Contents

Get Started with Tube and Pipe

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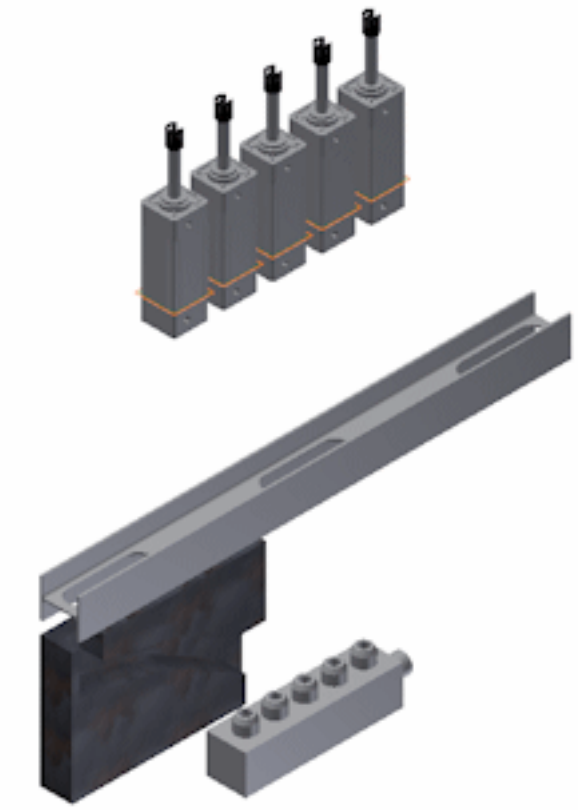
About this tutorial



Category	**Routed Systems**
Time Required	45 minutes
Tutorial Files Used	AirSystemAssy.iam

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

Objectives

- Create a tube and pipe assembly.
- Create a rigid piping and bent tubing route.
- Create a flexible hose route.
- Populate routes with library components.
- Set or edit a Tube and Pipe style.

Prerequisites

- Basic knowledge of Inventor parts and assemblies.
- Content Center libraries available in the Desktop Content location or on a server.

Navigation Tips

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

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Back up the tutorial files and set the project

1. To create a backup copy of the sample tutorial files, open the **Tutorial Files ➤ Tube and Pipe** folder. Then copy the files into your backup folder.

   TIP After you finish the exercises, you can replace the changed sample files with the original files.

2. To set the project, click  **➤ Manage ➤ Projects**.
3. In the Projects dialog box, double-click the **tutorial_files** project to make it the active project.
4. Review the setting of **Use Style Library** property. If appropriate, change the setting.

   NOTE To ensure that conduit parts are created with the correct material definition, Use Style Library must be set as Yes or Read Only.

5  Click the **Configure Content Center Libraries** button and confirm that Content Center libraries are available and selected. Then close the Configure Libraries dialog box.

   Tube and Pipe assemblies use Content Center parts. If Content Center libraries are not available, you cannot populate Tube and Pipe routes. If needed, see the Content Center or Tube and Pipe chapter in the Help for more information.

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

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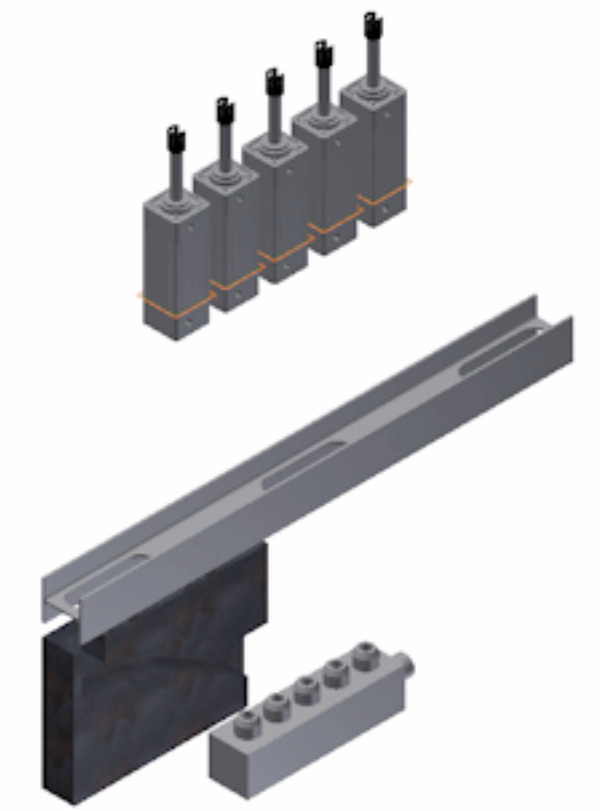
Create a tube and pipe assembly

1 Click  ➤ **Open**.

2 Open the **AirSystemAssy.iam** assembly in the **Tutorial Files ➤ Tube & Pipe ➤ Air System** folder.



3  On the ribbon, click **Environments tab ➤ Begin panel ➤ Tube and Pipe**.

4 On the Create Tube & Pipe Run dialog box, enter **AirSystem1** as the Run File Name. Accept the other defaults and click **OK**.

The individual run is activated in the browser. The Pipe Run tab displays on the ribbon.

5 On the Manage panel of the Pipe Run tab, select the **ASTM A53/A53M-ASME B16.11 - Welded Steel Pipe** from the list to set it the active style.

A tube and pipe style is a named group of settings that describe tube and pipe characteristics for automatic populating routes and runs. The selected style will be used after you finish creating the route.

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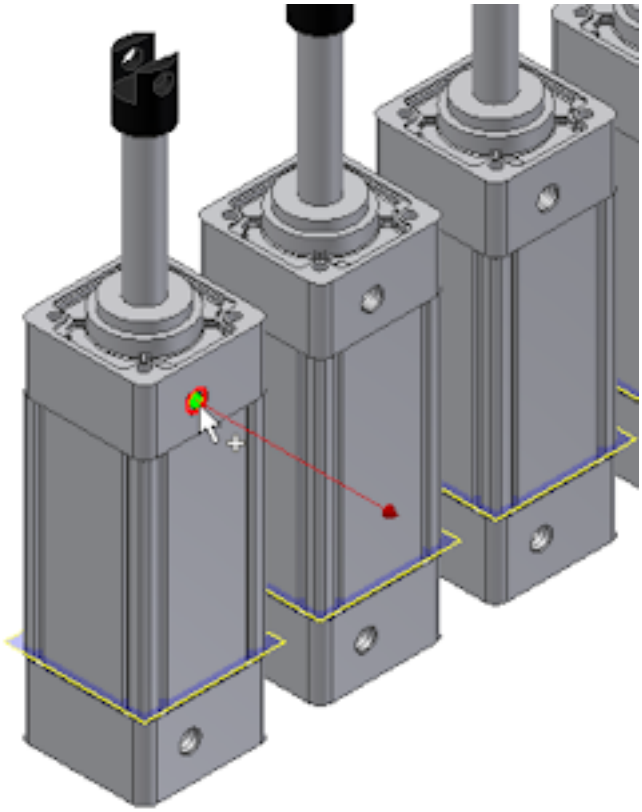
Create a route

1.  On the ribbon, click **Pipe Run tab ➤ Route panel ➤ New Route**.
2. On the Create New Route dialog box, accept the default route file name and location, and click **OK**.

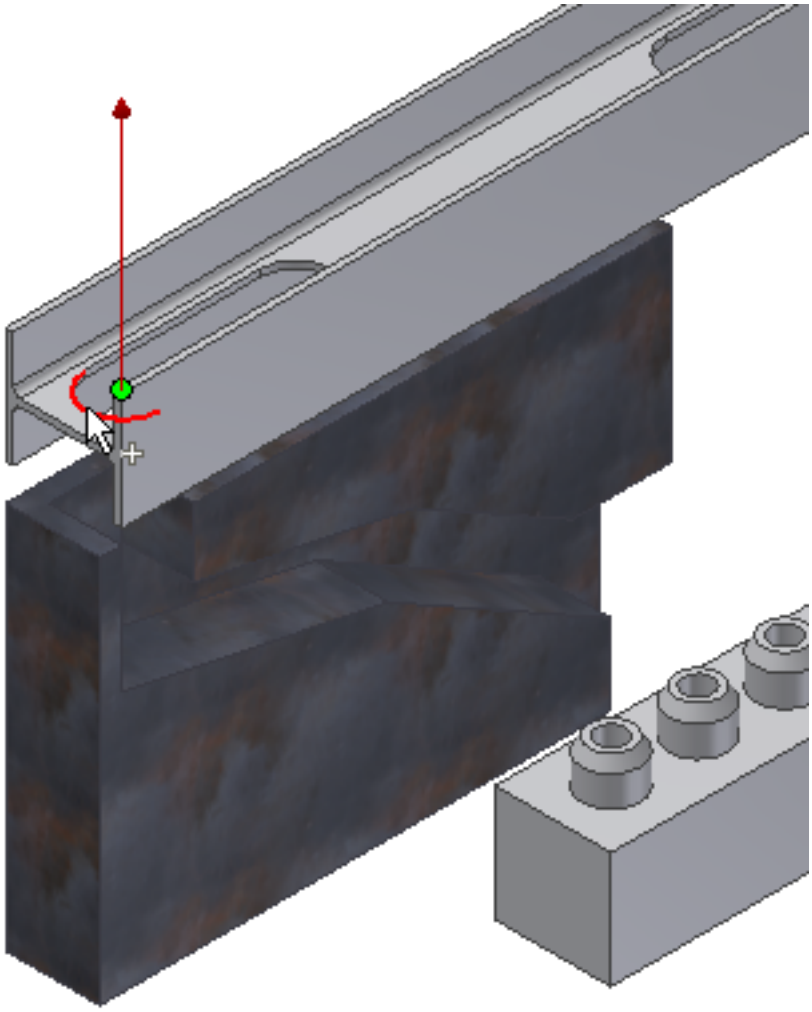
   A pipe route is added to the active run and activated. The Route tab displays on the ribbon.
3.  On the ribbon, click **Route tab ➤ Create panel ➤ Route**.
4. In the graphics window, zoom in the model. Then pause the cursor over the circular geometry of the first Air Cylinder as shown in the following image. If appropriate right-click and click Select Other Direction.



5 Click the circular geometry to select it as the start point.

   This point is associated with the component that contains the selected circular edge. A work point is added to the piping route.

6 For the second route point, select a point on the line extender or existing geometry. To route through the IBeam, pause the cursor over the left arc of the IBeam opening.



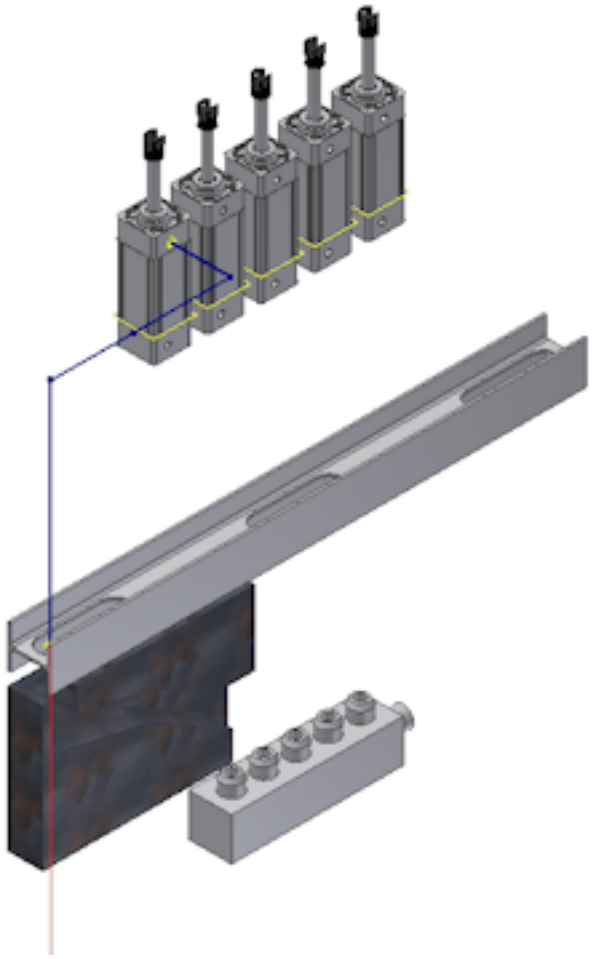
Rotate and zoom in to view the arc. Make sure that the line is pointing to the approach direction of the route.

NOTE To flip the direction of a displayed axis, press the spacebar or use the Select Other tool.

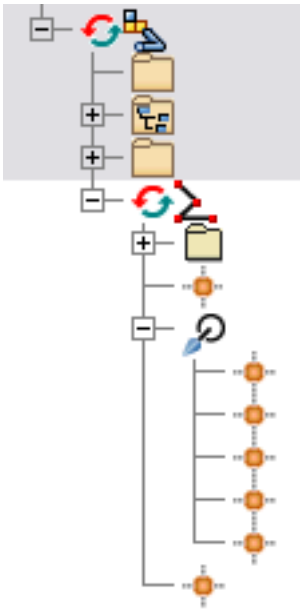
7 Click the geometry to select it as the second point.

As soon as you make your selection, the route generates to that point and the Select Other tool appears indicating that multiple solutions exist. Click the arrows to cycle through the solutions.

8 Click the green area in the Select Other tool to select the solution from the previous image.



9 In the Model browser, verify that the Autoroute browser node is added and contains five route points.



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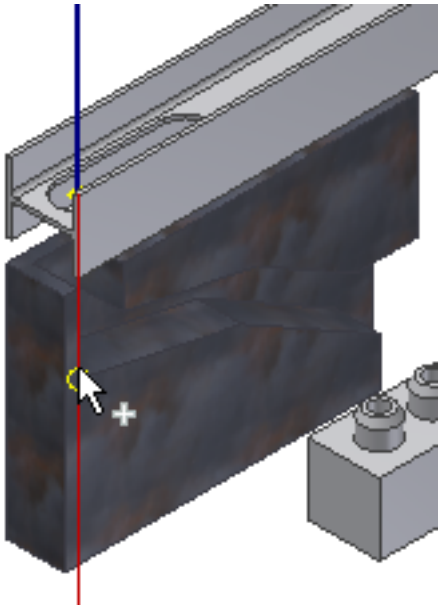
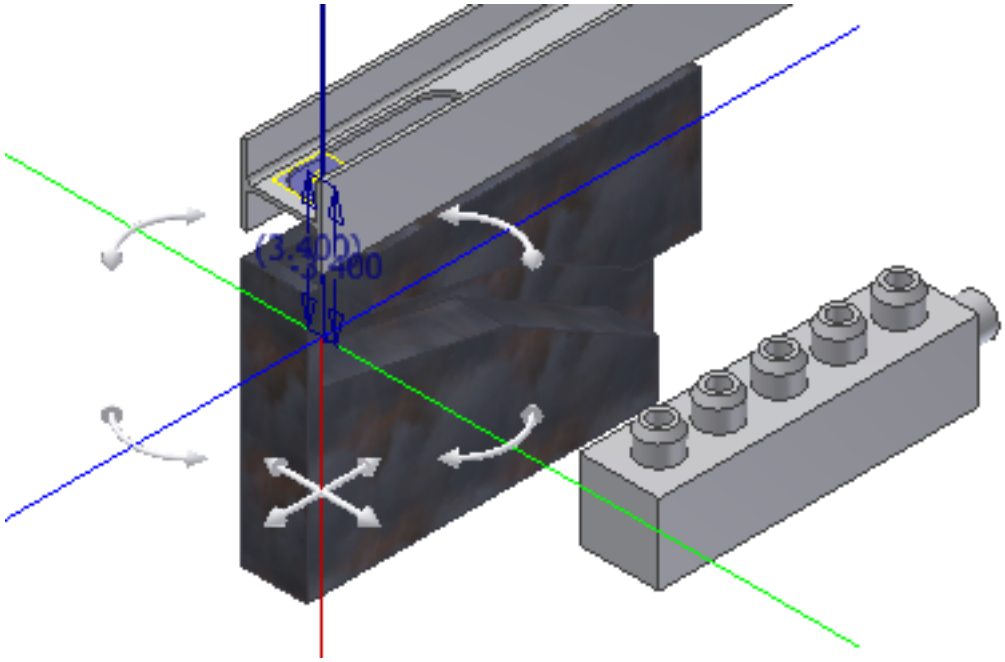
Create segments using precise dimension values

1. Right-click in the graphics window and verify that **Auto-Dimension** is enabled.
2. Pause the cursor over the red line extender, right-click, and select **Enter Distance**.



3 Enter **3.4 inch** in the Enter Distance box, and then click the green check mark.

The point is created on the screen, and the 3D Orthogonal Route tool is displayed at the new point. This time, since the new point is an arbitrary point in space, all directions are enabled and selectable, except backwards.



