset** edit box is enabled.

8. Enter **1** in the **Offset** edit box.

   With this, all welding options are defined and you can now apply the weld.

9. Choose the **Apply** button and then the **Cancel** button to exit the dialog box. Next, choose the **Return** button to finish the creation of the fillet weld. The welded plates are shown in Figure 16-25.

10. Save the assembly with the name *Butt-Joint* at the location *C:\Inventor_2026\c16\Tutorial1*.

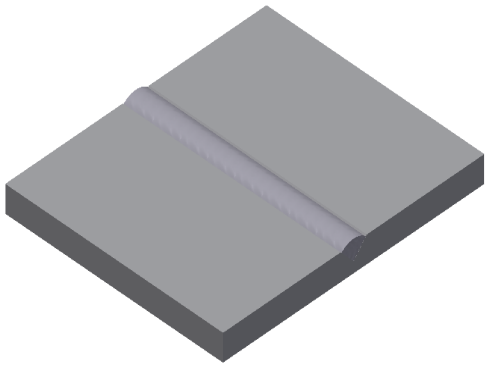


Figure 16-25 *Final weldment assembly after creating the fillet weld*

Tutorial 2

In this tutorial, you will create the Shock Assembly in the Assembly modeling environment. Also, you will switch to the Weldment environment and weld components, as shown in Figure 16-26a. The dimensions of the two components are shown in Figures 16-26b and 16-26c. **(Expected time: 45 min)**