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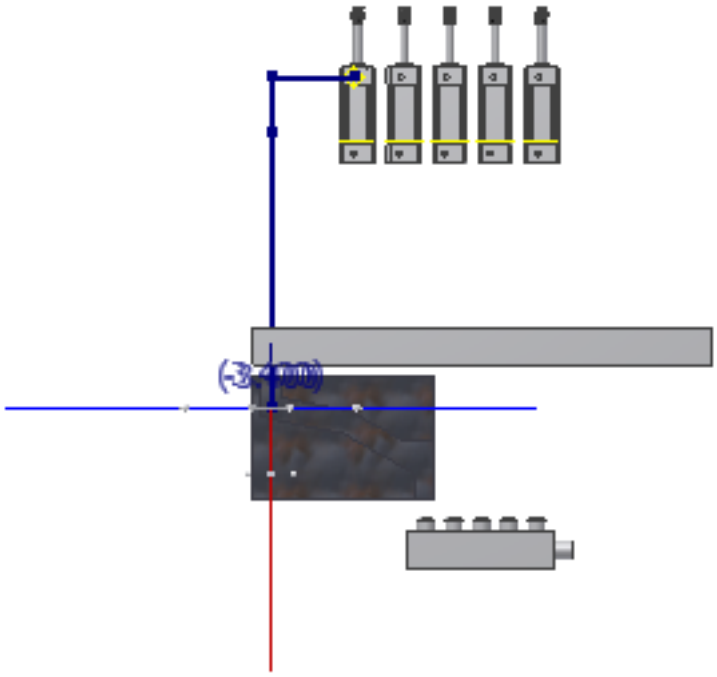
Create segments using existing geometry

1 Rotate your view to look at the front of the assembly.

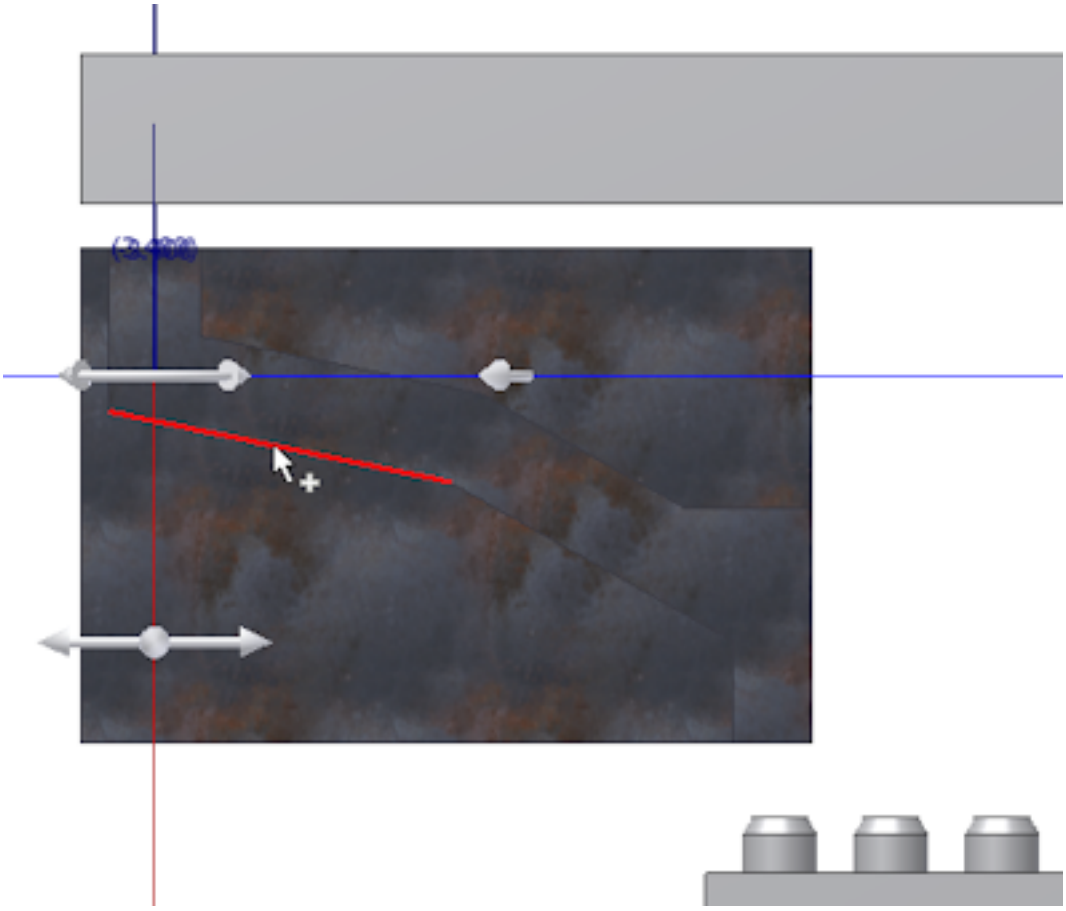
TIP Click Right on the view cube to rotate the view to the desired position.



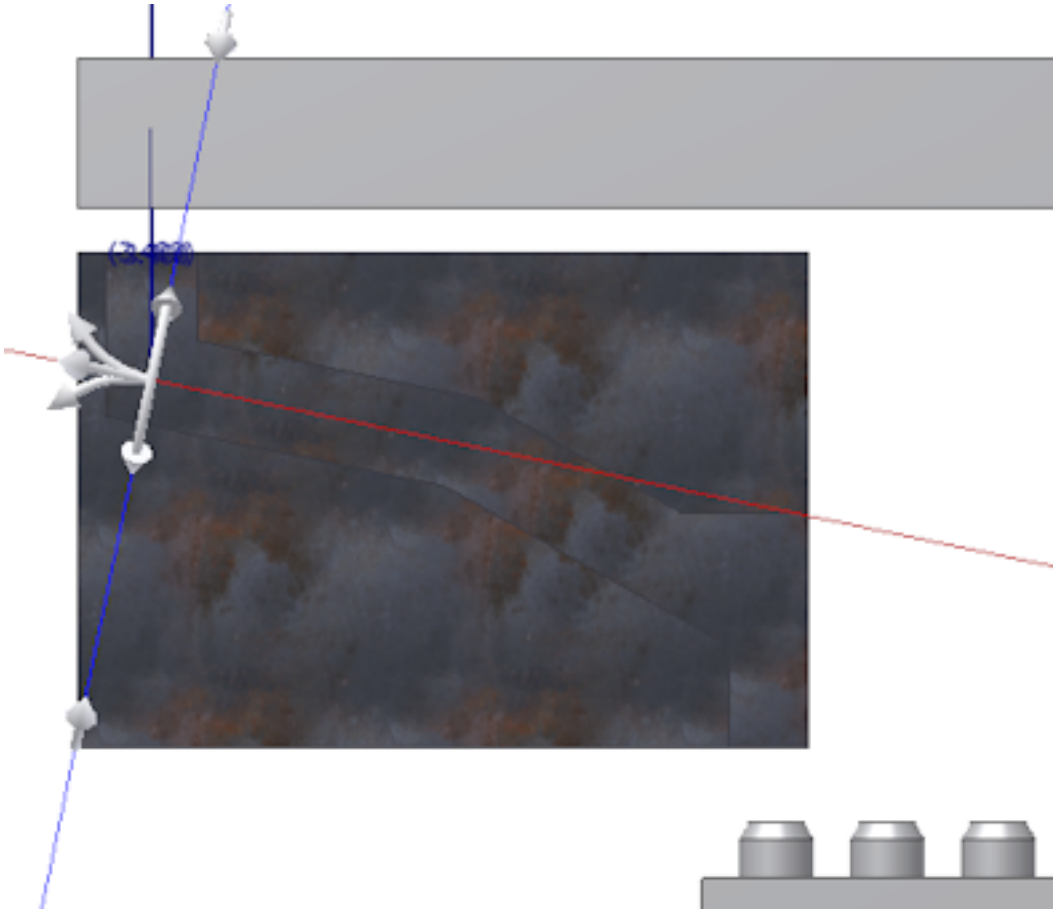
2 Pause the cursor over the red direction axis of the 3D Orthogonal Route tool, right-click, and select **Parallel With Edge**.

   The system prompts you to select a linear edge.

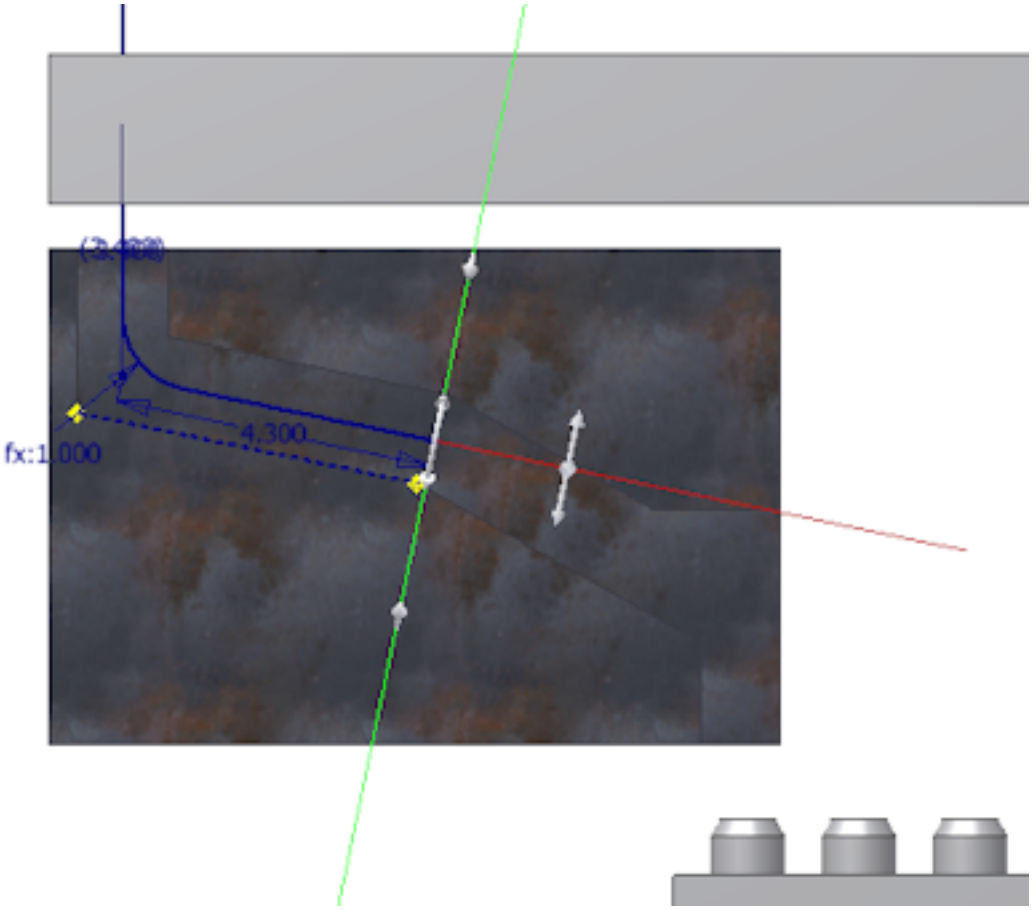
3 Select the linear edge on the path part as shown in the following image, and click to set the selection.



The red direction axis is reoriented to the direction of the selected edge. The included reference geometry is added to the Included Geometry folder in the Model browser.



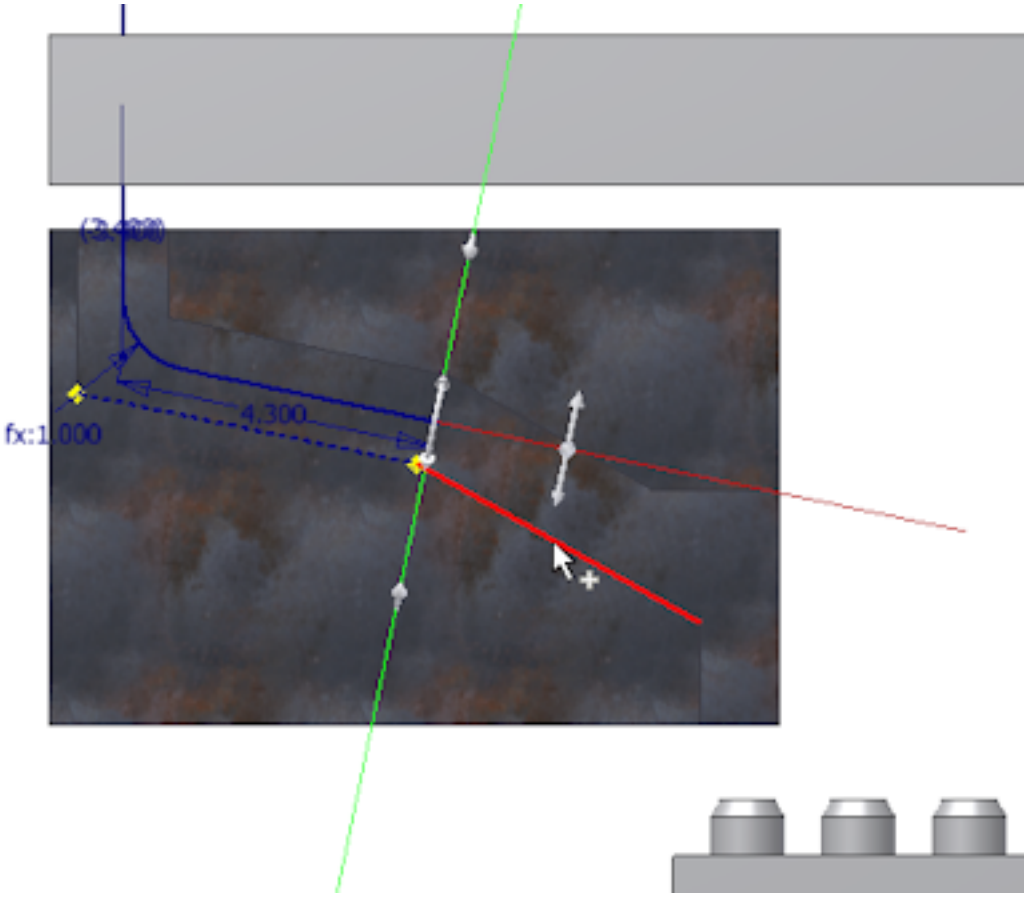
4 Pause the cursor over the red axis, right-click, and select **Enter Distance**. Then enter **4.3 inch** in the Enter Distance box, and then click the green check mark.



5 Verify that a bend is automatically added between the new segment and the preceding segment.

6 For the next point, right-click the red direction axis again and select **Parallel With Edge**.

7 Select the other linear edge on the path part as shown in the following image.

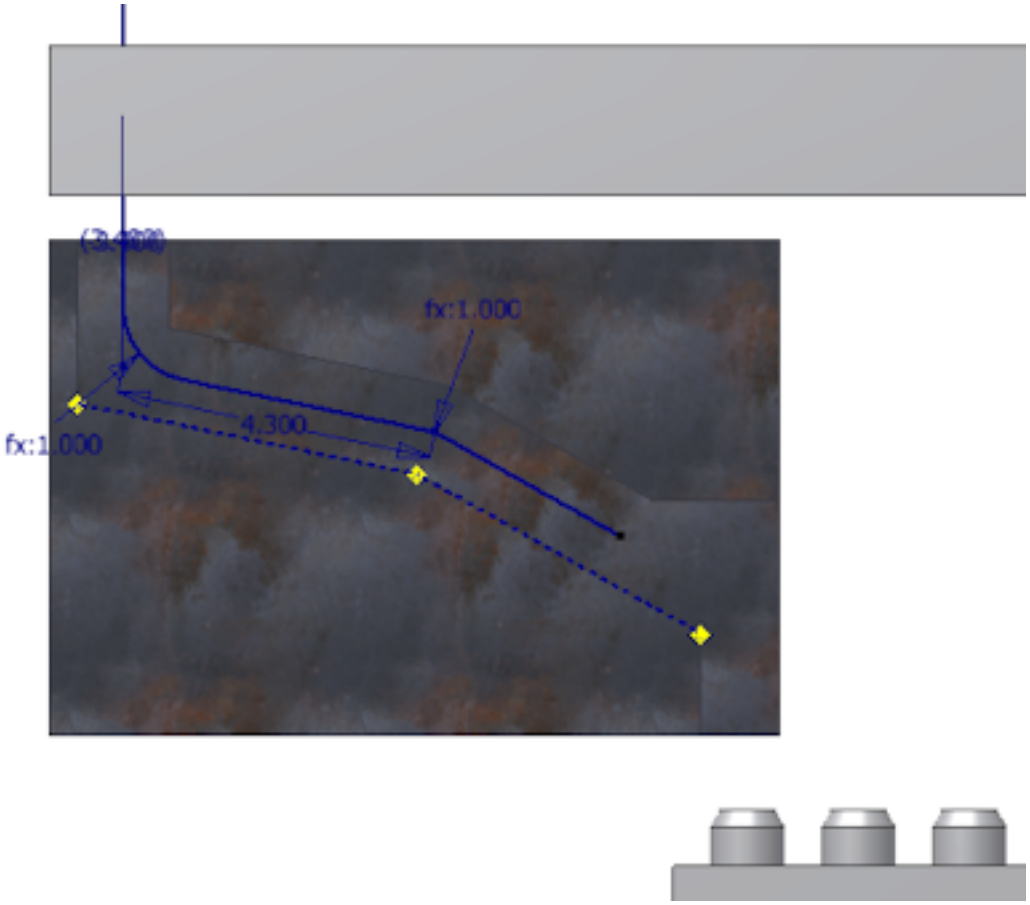


8 Before creating the next route point, right-click in the graphics window and clear the Auto-Dimension check mark.

9 Click anywhere in the forward direction on the red direction axis. The segment length is not important.

10 Right-click in the graphics window and select **Done**.

11 Verify that the new segment is not dimensioned.



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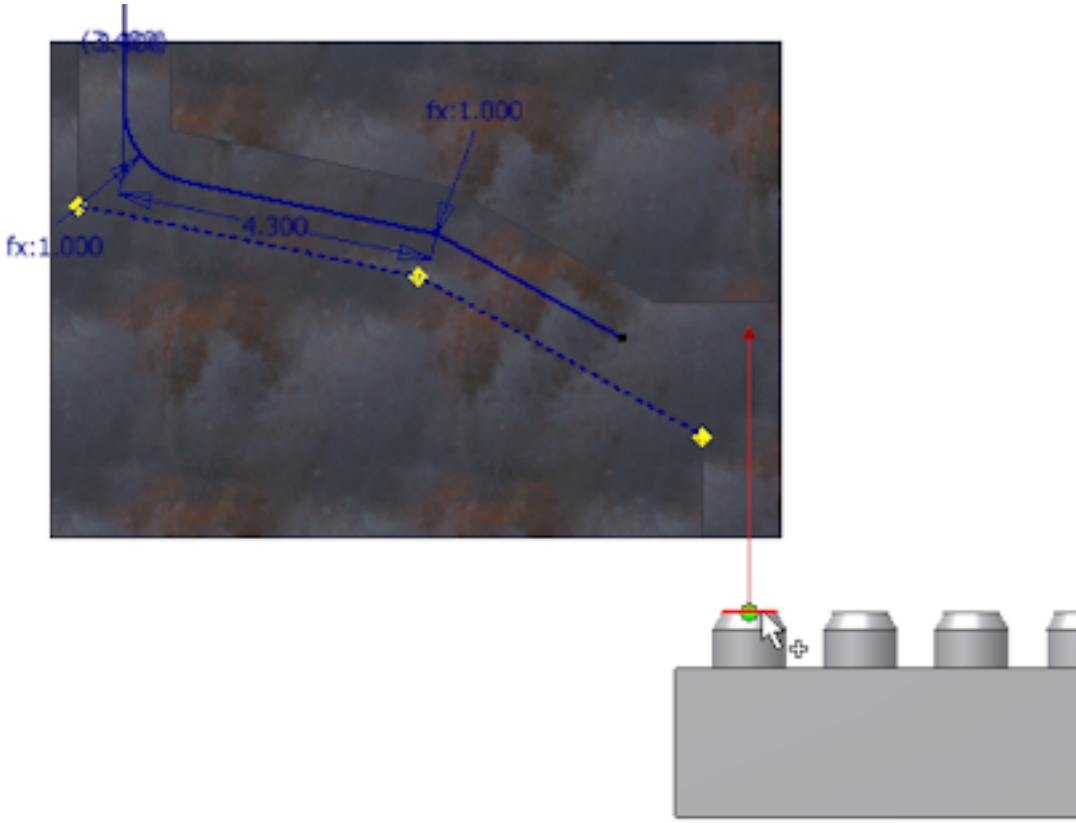
Snap route points to existing geometry

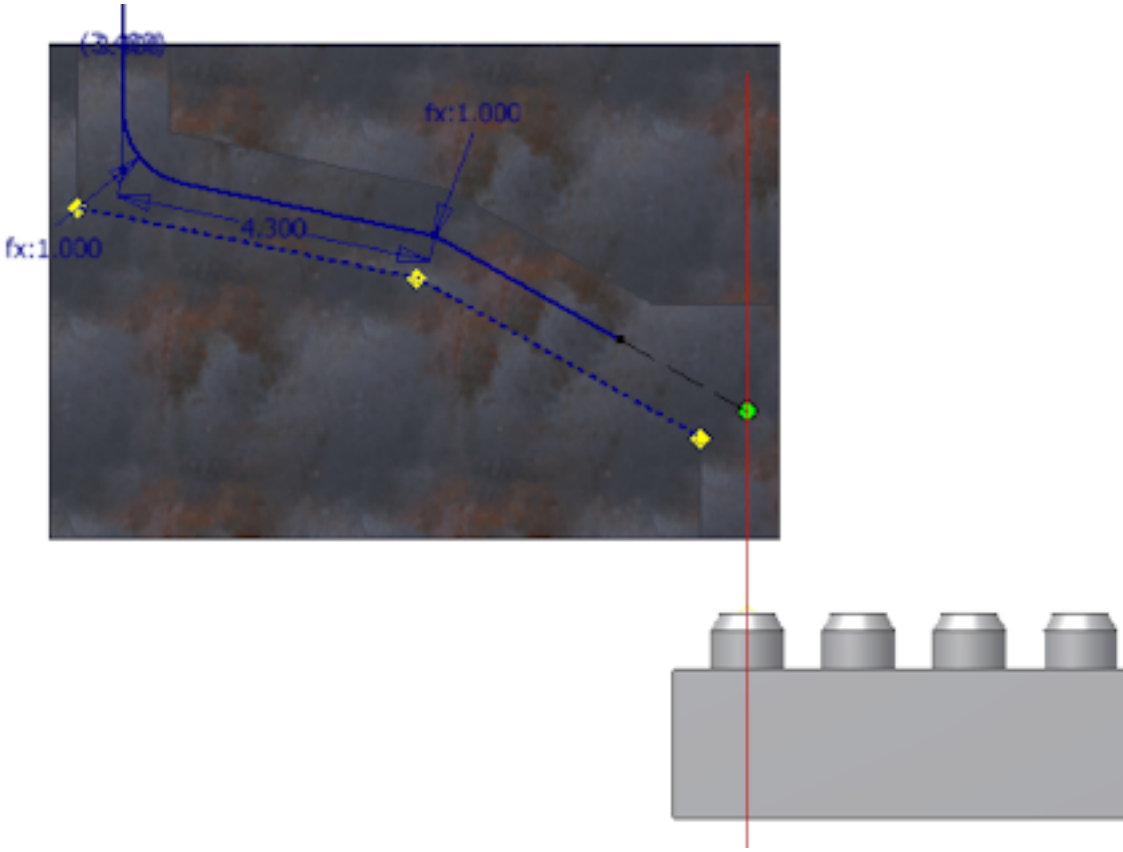
1. On the ribbon, click **Route tab** ➤ **Create panel** ➤ **Route**.
2. Click the left circular opening on the valve part.



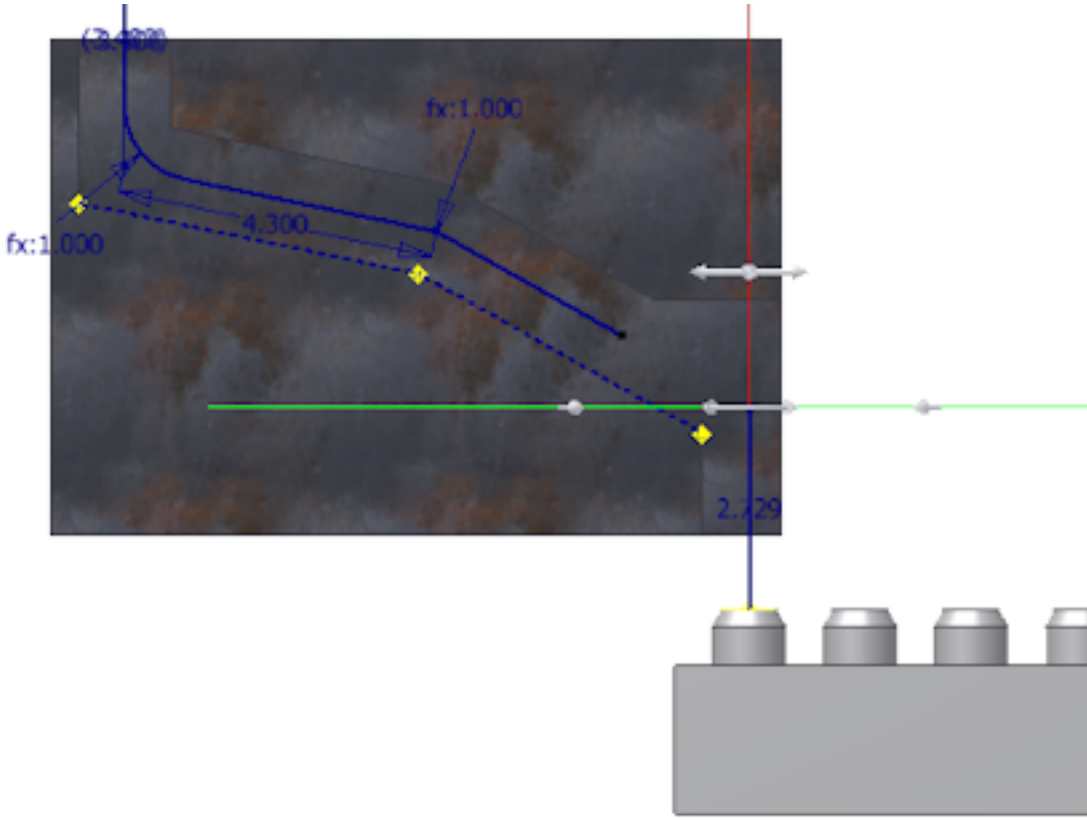
3 Right-click in the graphics window and select **Point Snap**.

4 Move the cursor over the last segment you created in the preceding exercise.

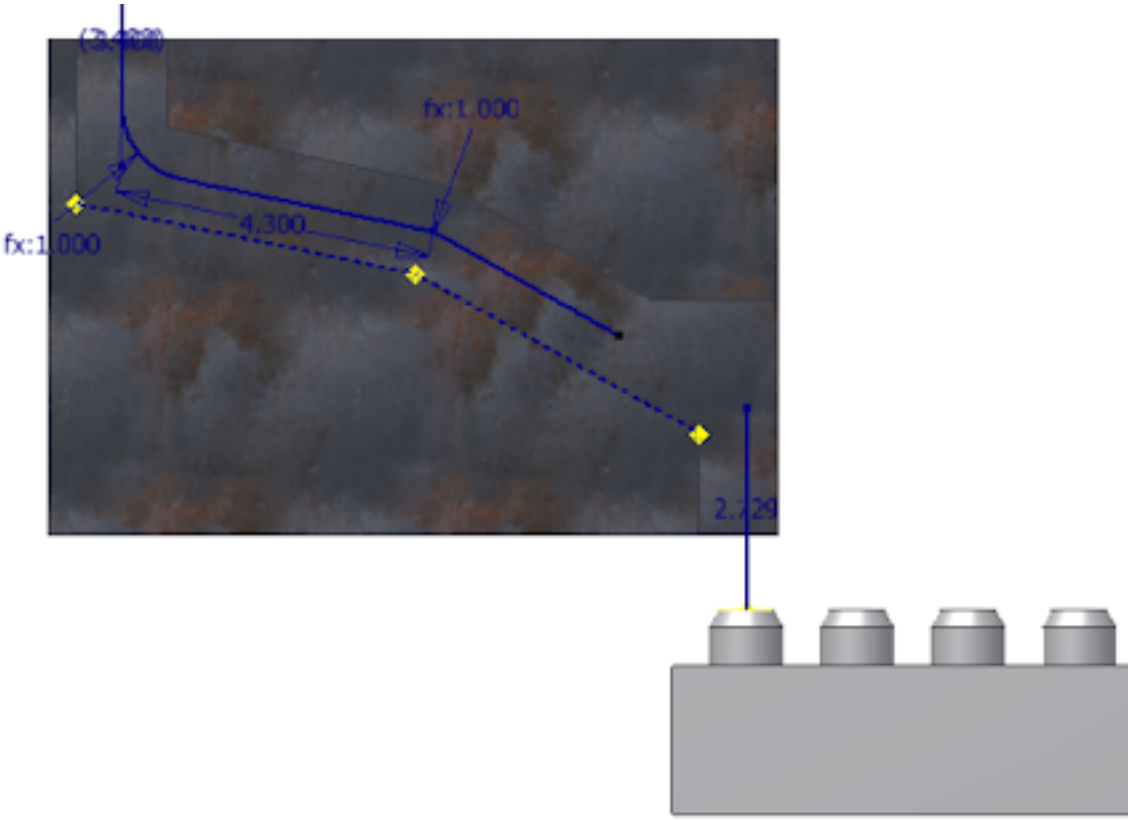
A dashed line representing the snap point is displayed from the cursor to the preview point. A preview point is displayed at the intersection of the line and the axis.



5 After the dashed line and the intersection point appear, click the sloping segment to set the snapped route point.



6 Right-click and select **Done**.



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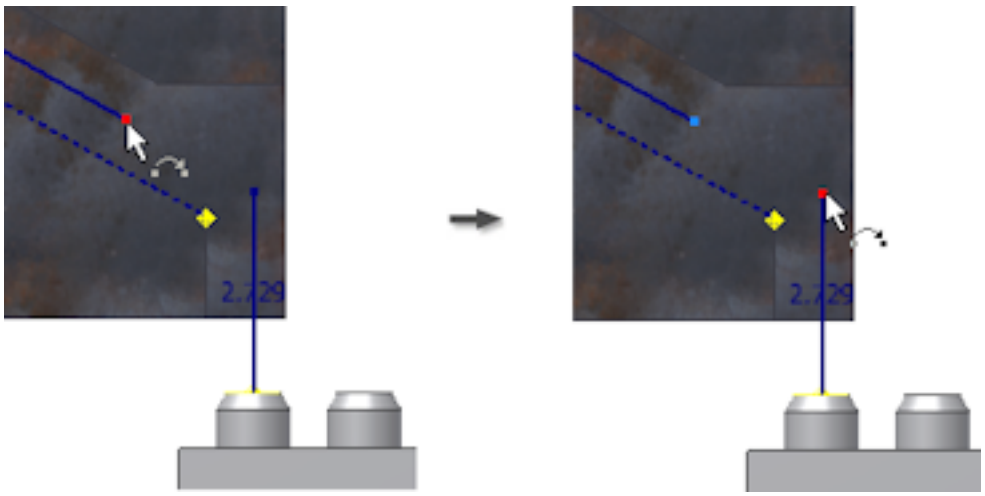
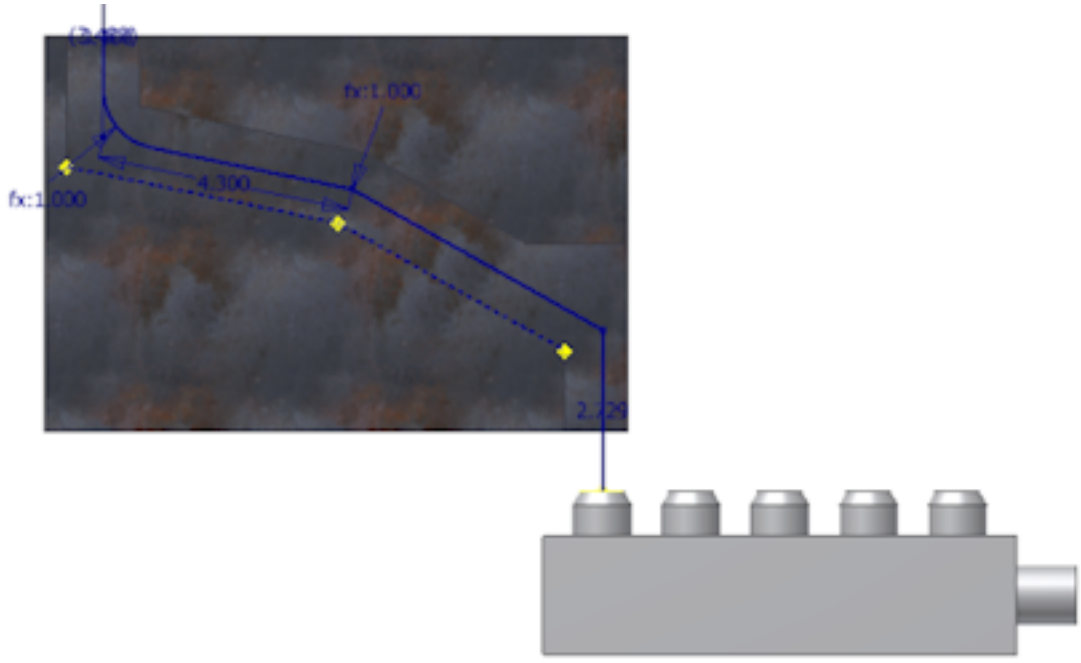
Join separate route points with the coincident constraint

1  On the ribbon, click **Route tab** ➤ **Constrain panel** ➤ **Coincident**.

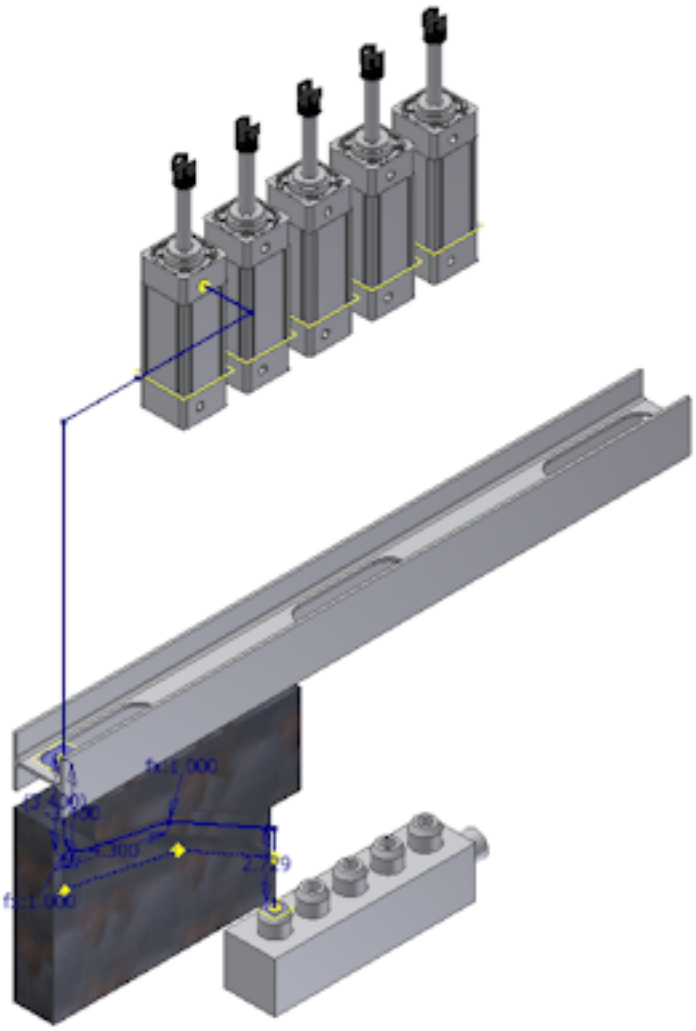
2 Select the two separate route points as shown in the following image.



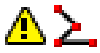
The separate sketches form a closed route.



3 Right-click in the graphics window and select **Done**.



4 Notice that an error icon displays beside the route.



Placing the coincident constraint causes an odd (not 90-degree, 45-degree, or combined135-degree) angle between the two segments so the route is in error. In the exercise that follows, you create a bend to fix the violation.