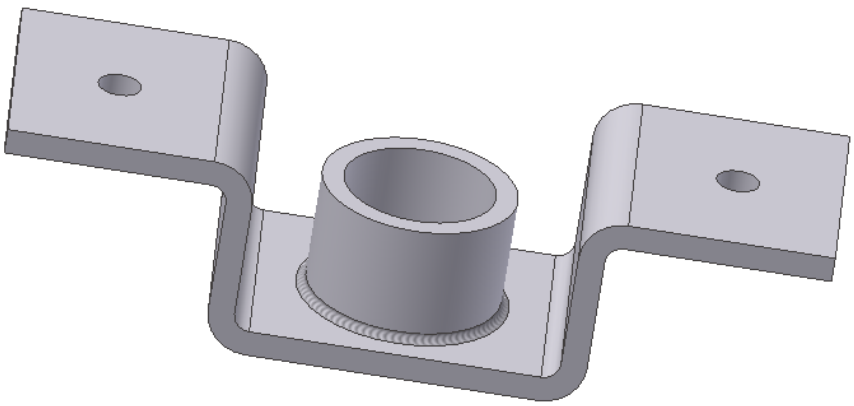


Figure 16-26a *Welded Shock Assembly*

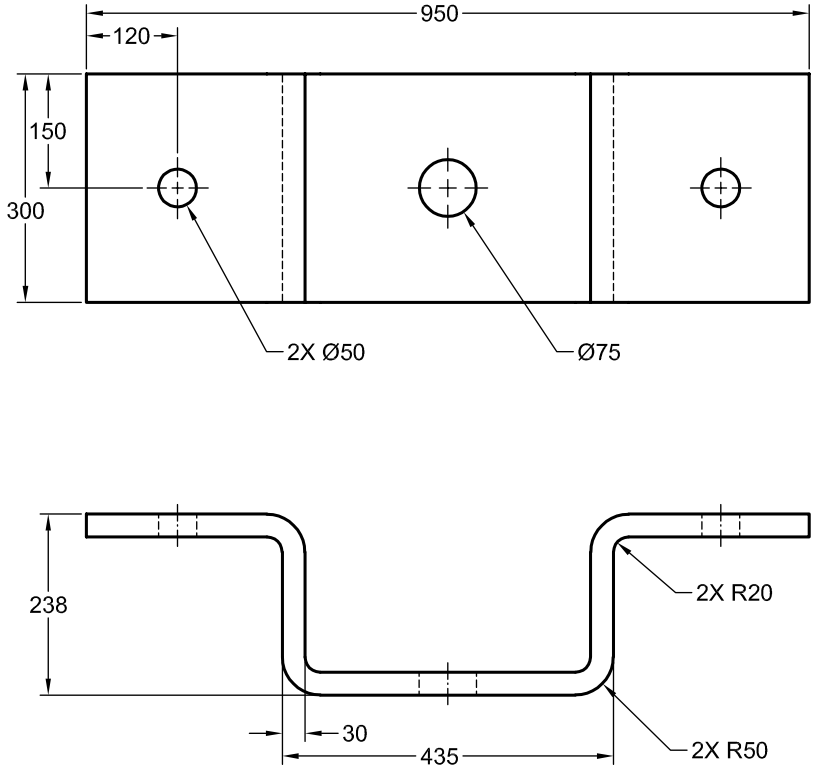


Figure 16-26b *Views and dimensions of the bracket*

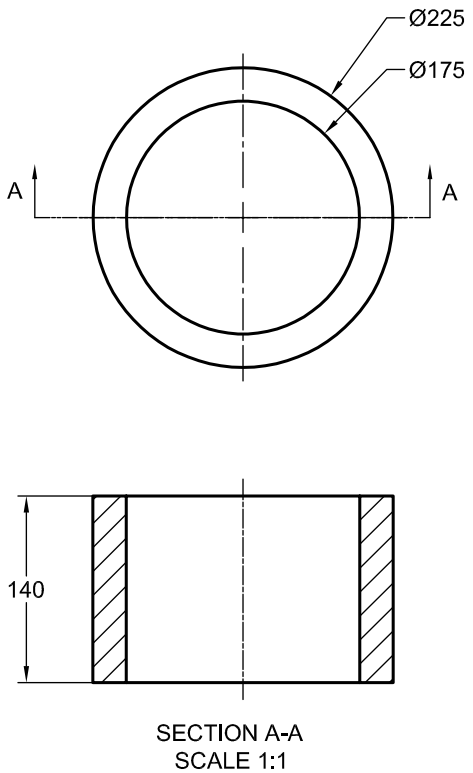


Figure 16-26c *Views and dimensions of the bracket*

The following steps are required to complete this tutorial:

a. Create the Bracket and the Cylinder as separate part files.
b. Start a new assembly file and then place the Bracket and the Cylinder in the assembly file.
c. Assemble both components by using the assembly constraints.
d. Switch to the Weldment environment.
e. Weld components using the fillet weld.

Creating Components

1. Create two components, Bracket and Cylinder, in separate part files and then save them at the location *C:\Inventor_2026\c16\Shock Assembly*.

Assembling the Components

As mentioned in the tutorial description, you need to assemble the components in the Assembly modeling environment and then switch to the Weldment environment. Therefore, you need to start a new assembly file using the **Create New File** dialog box to assemble the components.

1. Start a new assembly file by using the **Create New File** dialog box. Now, place one instance each of the Bracket and the Cylinder in the current assembly file.

2. Assemble the Cylinder with the Bracket using the assembly constraints. The assembly file after assembling the components is shown in Figure 16-27.