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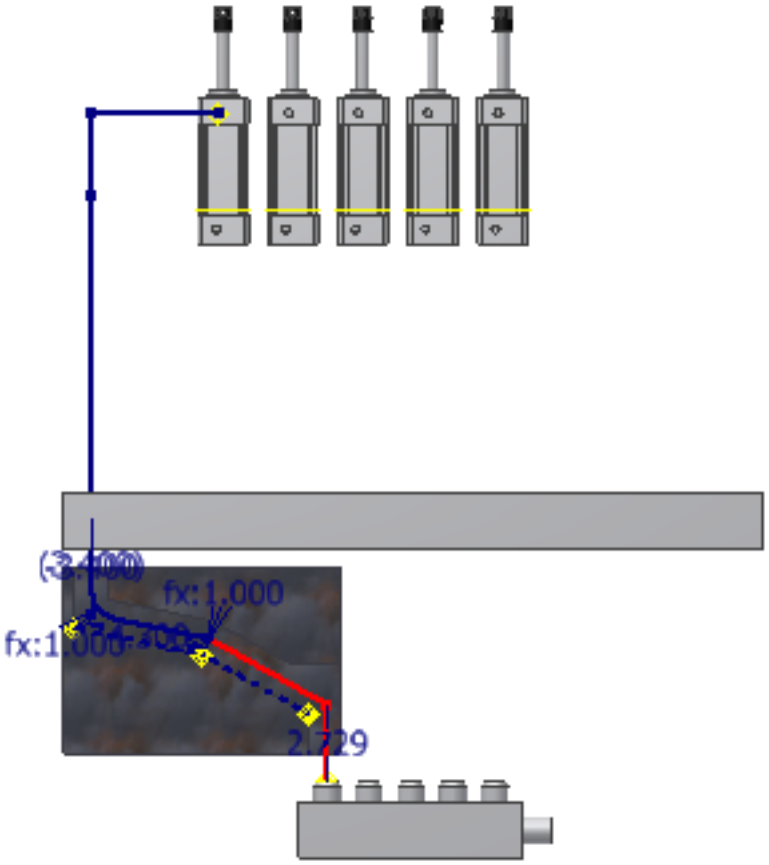
Add bends between segments to resolve errors

1. To verify the errors in Route01, right-click the route in the Model browser and select **Show Violations**.
2. To identify the route part in error, click the error on the Show Violations dialog box, and then view the error in the graphic window.

   The invalid angle is between the two segments as highlighted in the following image.

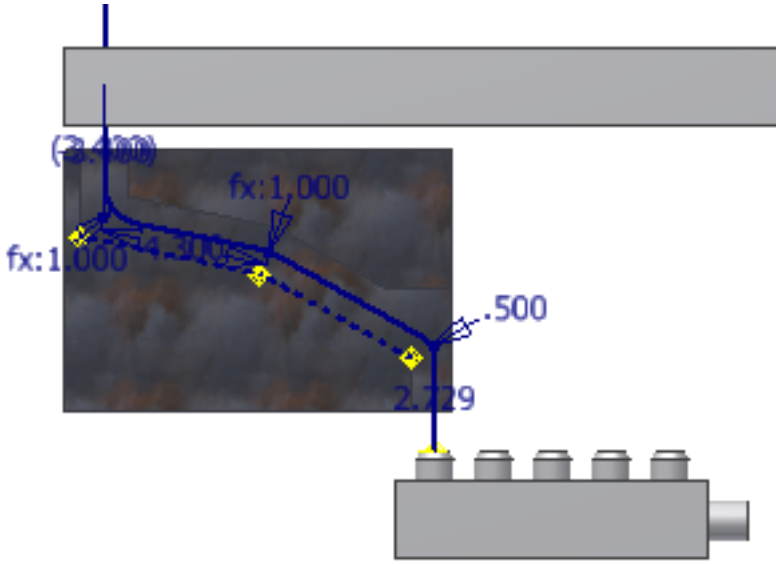


3 Click **OK** to close the Show Violations dialog box.

4 On the ribbon, click **Route tab ➤ Create panel ➤ Bends**.

5 Enter **0.5 inch** in the Bend dialog box. Then select the two segments that form the invalid angle.

The bend is created with a radius value of 0.5 inch. The error icon in the Model browser disappears.



6 Close the Bend dialog box.

7 To end the route, right-click and select **Finish Edit**.

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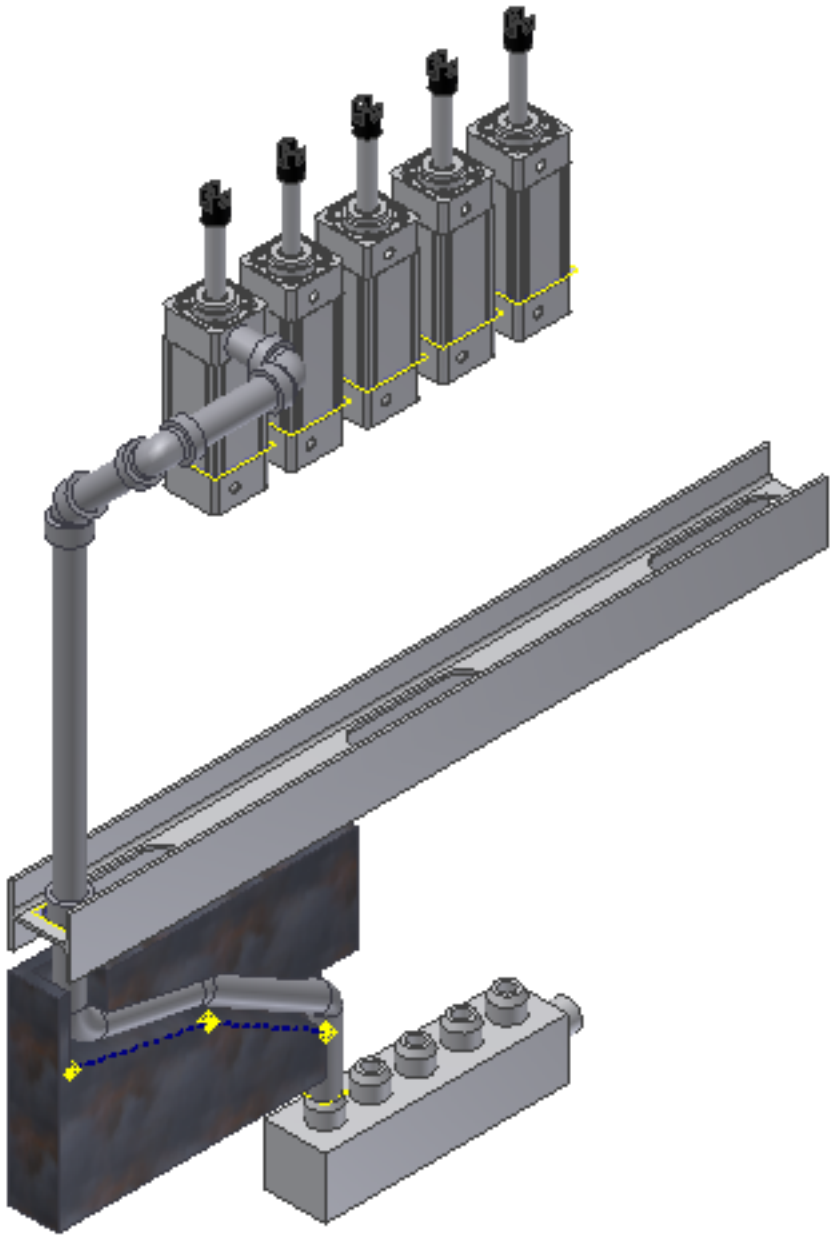
Populate the route

1 On the ribbon, click **Pipe Run tab ➤ Route panel ➤ Populate Route**.

The route is populated using the selected style. When a route is populated, all segments and fittings are added to the associated run folder.



2 To edit the current tube and pipe style, double-click Route01 in the Model browser.

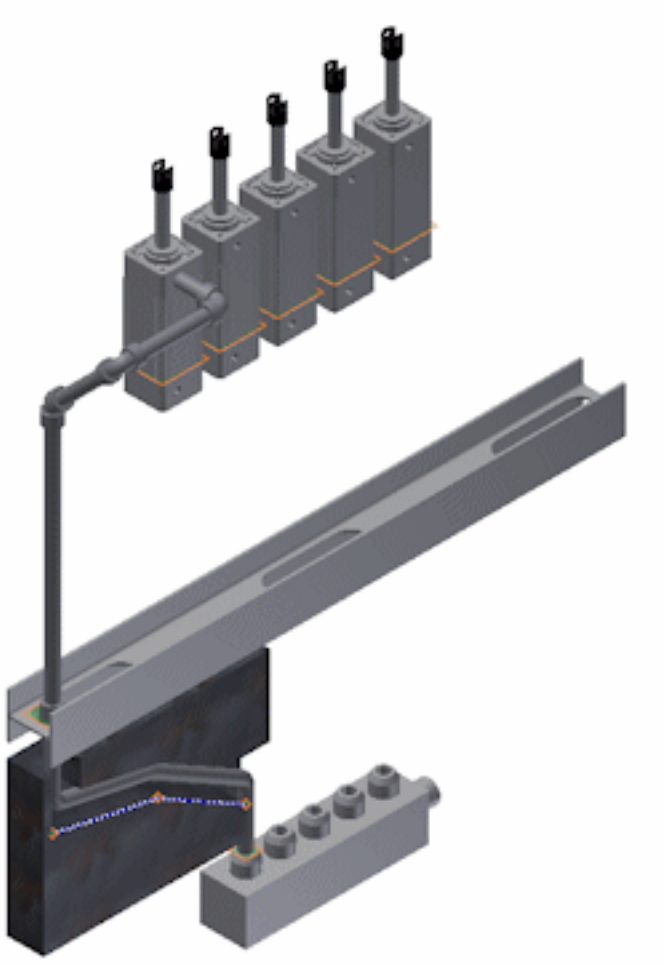
3 On the ribbon, click **Route tab ➤ Manage panel ➤ Tube and Pipe Styles**. The Tube & Pipe Styles dialog box displays.

4 Right-click the ASTM A53/A53M-ASME B16.11 - Welded Steel Pipe style in the style browser and click **Edit**.

5 Select **1/4 in** from the Diameter list, and click **Save**.

6 Click **Close** to close the dialog box.

7 Right-click and select **Finish Edit** to populate the route using the edited style.



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Create a hose route

1 Double-click **Tube & Pipe Runs** in the Model Browser.

2  On the ribbon, click **Tube and Pipe tab ➤ Run panel ➤ Create Pipe Run**.

3 On the Create Run dialog box, enter **AirSystem2** as the Run File Name. Accept the other defaults and click **OK**.

The individual run **AirSystem2:1** is added and activated in the browser. The Pipe Run tab displays on the ribbon.

4 On the Pipe Run tab, Manage panel, select the **Hydraulic Hose - Female Thread - Swivel** style from the Active Style list.

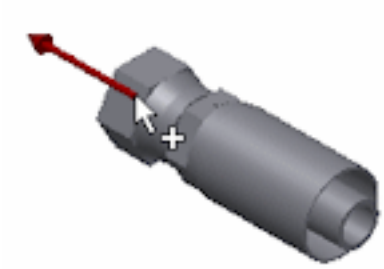
5  On the ribbon, click **Pipe Run tab ➤ Route panel ➤ New Route**.

6 On the Create New Route dialog box, accept the default route file name and location, and click **OK**.

A pipe route is added to the active run and activated. The Route tab displays on the ribbon.

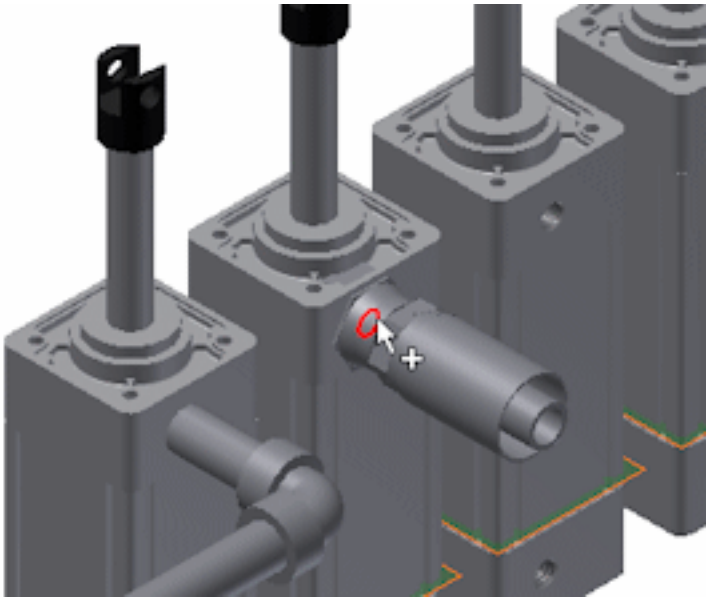
7  On the ribbon, click **Route tab ➤ Create panel ➤ Route**.

The start fitting specified by the hose style is attached to the cursor and ready for placement.



8 Move the cursor on the circular edge as shown in the following image. Then click to set the start fitting.

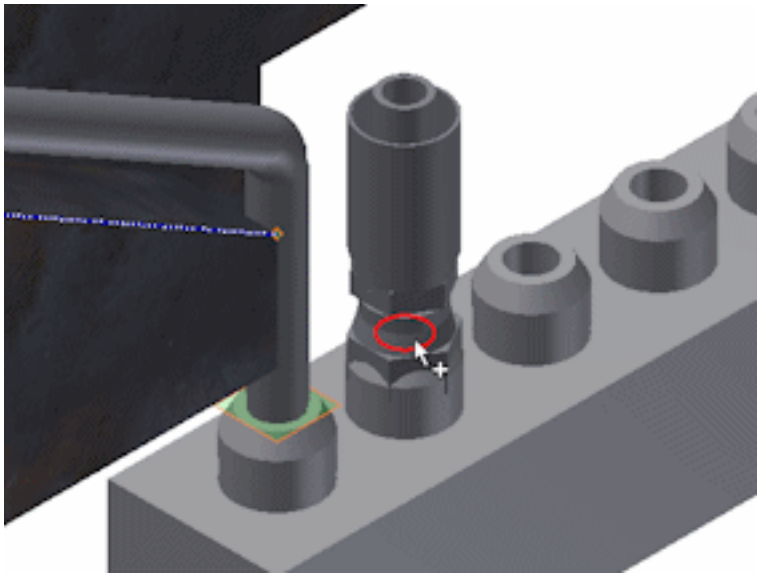
NOTE The direction axis on the part must point in the direction of the route.



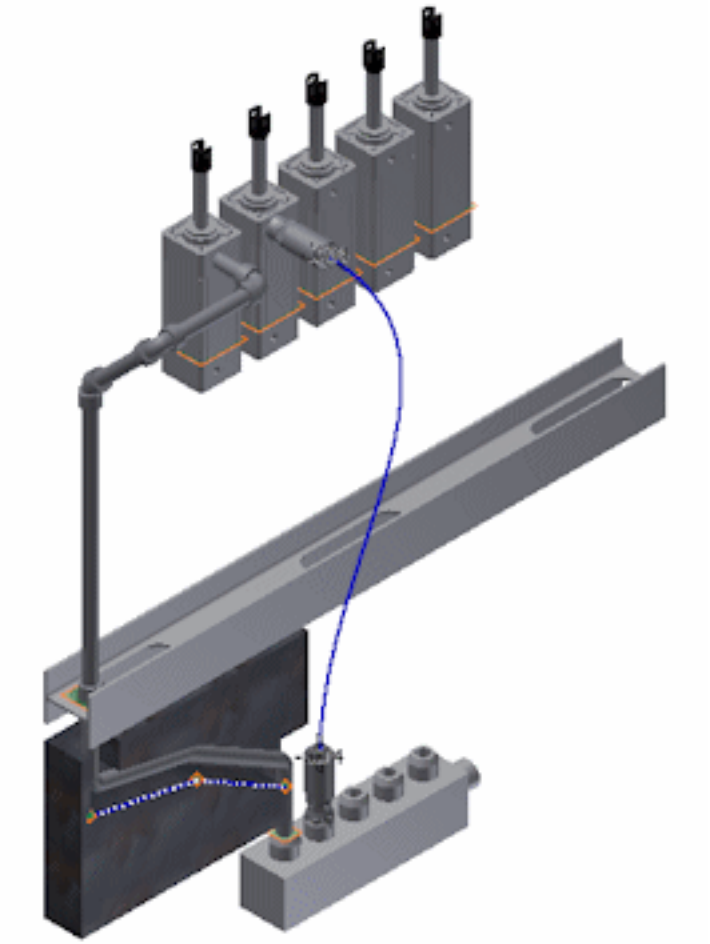
9 The end fitting appears and is attached to the cursor. Notice that the start fitting and end fitting reference the same part in this style.

10 Move the cursor to the circular opening on the valve part.

If the orientation of the fitting is not correct, right-click and select **Next Connection**.