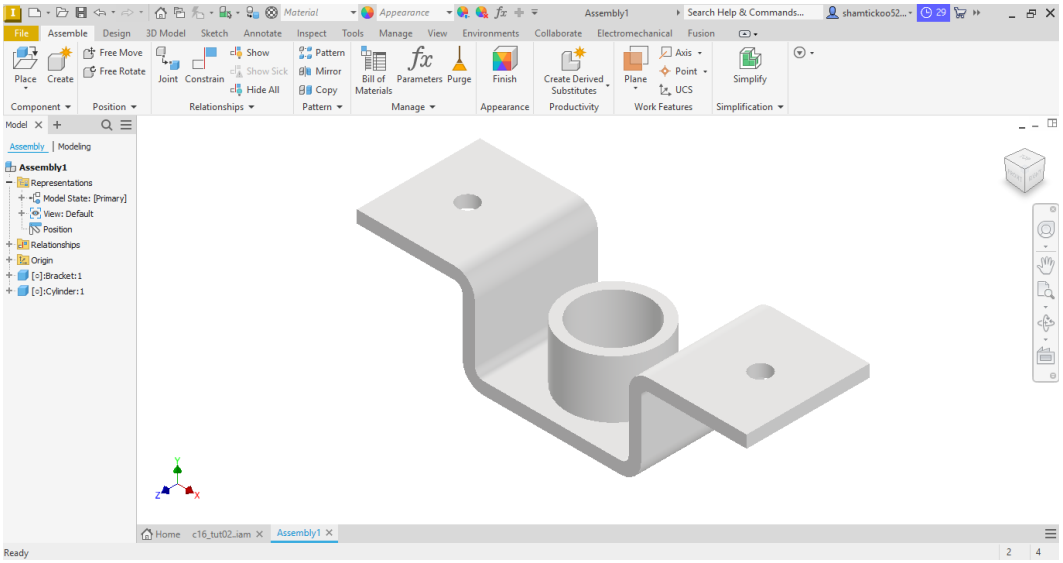


Figure 16-27 *Assembly file after assembling the Bracket and the Cylinder*

Welding the Components

Components can be welded in the Weldment assembly environment only, therefore, you need to switch from the Assembly environment to the Weldment assembly environment.

1. Choose the **Convert to Weldment** tool from the **Convert** panel of the **Environments** tab; the **Autodesk Inventor Professional** warning box is displayed, informing that once an assembly has been converted into a weldment, you cannot convert it back to an assembly.

2. Choose **Yes** in the warning box; the **Convert to Weldment** dialog box is displayed, as shown in Figure 16-28.

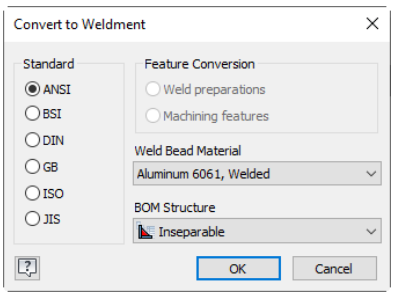


Figure 16-28 *The **Convert to Weldment** dialog box*

This dialog box is used to convert an assembly into a weldment of specified standard. You can also change the color of weldment by using this dialog box.

3. Select the **ANSI** radio button from the **Standard** area of this dialog box. Accept the remaining default options and choose **OK** to close this dialog box.

   On doing so, the Weldment environment is invoked and the **Weld** tab is activated. Since no preparation is required for welding the components, you can directly weld the components.

4. Choose the **Welds** tool from the **Process** panel of the **Weld** tab to activate the welding tools. You can also invoke this tool from the Marking menu which is displayed on right-clicking in the graphics window.

5. Choose the **Fillet** tool from the **Weld** panel of the **Weld** tab; the **Fillet Weld** dialog box is invoked and you are prompted to select the face to be welded.

6. Select the outer face of the Cylinder as the first face to be welded; the selected face turns blue.

7. Choose the **2** button from the **Fillet Weld** dialog box; you are prompted again to select the face to be welded.

8. Select the upper face of the Bracket with which the Cylinder needs to be assembled; the preview of the weld appears on the assembly.

9. Enter **10** in the **Leg 1** edit box below the **1** button.

   The fillet contour shown in Figure 16-26a is concave. Therefore, you need to specify the corresponding options from the **Contour** area.

10. Click on the down arrow on the right of the **Flat** button in the **Contour** area to display a flyout. Next, choose the **Concave** button from the flyout.

11. Enter **2** in the **Offset** edit box. Choose **Apply** and then **Cancel** to exit the dialog box. Next, choose the **Return** tool from the **Return** panel of the **Weld** tab to finish the welding.

    The final weldment assembly is shown in Figure 16-29.