11 Click to place the end fitting. The route line displays.



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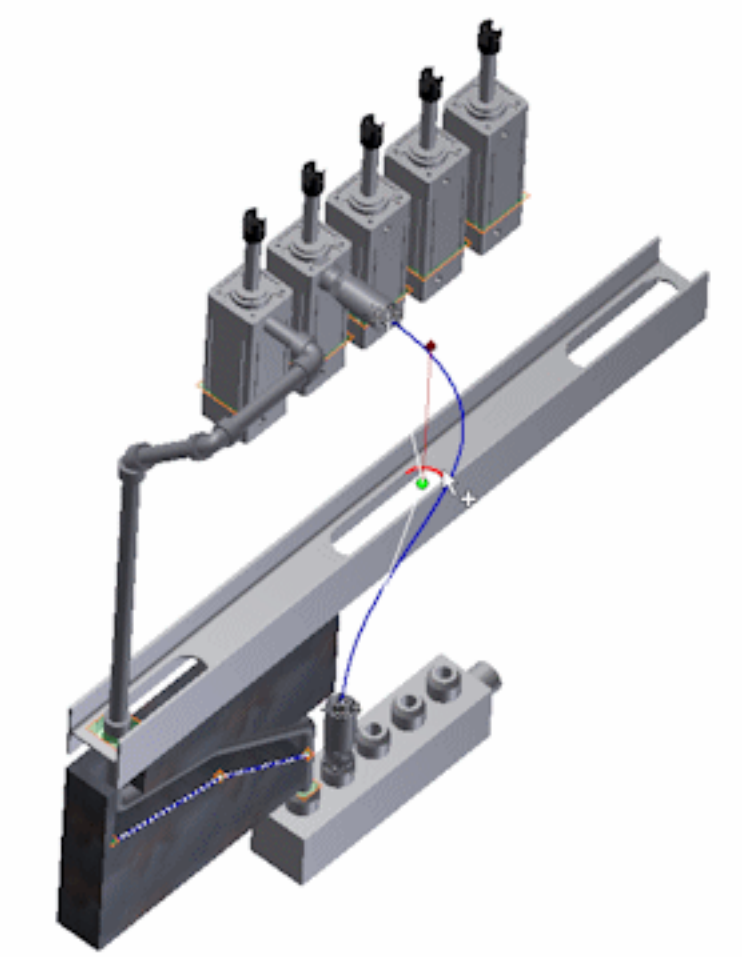
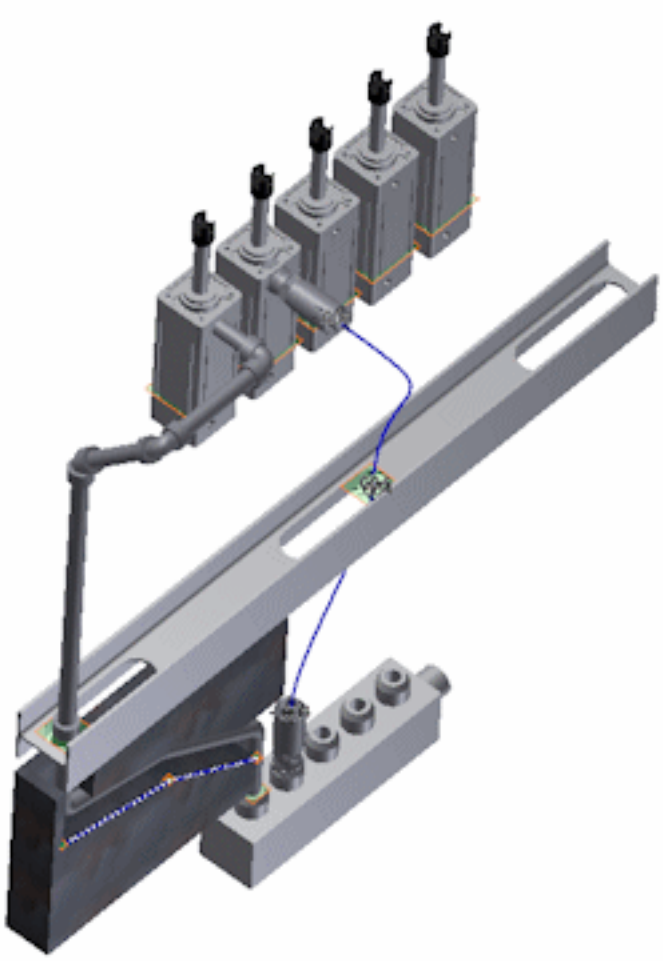
Finish and populate the hose route

1 To add an intermediate node tangent to a circular edge on IBeam, move the cursor close to the circular edge. The circular edge is highlighted.

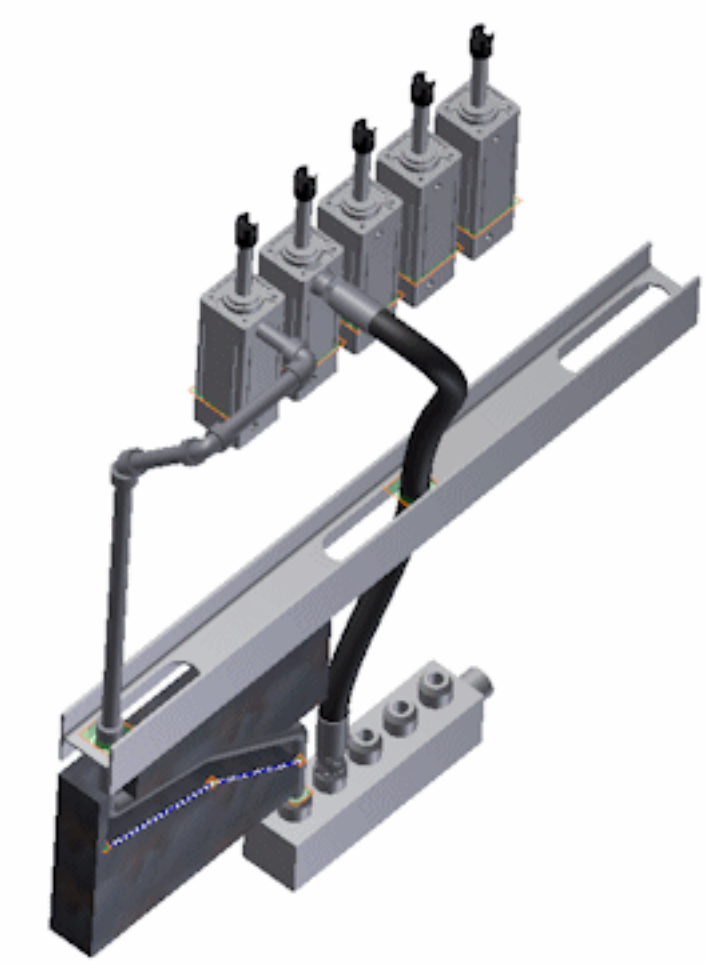


Then click to set the hose node.



2 Right-click and select **Finish Edit**.
The run environment is activated.

3  To populate the hose route, click **Pipe Run tab ➤ Route panel ➤ Populate Route**.
After the route is populated, a Flexible Hose part, start fitting, and end fitting are added to the Model browser. Notem them at the bottom of the flexible hose assembly.



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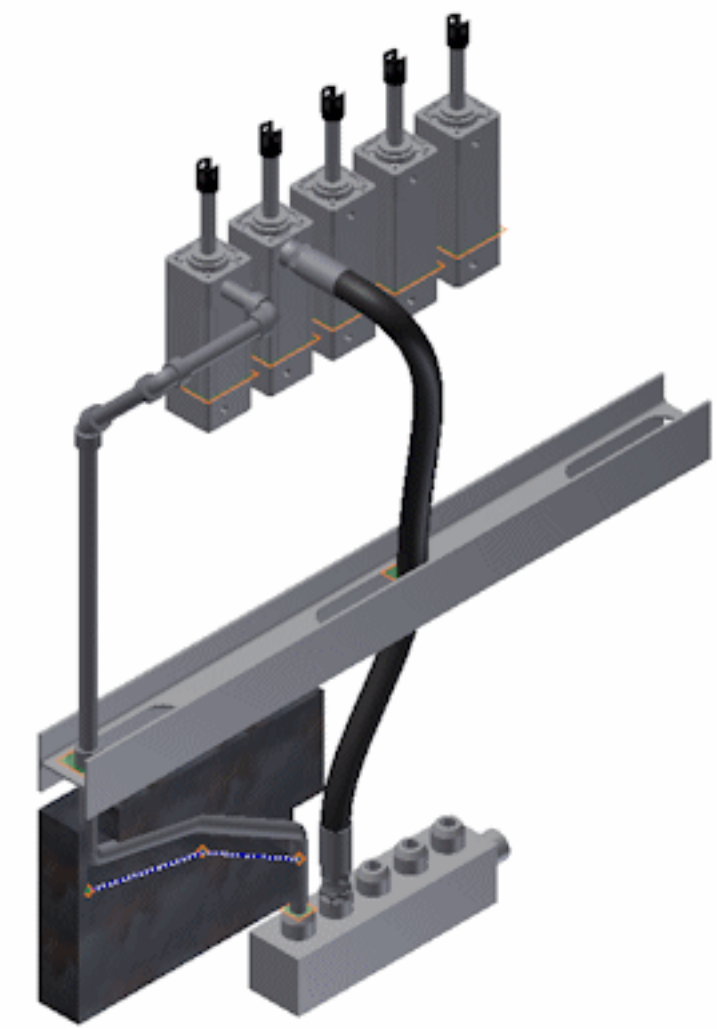
Summary



In this tutorial, you learned how to:

- Create a tube and pipe assembly.
- Set a tube and pipe style.
- Create rigid piping and bent tubing routes.

- Populate selected routes with library components.
- Edit a Tube and Pipe style.
- Create and populate a flexible hose route.

What Next? - As a next step, you can continue with the following Tube and Pipe tutorials. For example, learn more about Tube and Pipe Styles in the Tube and Pipe Styles. Then change the styles used on the AirSystemAssy.iam to correspond to your company standards.

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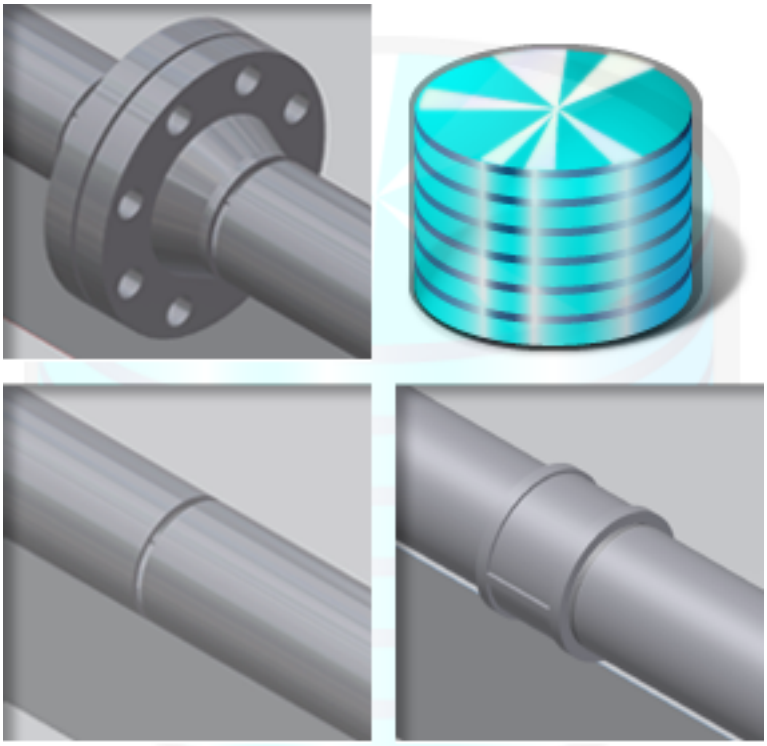
Tube and Pipe Styles

2

About this tutorial



Create and edit a tube style using the Tube & Pipe Styles dialog box.

Category	**Routed Systems**
Time Required	20 minutes
Tutorial File Used	Accumulator.iam

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

Tube and Pipe styles depend upon the Content Center. This tutorial uses components from the ANSI library. If this library is not available to you, substitute components from a different library or read along without performing the steps. If you use a library other than ANSI, it is not critical that components are the correct sizes, only that they work to demonstrate the workflow.

This tutorial is created for a single-user environment with libraries stored in a Desktop Content location. If you are a member of a workgroup that shares libraries, you must be a library administrator and have editor permissions to perform library editing tasks. Also the method for creating user libraries on the server is different than presented in this tutorial.

Objectives

- Create a style from scratch.
- Apply that style to an existing run.
- Edit a style.
- Create styles based on existing styles.
- Export and import styles.
- Add a style to the global template.

Prerequisites

- Connect to Content Center.
- Know how to set the active project and navigate the model space with the various view tools.
- See the Help topic “Getting Started” for further information.

Navigation Tips

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

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Get Started

Tube and Pipe styles use components from Content Center. To begin, set your active project to **tutorial_files**, and then verify that Content Center is installed and configured.

1. Click  ➤ **Manage** ➤ **Projects**.
2. On the Projects dialog box, double-click the **tutorial_files** project in the projects list to set it as the active project.
3. In the lower-right corner of the Projects dialog box, click **Configure Content Center Libraries**. 
4. On the Configure Libraries dialog box, verify that the **Inventor ANSI** library is available, and that the **In Use** box is selected.

   NOTE To perform the tutorial steps, at least one standard Content Center library must be available for use. A standard library is available if it is identified as **In Use** and has a status of **Read Only** in the **Access** column.

   TIP If no libraries are available, set up Content Center libraries first. See the Help for more details or contact your CAD Administrator.

5. Click **OK** to close the Configure Libraries dialog box.
6. Click **Save** in the Projects dialog box if it is available, and then click **Done**.

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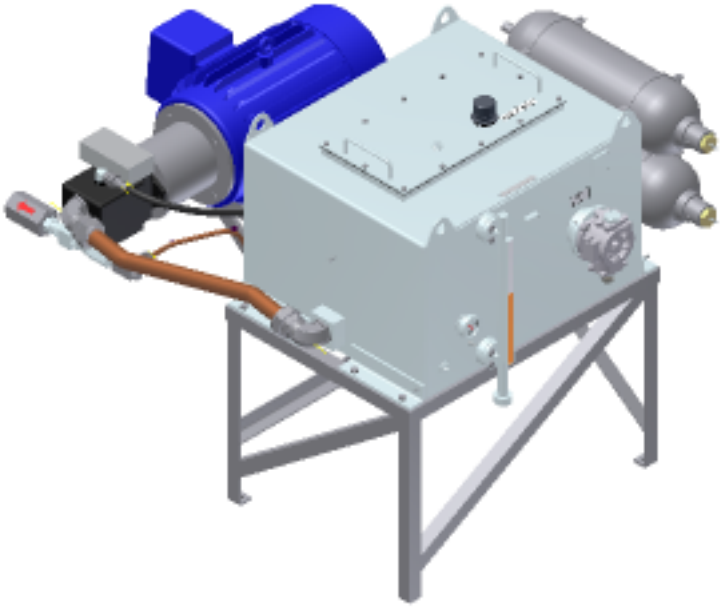
Open Tutorial File

Open an assembly from the tutorials files, and save a copy to work on.

1. Open **Accumulator.iam**, located in *\Tutorial Files\Tube & Pipe\Accumulator*.



2. Select  ➤ **Save As**, and use **Accumulator_TPStyles.iam** for the file name.

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The Tube & Pipe Styles Dialog Box

You use the Tube & Pipe Styles dialog box to create and modify styles. When the dialog box displays, the active style is listed in the **Active Style** field, and is also highlighted in bold in the style browser. The components used in this

style, along with various settings and parameters, display on the **General** and **Rules** tabs.

1. Double-click the **Tube & Pipe Runs** node in the browser to enter the Tube and Pipe environment.
2. On the ribbon, click **Tube and Pipe tab ➤ Run panel ➤ Tube and Pipe Styles**  to display the Tube & Pipe Styles dialog box.

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Create a Style

Next, you create a tube style. The name, pipe component, and rules are required. You also set the component appearance to differentiate this style from the style used in the existing run.

1. Click **New**  .
2. Enter **Medium Copper Tubing** in the **Name** field of the **General** tab. This field is the name of the style and is a required entry.
3. Enter **My Styles** in the **Category** field. Populating the **Category** field creates a folder nested under the active folder. This new category folder contains the **Medium Copper Tubing** style.

   NOTE Creating a category folder is optional. In this example, if you did not create a category folder, the **Medium Copper Tubing** style would be nested under the currently active folder.

   The **Category** field is also a drop-down menu. Had there been other user-created categories, they would also be available for selection from the menu.

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Specify Components

Specify the component from Content Center that is used for pipe when this style is active. The icons next to each component type indicate the status of the component within this new style definition.

 Indicates that the component is required for the style and must be selected from a Content Center library to define the style successfully.

 Indicates that the component can be selected from a Content Center library but is optional.

 Indicates that the component selection is satisfied.

 Indicates that the component is required for the style but is not available in your Content Center libraries.

1. Right-click the **Pipe** row, and then select **Browse** to display the Library Browser dialog box.
2. In the **Filters** area of the dialog box, check the **Standard** box, and then select **ASTM B 88** from the drop-down list.
3. Click  to apply the filter to the browser.
4. Select **ASTM B 88 Tube** from the browser.
5. Click **OK**. When this operation is finished, notice that the  icon appears next to the Pipe component, indicating that this selection is satisfied.

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Specify Components (continued)

Specify the component that is used for couplings when this style is active.

1. Right-click the **Coupling** row, and then select **Browse**.
2. In the **Filters** area of the dialog box, check the **Standard** box then select **ASME B16.18** from the drop-down list.

3 Click  to apply the filter to the browser.

4 Select **ASME B16.18 Coupling C x C** from the browser.

5 Click **OK**.

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Specify Diameter and Appearance

Next, you change the diameter to meet design criteria and the appearance to differentiate this style from the existing styles.

1 Ensure that **Nominal** is selected in the **Diameter** area, and then select **1 1/2 in** from the **Diameter** drop-down list.

2 Select **Aluminum (Machined)** from the **Component Appearance** drop-down menu.

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Specify Segment Rules

The final step for creating a style is specifying the rules for segments created with auto-routing. These rules include the minimum and maximum length, increment, and default bend radius.

1 Select the **Rules** tab.

2 In the **Minimum** field, enter `1 in`.

3 In the **Maximum** field, enter `200 in`.

4 In the **Increment** field, enter `0.5 in`.

5 In the **Default Radius** field, enter `2 in`.

6 Click **Save**. The **Medium Copper Tubing** style is added to the styles browser, nested in the new category. The new category is nested under the originally active existing category **Tubing with Bends**.

7 Select the **General** tab. The background color of the **Components** table changes, and the various options are not available to indicate that the style is no longer in edit mode.

8 Click **Close** to close the dialog box.

NOTE The new style is not saved in the assembly (the local template) until you save the assembly.