12. Save the assembly with the name *Tutorial 2* at the location *C:\Inventor_2026\c16\Shock Assembly*.

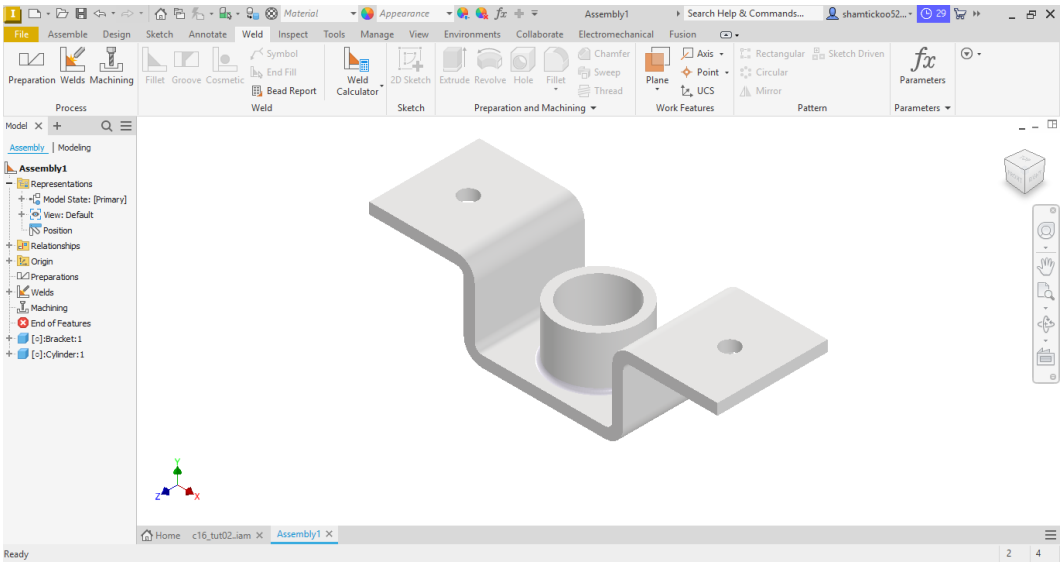


Figure 16-29 *Final weldment assembly*

Self-Evaluation Test

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

1. Which of the following types of welds does not affect the physical properties of an assembly?

   (a) **Cosmetic** (b) **Groove**
   (c) **Fillet** (c) None of these

2. Which of the following contours is not a type of weld bead?

   (a) Flat (b) Convex
   (c) Concave (d) Round

3. Which of the following welds is not considered as an actual weld?

   (a) **Fillet** (b) **Cosmetic**
   (c) **Groove** (d) None of these

4. You can convert an assembly created in the Assembly modeling environment into a Weldment assembly by choosing the _____ tool from the **Convert** panel of the **Assemble** tab.

5. You can create a _____ weld between two parts with or without including internal loops.

6. Select the _____ check box to add weld symbols to a welded assembly.

7. The Weldment environment is used to weld the components of an assembly. (T/F)

8. You can convert a weldment file into an assembly file. (T/F)

9. Groove welds are added to the parts that are placed at certain offset from each other or have grooves between them. (T/F)

10. You can create the extruded and revolved join features while preparing welds. (T/F)

Review Questions

Answer the following questions:

1. Which of the following options is used to create a fillet that extends throughout the selected edges of components?

   (a) **All** (b) **From-To**
   (c) **Start-Length** (d) None of these

2. Which of the following sub-nodes of the **Browser Bar** lists all welds of a weldment assembly?

   (a) **Preparations** (b) **Welds**
   (c) **Machining** (d) **Representations**

3. You can add a fillet symbol by using the **Fillet** tool or by using the _____ tool.

4. To create a cosmetic weld, you just need to select the edge that requires welding. (T/F)

5. The process of creating weldment assemblies is completed in four steps. (T/F)

6. You can prepare the assembled components for welding by removing material from them to accommodate weld beads. (T/F)

EXERCISES

Exercise 1

In this exercise, you will create the Base Plate and the Top Mounting as separate part files and then assemble them using the assembly constraints in the Weldment environment. The weldment assembly is shown in Figure 16-30. The dimensions of both components are shown in Figures 16-31 and 16-32. **(Expected time: 45 min)**