9 Click **Save** to save the assembly.

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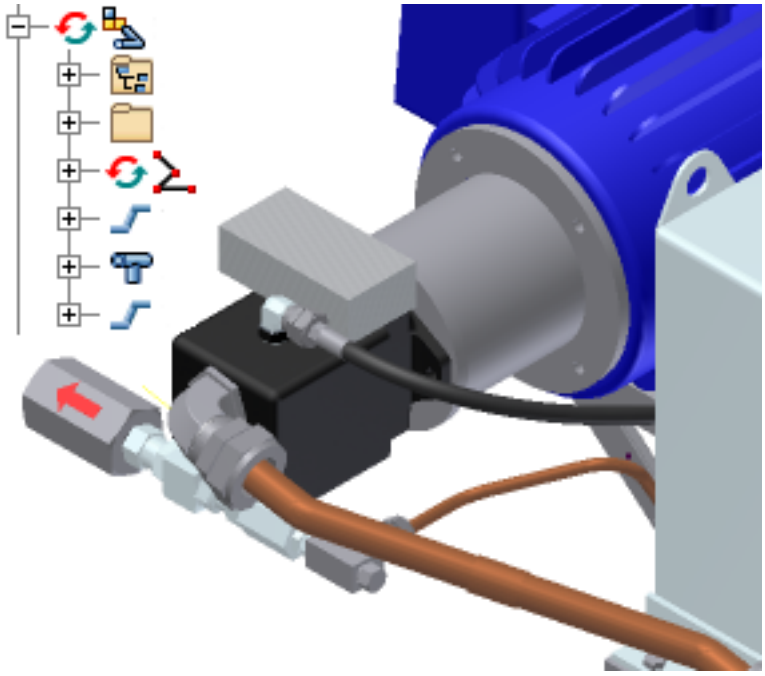
Apply the Style

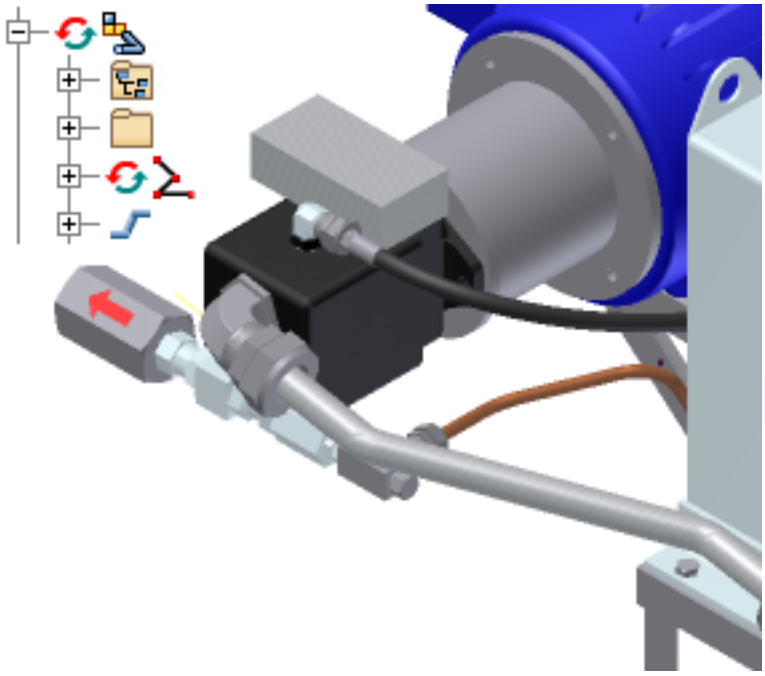
Next, you assign this style to an existing run. To make this assignment, edit a route and change the active style for that route. The segments update to the new style properties.

1 In the model browser, expand the **Pipe Run 1** node. Double-click the **Pipe Route** node to edit it.



2 The active style is listed in the **Manage** panel of the **Route** tab. Select **Medium Copper Tubing** from the drop-down list.

3 Click **Tube and Pipe tab ➤ Exit panel ➤ Finish Route** to return to the **Tube & Pipe Runs** level. The specified style is applied to the run.



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Edit the Style

Now, you edit the style and modify the rules for the Medium Copper Tubing style.

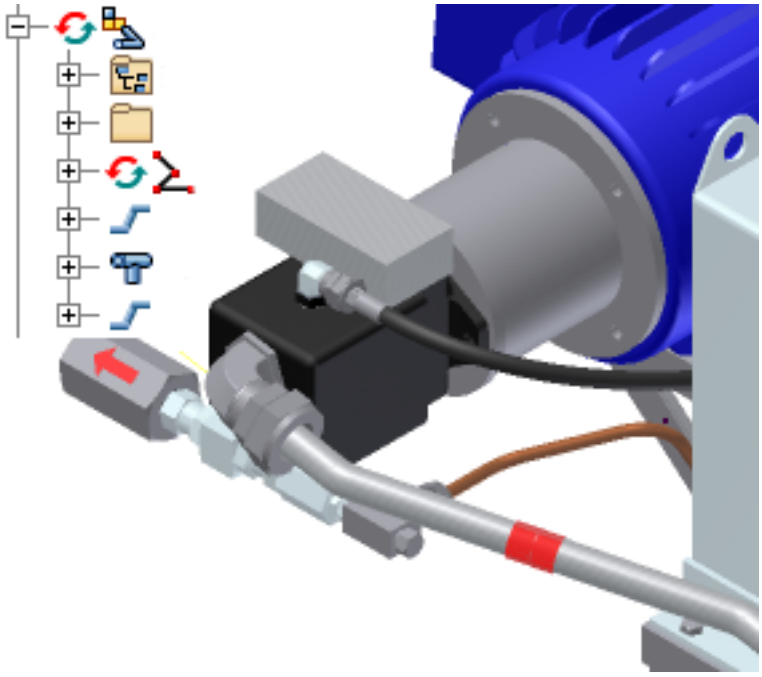
1 On the ribbon, click **Tube and Pipe tab ➤ Run panel ➤ Tube and Pipe Styles** .

2 Double-click **Medium Copper Tubing** to edit the style.

3 Click the **Rules** tab.

4 Enter `10 in` in the **Maximum** field of the **Segment Length** area.

5 Click **Save** and **Close** to close the dialog box. The run using the Medium Copper Tubing style is updated. The main segment exceeds 10 in, so a coupling is added.



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Copy and Export Style

Let's return to the Tube & Pipe Styles dialog box to examine a few more options.

1 On the ribbon, click **Tube and Pipe tab ➤ Run panel ➤ Tube and Pipe Styles**  .

2 In the styles browser, select a style other than the active style (**Medium Copper Tubing**). Right-click and take note of the various menu options. The commands at the top of the dialog box are also available on the context menu. Notice that you can select **Active** to make that style the active style.

3 Select **Medium Copper Tubing** and click  to create a copy of the style. (You could then select this copied style, click **Edit**, and edit the style as needed.)

4 Select the copy you just created (**Medium Copper Tubing_Copy(1)**) and click  to export the style.

5 In the Export Tube & Pipe Style Data File dialog box, enter **Style_Copy** in the **File Name** field, and then click **Save**. Take note of where you save the style. You need it to complete this tutorial.

6 An alert dialog box displays stating that the style exported successfully. Click **OK**.

NOTE In the operations above, you can use `Ctrl`-click or `Shift`-click to select multiple styles. With the **Export** command, the styles are exported to a single XML file.

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Delete and Import style

Use the **Delete** command to delete a style. The style is deleted from the local template only, not the global template. Select **Medium Copper Tubing_Copy(1)**, and then click **Delete** (as with **Copy** and **Export**, you can select multiple styles).

1 Select **Medium Copper Tubing_Copy(1)**, and then click  to delete the style (as with **Copy** and **Export**, you can select multiple styles).

2 Click **Yes** in the alert dialog box to confirm deleting the file.

3 Click to  import the style you deleted.

4 Select **Style_Copy.xml**, and click **Open**. The Import Tube & Pipe Styles dialog box activates and previews how the imported style appears in the styles browser structure.

NOTE If any issues are noted, an **Alert** icon appears next to the affected style. You can right-click the style to access more options to resolve the issue.

5 Click **OK** to import the style.

6 An alert dialog box displays containing information about the import. Click **OK**.

7 Click **Close** to close the Tube & Pipe Styles dialog box.

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Global Template

To this point, all edits have taken place in the local template only. You can create and import styles into the global template. Editing the global template would affect any new Tube and Pipe Runs you create. The global template is called **piping runs.iam** and is located in *\(your installation of Autodesk Inventor)\Design Data\Tube & Pipe*.

NOTE The **piping runs.iam** file is read only. To save changes to the file, disable the **Read-only** attribute in the File Properties dialog box.

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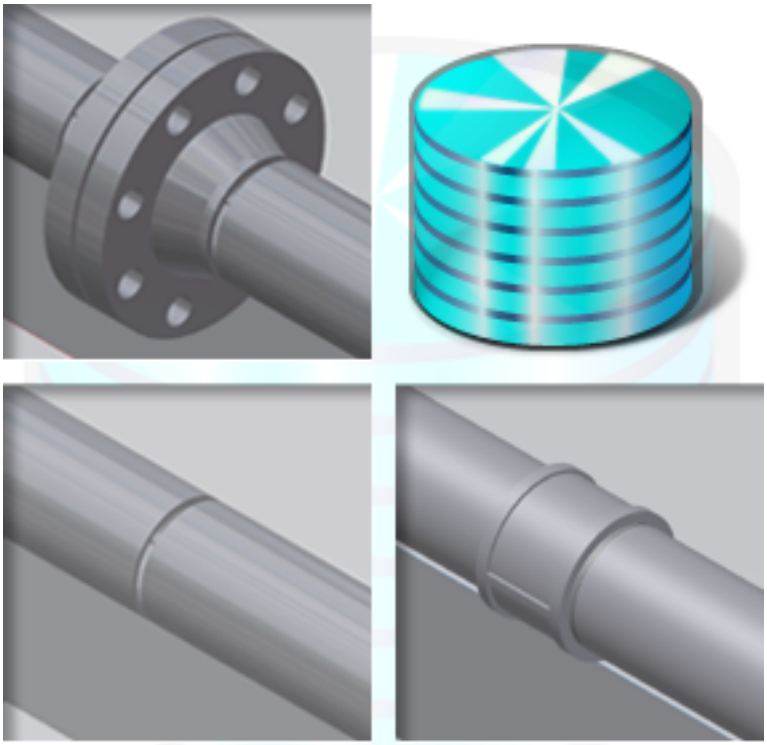
Summary

In this tutorial, you learned how to:

- Create a style from scratch.
- Apply that style to an existing run.
- Edit a style.
- Create styles based on existing styles.
- Export and import styles.
- Add a style to the global template.



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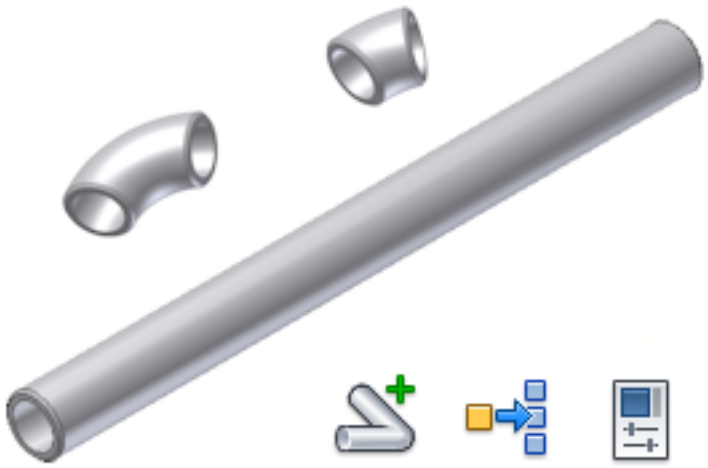
Author and Publish Tube and Pipe iParts

3

About this tutorial



Category	**Routed Systems**
Time Required	30 minutes
Tutorial Files Used	pipe.ipt, 45Elbow.ipt, 90LongElbow.ipt, Accumulator.iam

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

Objectives

- Author iParts.
- Define connection points and connection axes.
- Set up a new library.
- Publish iParts.
- Set a new style with published parts.

Prerequisites

- Basic knowledge of Tube and Pipe functionality. We recommend that you complete the other Tube and Pipe tutorials first.
- Basic knowledge of Content Center and user libraries.
- Basic knowledge of iParts.

Navigation Tips

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

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Author a pipe iPart

Create a custom component and transform it to an iPart factory. Then use the Tube and Pipe Authoring to set specific parameters for the iPart. After that, publish the iPart into a Content Center library.

You can also author and publish a part (not an iPart) as a fitting.

NOTE Authored non-library components can be directly placed into a tube and pipe assembly using the Place Fitting tool on the Pipe Run tab.

1 Open the **pipe.ipt** iPart file from the **Tutorial Files ➤ Tube & Pipe ➤ Example_iparts** folder.

Optionally, right-click **Table** in the Model browser and select **Edit Table** to review the iPart parameters in iPart Author dialog box. Then close the dialog box.

2 On the ribbon, click **Manage tab ➤ Author panel ➤ Tube and Pipe**. 

TIP If needed, expand the drop-down menu to select Tube and Pipe Authoring.

3 In the **Type** list, select **Pipes**. The default value in the Connections list is 2 and cannot be modified.

4 Verify that **Connection Number 1** is selected to indicate that you are setting the information for Connection 1.

In the **End Treatment** list, select **Welded**.

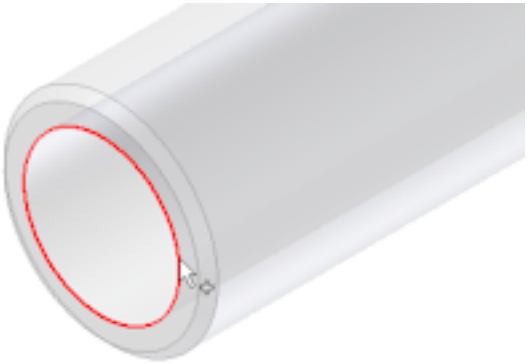
5 Click the red required fields, and select the attributes from the list as shown in the following image.

Parameter	Table Mapping
Nominal Size	NPS
Schedule Number	SCH
Inside Diameter	ID
Outside Diameter	OD
Pipe Length	PL

NOTE The table defines mapping between attributes in the iPart table and required attributes of a part authored in the Pipes category.

6 Click **Point** if needed, and then select the connection point in the graphics window.

- Pause the cursor over the geometry until the circular edge is highlighted as shown.

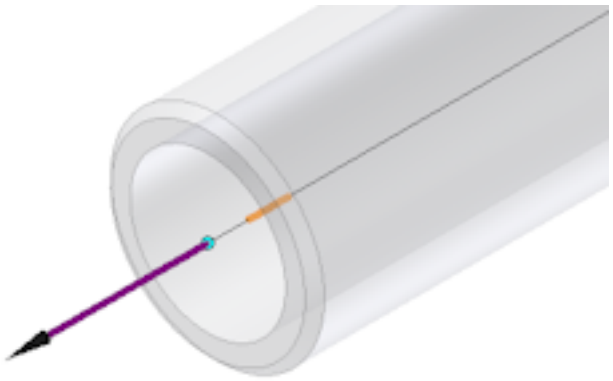


- Click to select the highlighted edge.

  A connection point is displayed at the center of the circular edge.

7 Click **Axis** in the Tube and Pipe Authoring dialog box, and then define the connection axis in the graphics window.

Pause the cursor over the same circular edge that you selected for the work point, and click to select. An arrow shows the axis direction.



TIP If the axis points toward the inside of the part, click **Flip Direction**. The connection direction must point toward the engaging fitting, so it often points outward.

8 Ensure **Neutral** connection is select. Neutral is automatically selected when the Welded end treatment is set.

9 Specify the **Engagement** setting: Select **Distance** in the list. Confirm that **0** is set in **Max** and **Min** fields.

NOTE If the Engagement section is not available, a required setting was not yet set. Complete all previous settings first.

TIP When connecting pipe parts, the connection only applies to the engagement definition of the adjacent fittings by default. Any engagement parameters specified for a pipe iPart are ignored by default for the connection to the adjacent fittings. It is recommended that you set a fixed value of 0 in such cases.

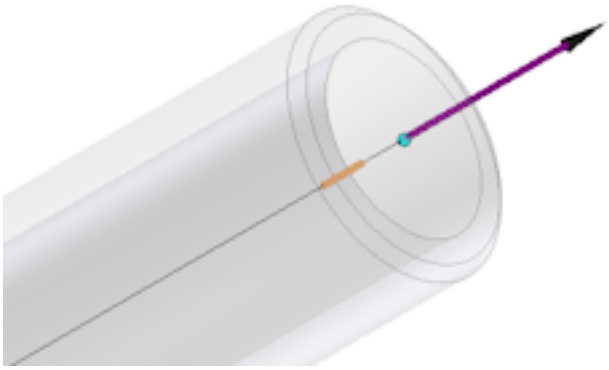
The Connection Number 1 is now black, indicating that all connection criteria are satisfied for Connection 1.

10 Click the **Connection Number 2** to begin defining the second connection.

The parameter mapping is populated with information set for Connection 1. Since the nominal sizes are the same, no editing is needed.

11 Set **End Treatment** to **Welded**.

12 To define the connection information, click the circular edge on the other pipe end to set the connection point and connect axis.



NOTE For the Tubes, Pipes, and Hoses part types being authored, you do not have to specify the Pipe Length parameter for connections other than Connection 1. The Pipe Length parameter is unique for a conduit part.

13 Specify the **Engagement** setting: Select **Distance** and confirm that **0** is set in **Max** and **Min** fields.

14 The Connection Number 1 and 2 are now black, indicating that all required connection criteria are set. Verify that all values are set properly.

Click **OK** to close the Tube & Pipe Authoring dialog box.

Then click **OK** on the Authoring Result dialog box.

15 Save the authored pipe iPart and close the part file.

All authoring changes are saved for the next time you author the iPart again. You can publish the authored pipe to the Content Center.

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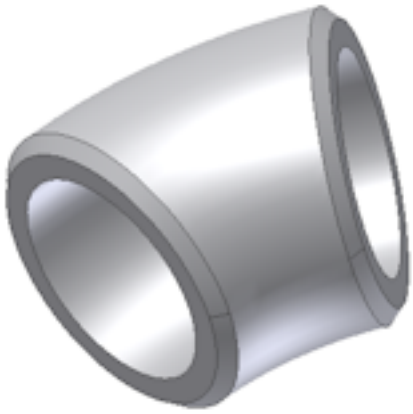
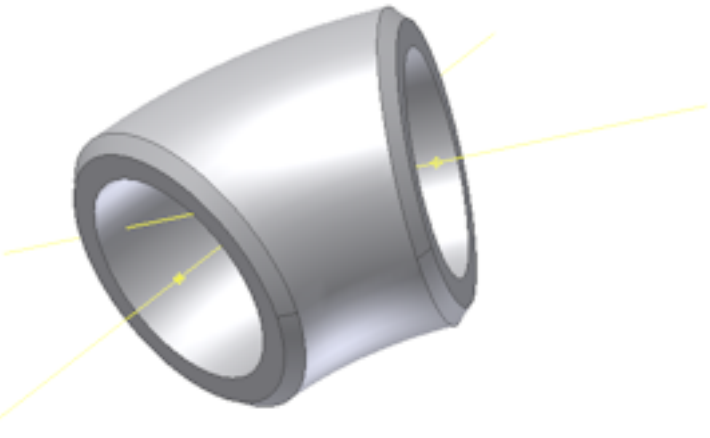
Author the 45-degree elbow iPart

1 Open the **45Elbow.ipt** iPart file. Use the View Cube or the Orbit to adjust the viewpoint to approximate the following image.



2 Switch on the Visibility of all part work axes and points. Select **Work Point 1 and 2**, and **Work Axis 1 and 2** in the Model browser, right-click, and select **Visibility**.



3 On the ribbon, click **Manage tab ➤ Author panel ➤ Tube and Pipe**. 

4 On the Tube & Pipe Authoring dialog box, specify:

`Type: Elbows`

`Connections: 2`

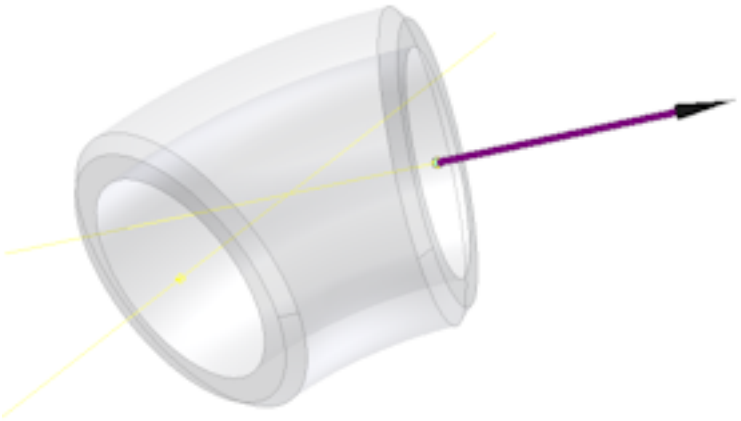
5 Set the following attributes for **Connection Number 1**:

`End Treatment: Welded`

`Parameter and Table Mapping: Nominal Size: NPS`

`Connection: Female`

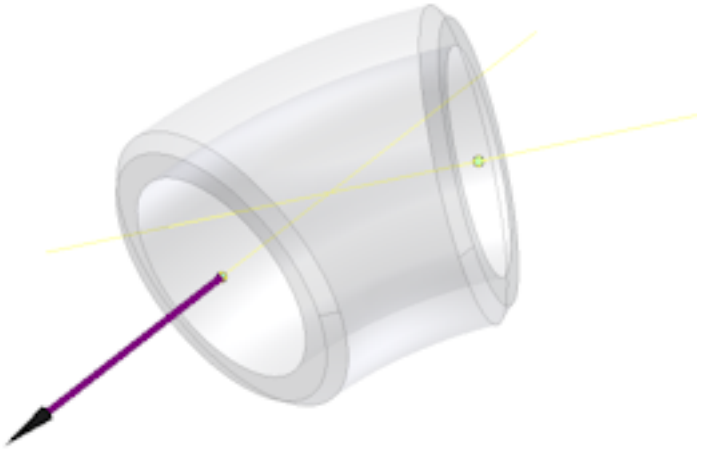
`Connection Point and Connection Axis: Select Work Point 1 and Work Axis 1`



Engangement

```
Max: Distance: 0 inch
Min % of Max: 0
```

6 Click **Connection Number 2**, and set the same attributes as for Connection Number 1 with the following exception: Select Work Point 2 as Connection Point and Work Axis 2 Connection Axis.



7 Specify the following ISOGEN properties:

```
Type: ELBOW
SKey: ELBW - Elbow - Butt Weld (90 degree and 45 degree)
```

ITEM-CODE: Enter 90 45 LLR

Description: Enter ELBOW 90 45 DEGREE BW ASTM A043 WPF316 SCH.40

8 Review all values for Connection 1 and 2, and then click **OK** to close the Tube & Pipe Authoring dialog box.

9 On the Authoring Result dialog box, click **OK**.

10 Save the authored 45-degree elbow iPart, and close the part file.

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Author the 90-degree elbow iPart

1 Open the **90LongEelbow.ipt** iPart file. Use the View Cube or the Orbit to adjust the viewpoint to approximate the following image.

2 Switch on the Visibility of all part work axes and points. Select **Work Point 1 and 2**, and **Work Axis 1 and 2** in the Model browser, right-click and select **Visibility**.

3 On the ribbon, click **Manage tab ➤ Author panel ➤ Tube and Pipe**. 

4 On the Tube & Pipe Authoring dialog box, specify:

Type: Elbows

Connections: 2

5 Click **Connection Number 1** and set the following attributes:

`End Treatment: Welded`

`Parameter and Table Mapping: Nominal Size: NPS`

`Connection Point and Connection Axis: Select Work Point 1 and Work Axis 1`

`Connection: Female`

Engagement

`Max: Distance: 0 inch`

`Min % of Max: 0`

6 Click **Connection Number 2**, and set the same attributes as for Connection Number 1 with the following exception: Select Work Point 2 as Connection Point and Work Axis 2 Connection Axis.

7 Specify the following ISOGEN properties:

`Type: ELBOW`

`SKey: ELBW - Elbow - Butt Weld (90 degree and 45 degree)`

`ITEM-CODE: 90 45 LLR`

`Description: ELBOW 90 45 DEGREE BW ASTM A043 WPF316 SCH.40`

8 Review all values for Connection 1 and 2, and then click **OK** to close the Tube & Pipe Authoring dialog box.

9 On the Authoring Result dialog box, click **OK**.

10 Save the authored 90-degree elbow iPart, and close the part file.

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Set up a user Content Center library

Read/write Content Center library must be available to publish parts. Review your current Content Center configuration and create a read/write library, if appropriate.

1 Click  ➤ **Manage** ➤ **Projects**.

2 On the Projects dialog box, confirm that the **tutorial_files** project is the active project. If needed, double-click the **tutorial_files** project in the projects list to set it the active project.

3 In the lower-right corner of the Projects dialog box, click the **Configure Content Center Libraries** button. 

The Configure Libraries dialog box displays a list of Content Center libraries in your library storage location.

4 Review the list of libraries. If a read/write library is available in the list, you can use this library and continue with step 9.

5 If you work in the Desktop Content environment, click the **Create Library** button on the Configure Libraries dialog box. 

NOTE If your Content Center libraries are stored on a server, create a library by using the server console. Then add the library to the configuration in the project. See the Help for more details.

6 Enter **My Tube and Pipe Library** in the **Display Name** field. The File Name uses the same string automatically.

7 Click **OK**. My Tube and Pipe Library is created in the Desktop Content folder.

8 A newly created library is automatically added to the Content Center configuration. Verify that the **In Use** box is selected for the My Tube and Pipe Library, and the library status is **Read/Write**.

9 Click **OK** to close the Configure Libraries dialog box.

10 Click **Save** in the Projects dialog box, and then click **Done**.

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Publish a pipe iPart

1 Open the **pipe.ipt** part you previously authored.

2  On the ribbon, click **Manage tab ➤ Content Center panel ➤ Publish Part**.

The Publish Guide dialog box displays.

3 Select **My Tube and Pipe Library** from the list and accept the default language, and then click **Next**.

4 Confirm that **Pipes** is selected from the category list and click **Next**.

5 Accept the default mapping between the iPart family columns and category parameters and click **Next**.

The default mapping comes from authoring. You do not have to change it.

NOTE In the mapping table, required family columns and category parameters represent a different background color from optional properties that follow.

6 Ensure that you have the three family key columns for the pipe part in the order of NPS, SCH, and PL. Click **Next**.

TIP If key columns from the published iPart do not satisfy the design, you can change the family key columns in this step. Key columns determine a part placed from the library to the assembly.

7 Specify the pipe part family properties as follows:

```
Family Name: pipe

Family Description: Custom Welded Pipe

Family Folder Name: pipe

Standard Organization: SampleOrganization

Manufacturer: SampleCompany

Standard: SampleStandard

Standard Revision: 1
```

NOTE Multiple styles can be used in a single tube and pipe run. Different styles can use different standards. So the Standards Organization and Standard information is required when you plan to use a differing standard part in a style.

8 Click **Next**.

The thumbnail image of the published part is displayed.

NOTE If appropriate, you can change the thumbnail image for future use in the Content Center. Click the Browse button and select a new thumbnail image.

9 Click **Publish**.

10 When the Publish dialog box prompts about the successful publish, click **OK**.

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Review the published part in Content Center

The published part is added to **Custom Library ➤ Tube & Pipe ➤ Conduits ➤ Pipes** category. Use Content Center Editor to verify the published pipe part family.

1. On the ribbon, click **Manage tab ➤ Content Center panel ➤ Editor**. 
2. On the Category View panel on the left, expand the **Tube & Pipe ➤ Conduits**, and click the **Pipes** category. Part families available in the Pipes category are listed in the panel on the right.
3. Scroll down in the Pipes category to locate the previously published pipe.
4. Right-click the **Pipe** family, and click **Family Properties** on the menu.
5. Review family properties in the Family Properties dialog box. Notice that the name of the Library is My Tube and Pipe Library. Then click Cancel to close the Family Properties dialog box.

   NOTE Family properties identify the part in Content Center. They can be used to locate the part in the Detail View or using Content Center Search. If appropriate, change family properties for the published part.

6. Right-click the **pipe** family, and click **Family Table** on the menu.
7. Review the family table of the **pipe** family. Then click Cancel to close the Family Table dialog box

   NOTE The family table includes all family members (parts) that can be placed in assemblies. If appropriate, edit the family table to add new family members, define new columns, change properties of existing columns, and so on.

8. Select **My Tube and Pipe Library** from the Library View list on the Content Center Editor dialog box. Notice that only content included in My Tube and Pipe Library displays and other content is filtered out.
9. Click **Done** to close the Content Center Editor dialog box.

TIP See the Content Center User Libraries tutorial or the Content Center Help for more information.

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Publish the 45-degree elbow iPart

1. Open the **45Elbow.ipt** iPart.
2.  On the ribbon, click **Manage tab ➤ Content Center panel ➤ Publish Part**.

   The Publish Guide dialog box displays.
3. Select **My Tube and Pipe Library** from the list and accept the default language, and then click **Next**.
4. Confirm that **Elbows** is selected from the category list and click **Next**.
5. Accept the default mapping between the iPart family columns and category parameters and click **Next**.
6. Accept the default family key column and click **Next**.
7. Specify the following family properties and click **Next**.

   ```
   Family Name: 45Elbow
   Family Description: Butt Weld 45 Elbow
   Family Folder Name: 45Elbow
   Standard Organization: SampleOrganization
   Manufacturer: SampleCompany
   Standard: SampleStandard
   Standard Revision: 1
   ```
8. Review the Thumbnail image and click **Publish**. Then click **OK**.

The part is published in **Tube and Pipe ➤ Fittings ➤ Elbows** category. You can review the published part by using Content Center Editor.

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Publish the 90-degree elbow iPart

1. Open the **90LongElbow.ipt** iPart.
2.  On the ribbon, click **Manage tab ➤ Content Center panel ➤ Publish Part**.

The Publish Guide dialog box displays.

3. Select **My Tube and Pipe Library** from the list and accept the default language, and then click **Next**.
4. Confirm that **Elbows** is selected from the category list and click **Next**.
5. Accept the default mapping between the iPart family columns and category parameters and click **Next**.
6. Accept the default family key column and click **Next**.
7. Specify the following family properties and click **Next**.

   `Family Name: 90LongElbow`

   `Family Description: Butt Weld 90 Long Elbow`

   `Family Folder Name: 90LongElbow`

   `Standard Organization: SampleOrganization`

   `Manufacturer: SampleCompany`

   `Standard: SampleStandard`

   `Standard Revision: 1`
8. Review the Thumbnail image and click **Publish**. Then click **OK**.

The part is published in **Tube and Pipe ➤ Fittings ➤ Elbows** category. You can review the published part by using Content Center Editor.

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Set a new style with published parts

1. Open the Accumulator.iam file from the **Tutorial Files ➤ Tube & Pipe ➤ Accumulator** folder.
2. Double-click Tube & Pipe Runs in the Model browser to enter the Tube and Pipe environment.
3.  On the ribbon, click **Tube and Pipe tab ➤ Run panel ➤ Tube and Pipe Styles** to display the Tube & Pipe Styles dialog box.
4. On the Tube & Pipe Styles dialog box, expand **Rigid Pipes and Fittings**. Then select **ASTM A53/A53M-ASME B16.11 - Welded Steel Pipe** from the list.

5  Click **New** to create a style based on the selected style.

6 On the General tab, enter **Custom Welded Pipe (1/4, 90)** in the Name field. Enter **My Styles** in the Category field.

7 In the Components table, right-click the Pipe row and select **Browse**.

8 In the Library Browser dialog box, select **Standard**. Then select **SampleStandard** from the list.

9 Click **Filter**, so the custom pipe you created is the only item listed.

10 Select the **pipe**, and then click **OK**.

11 In the Components table, right-click the **Elbow 90** and select **Browse**.

12 In the Library Browser dialog box, select **Standard**. Then select **SampleStandard** from the list.

13 Click **Filter**, so the custom elbow you created is the only item listed.

14 Select the **90LongElbow**, and then click **OK**.

15 In the Components table, right-click the **Elbow 45** and select **Browse**.

16 In the Library Browser dialog box, select **Standard**. Then select **SampleStandard** from the list.

17 Click **Filter**, so the custom elbow you created is the only item listed.

18 Select the **45Elbow**, and then click **OK**.

19 On the General tab under Diameter, set the following options:

```
Nominal Diameter: 1/4 inch
Schedule: 80
```

TIP Notice that the couplings are not required for welded routes, so they are listed as optional. Pause the cursor over the symbol to view the tooltip.

20 On the Rules tab, specify:

```
Min: 1 inch
Max: 200 inch
Inc.: 0.1 inch
```

21 Click **Save**.

22 Click **Close**.

The new style Custom Welded Pipe (1/4, 90) is created and available for creating new rigid pipe routes.

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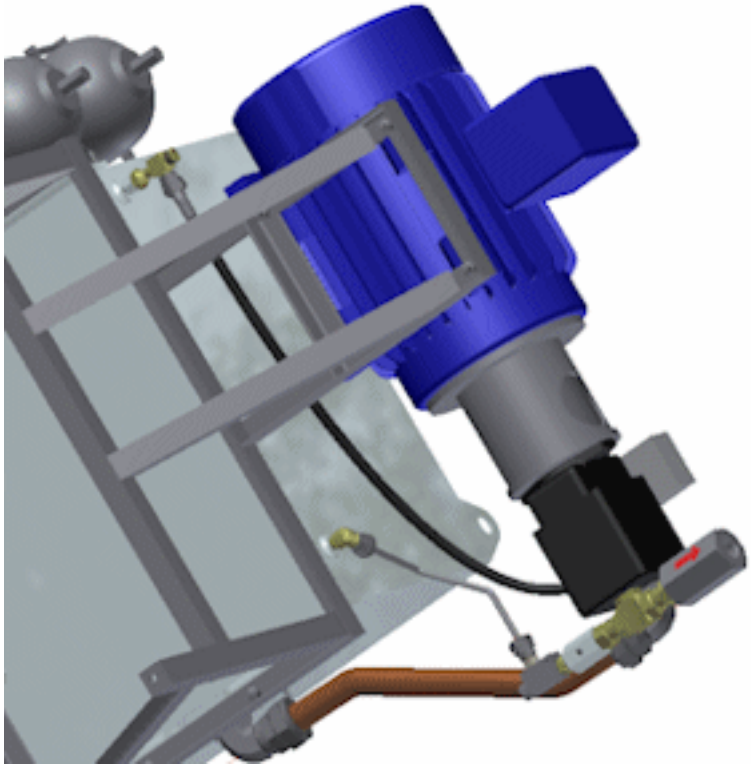
Summary



In this tutorial, you:

- Authored three iParts.
- Set up a new Content Center library.
- Published iParts in the library.
- Created a style using published parts.

What Next? - As a next step, you can create a rigid pipe route in the Accumulator assembly, and use the new style to populate the route. Or, you can apply the new style to Pipe Run 2 in the Accumulator assembly. See the Get Started with Tube and Pipe tutorial or the online help for more information. Alternatively, continue with other Tube and Pipe tutorials.