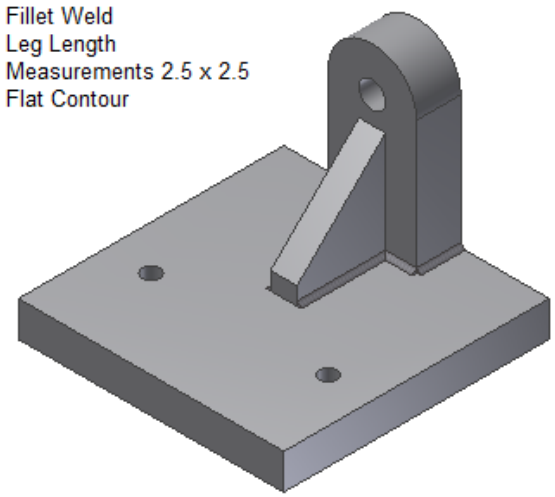


Figure 16-30 *Weldment assembly for Exercise 1*

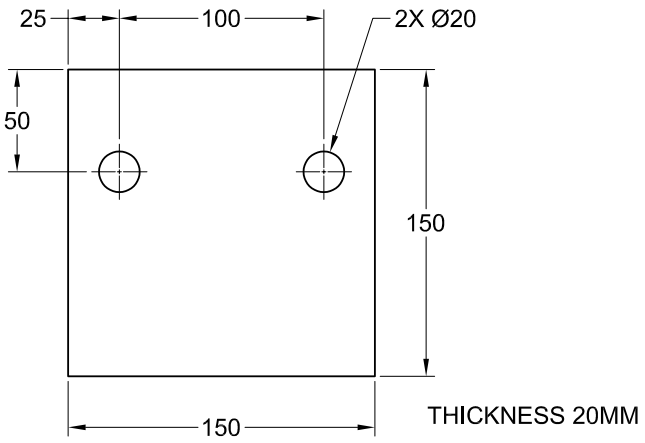


Figure 16-31 *Dimensions of the Base Plate*

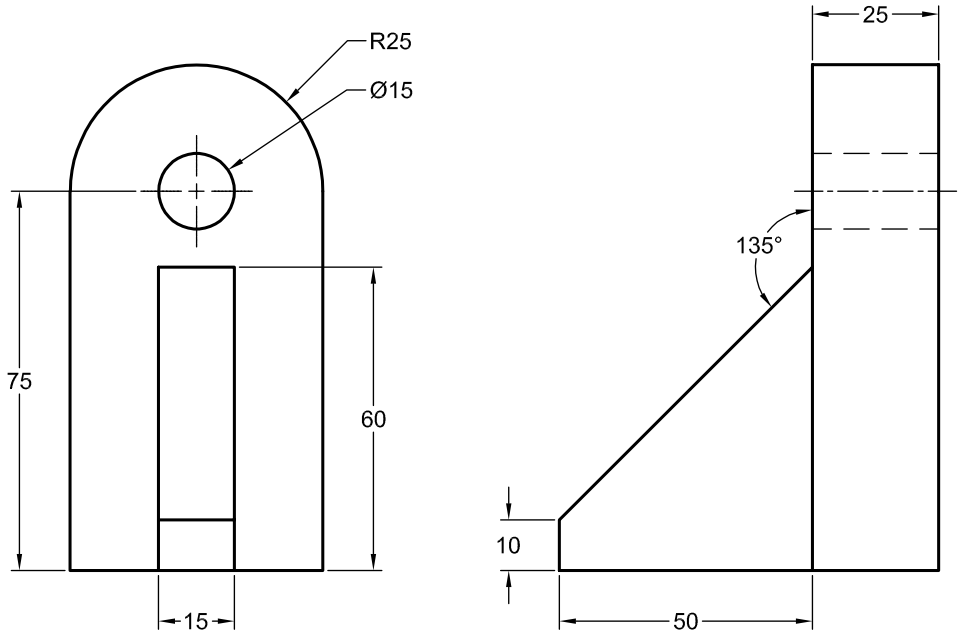


Figure 16-32 *Dimensions of the Top Mounting*

Answers to Self-Evaluation Test

1. a, **2.** d, **3. b**, **4. Convert to Weldment**, **5.** groove, **6. Create Welding Symbol**, **7. T**, **8. F**, **9. T**, **10. F**