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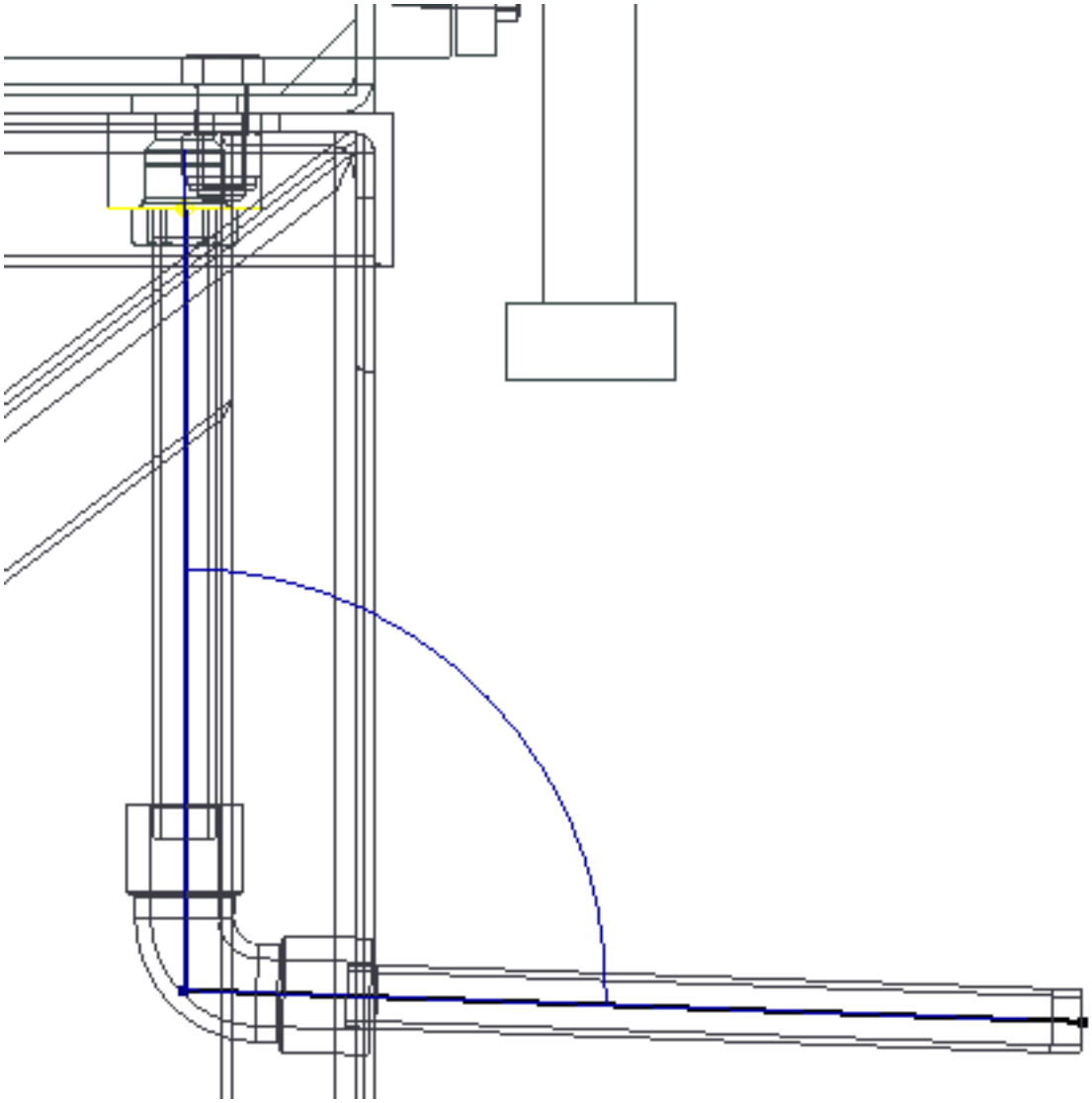
Self-draining Lines

4

About this tutorial



Author and publish a custom elbow to use in a self-draining lines style.

Category **Routed Systems**

Time Required	30 minutes
Tutorial File Used	Connect to Content Center and access a read/write library. If no read/write library is available, the tutorial shows you how to create one.

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

Self-draining lines maintain a slope, or fall, so that gravity drains the run. For example, the portions of a route that would otherwise be horizontal are sloped two degrees relative to horizontal.

To create a self-draining run, you first create a self-draining lines style, then create the run with that style active. Self-draining lines utilize fittings with bend or junction angles greater than 45 degrees, but less than 90, such as 88 degrees.

This tutorial is created for a single user environment with libraries stored in a Desktop Content location. If you are a member of a workgroup that shares libraries, you must be a library administrator and have editor permissions to perform library editing tasks. Also method for creating user libraries on the server is different than presented in this tutorial.

The examples in this tutorial use content based on several different standards. If your Content Center configuration does not contain the same library, read along without performing the steps, or use a similar part from a different library. For example, if the tutorial calls for a machine screw from the ISO standard, and your library contains only ANSI parts, substitute a similar screw from your ANSI library.

Objectives

- Create a self-draining lines style.
- Create a self-draining lines route.
- Populate the self-draining lines route.
- Create a read/write library in Content Center.

Prerequisites

- Know how to set the active project and navigate the model space with the various view tools.

- See the Help topic “Getting Started” for further information.

Navigation Tips

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

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Configure Standard and User Libraries

A standard Content Center library and a read/write library must be available to perform this exercise. You review your current Content Center configuration and create a user library.

1. Click  ➤ **Manage** ➤ **Projects**.
2. In the Projects dialog box, double-click the **tutorial_files** project in the projects list to set it as the active project.
3. In the lower-right corner of the Projects dialog box, click **Configure Content Center Libraries**. 
4. In the Configure Libraries dialog box, verify that the **Inventor ANSI** library is available and that the **In Use** box is selected.

   NOTE To perform the tutorial steps, at least one standard Content Center library must be available for use. A standard library is available if it is identified as **In Use** and its status is listed as **Read Only** in the **Access** column.

   TIP If no libraries are available, set up Content Center libraries first. See the Help for more details or contact your CAD Administrator.

5. In the Configure Libraries dialog box, click **Create Library**. 
6. Enter **RW Temp** in the **Display Name** field. The **File Name** uses the same string automatically.
7. Click **OK**. The **Tutorial Library** is created in the **Desktop Content** folder.

8. A new library is automatically added to the Content Center configuration. Verify that the **In Use** box is selected for the **Tutorial Library**, and the library status is **Read/Write**.
9. Click **OK** to close the Configure Libraries dialog box.
10. Click **Save** in the Projects dialog box, and then click **Done**.

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Author the Elbow

The custom fitting is the definitive component in a self-draining style. The custom fitting determines the slope angle and is the functional difference between a self-draining style and a style that is not self-draining.

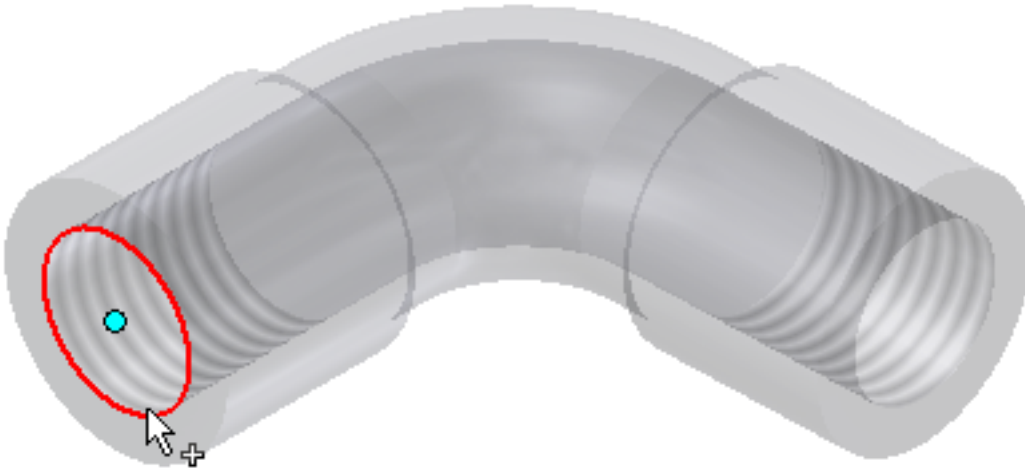
Authoring prepares a part for publishing to the Content Center by adding specific data to the model. Use Tube and Pipe Authoring to complete the authoring process before publishing to Content Center.

1. Open **self-drain_elbow.ipt**, located in the **Tube & Pipe** directory.

This elbow was created so that there are 88 degrees between connection axes. The elbow is ready for authoring and publishing. No further design work is needed.

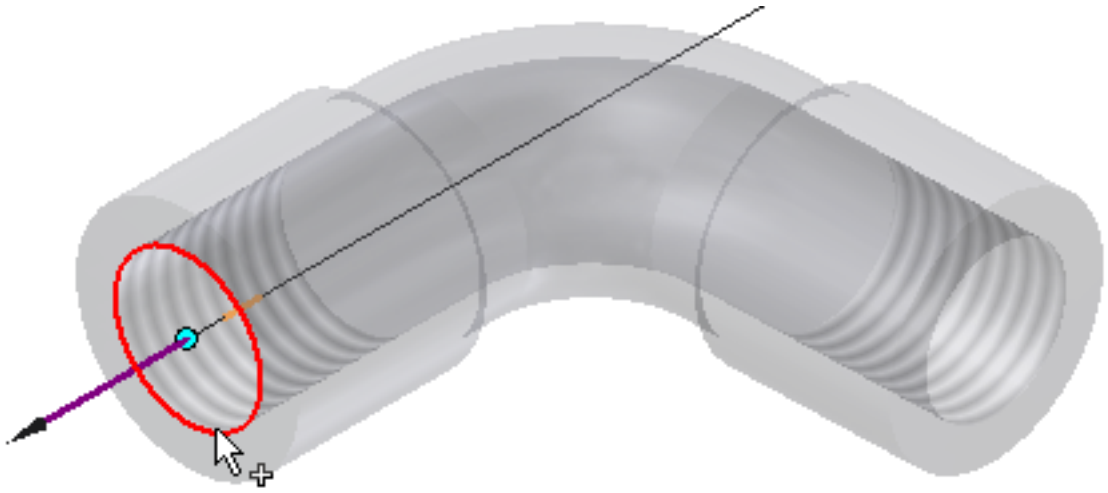
2 On the ribbon, click **Manage tab ➤ Author panel ➤ Tube and Pipe**  .

3 To set the connection point for the first connection, select the circular edge.



4 Click the **Axis** selector.

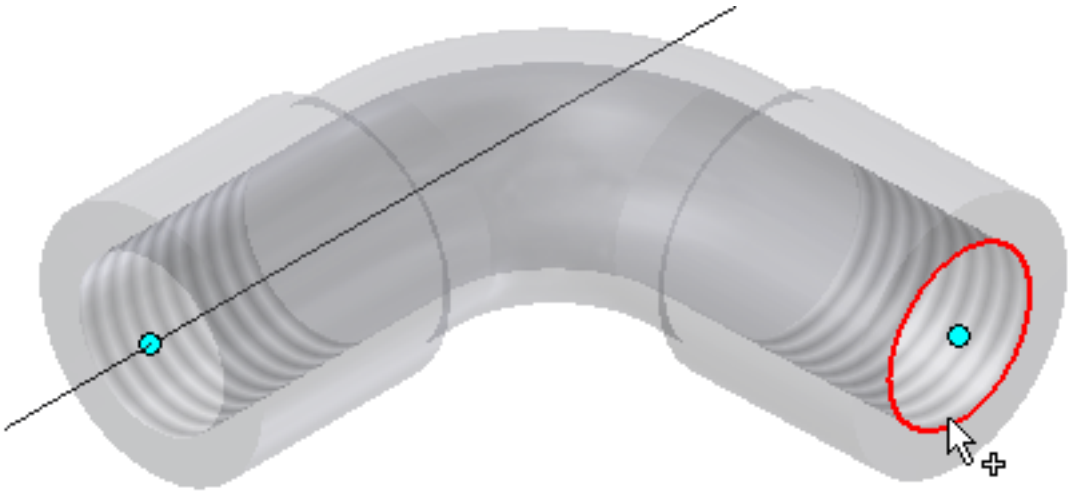
5 Select the circular edge again.



6 Set the **Nominal Size** to **3/4**.

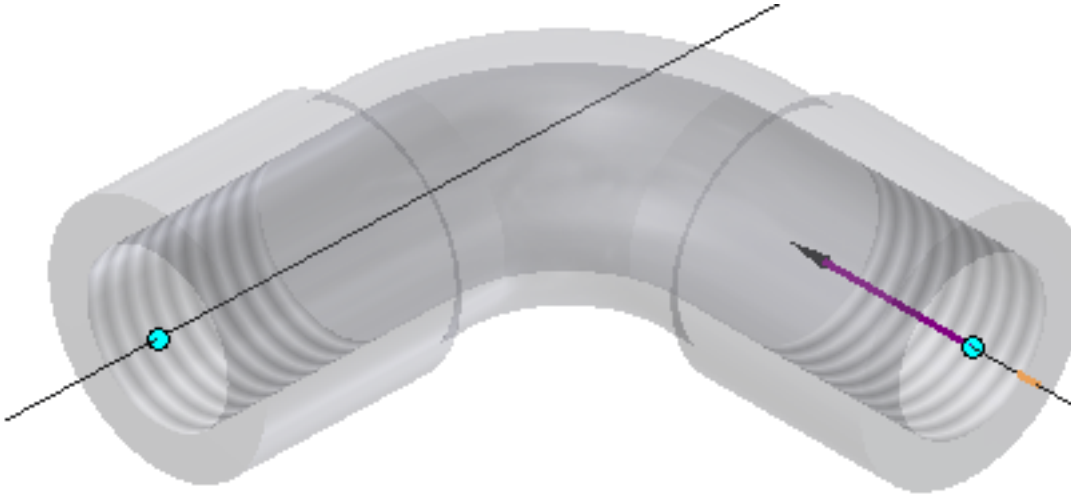
7 Change your selection to **Connection Number 2** .

8 Select the other circular edge to set this connection point.

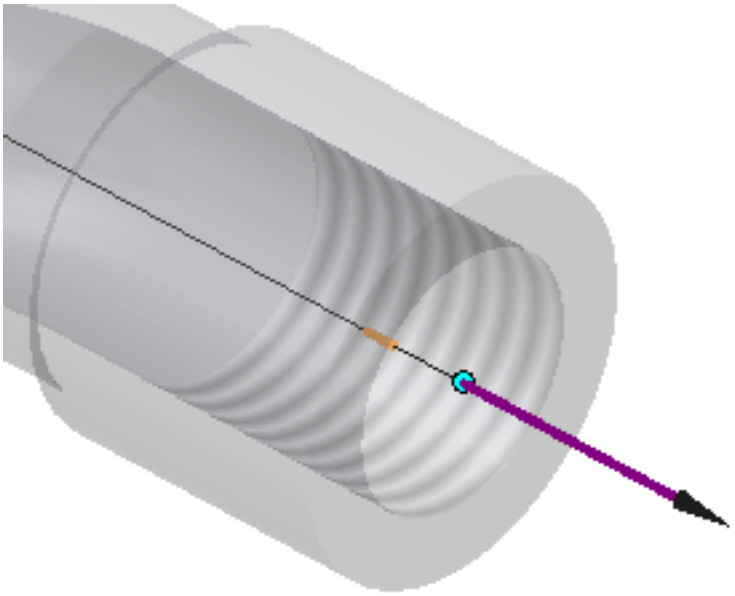


9 Click the **Axis** selector.

10 Select the circular edge again.



Flip the direction of the connection vector so that it points out and away from the elbow. Both connection vectors point out and away from the fitting.



The **Angle** field shows the angle between the two axes. In this example, 88 degrees is used.

11 Set the **Nominal Size** to **3/4**.

12 In the **Engagement** area, select **Distance** from the **Max** pull-down menu, and enter `1` as the value.

13 Ensure that the **Min** value is set to **70** (for **% of Max.**).

14 Click **OK**.

A message indicates that the authoring was successful.

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Publish the Elbow

Publishing the part adds it to the Content Center. Publishing parts requires a read/write library which we configured earlier in the tutorial.

1 On the ribbon, click **Manage tab ➤ Content Center panel ➤ Publish Part**  .

2. Select **RW Temp** (or your read/write library) from the library menu.
3. Click **Next**.
4. Ensure that **Elbows** is specified in the category menu.
5. Click **Next**.
6. Click **Next** again.
7. In the **Table Columns** field, select the **NominalSize** parameter, then click **Add** to add this parameter to the **Key Columns** field.
8. Click **Next**.

   The part name is entered in the **Family Name** field by default. Leave this field unchanged.
9. Click **Next**.
10. Click **Publish**.

    A message indicates that the publish operation was successful.
11. Close **self-drain_elbow.ipt**. Do not save changes.

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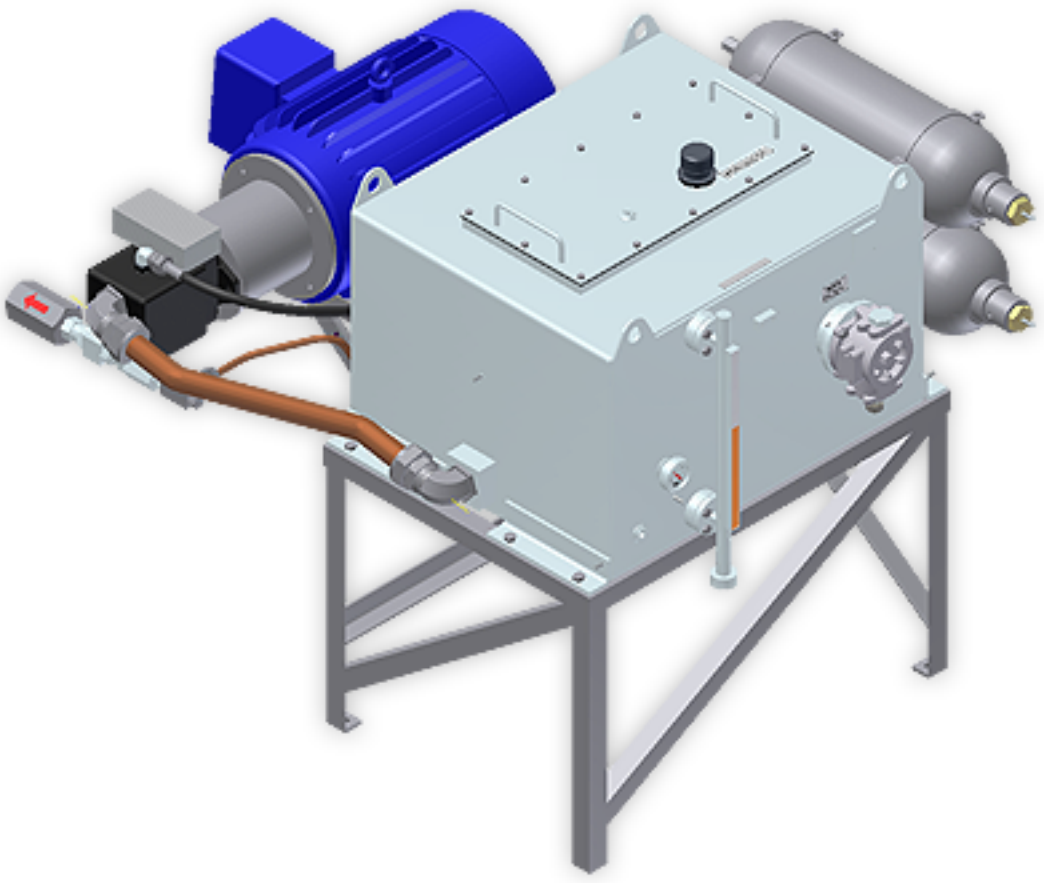
Open Tutorial File

Open an assembly from the tutorials files and save a copy to work on.

1. Open **Accumulator.iam**, located in *\Tutorial Files\Tube & Pipe\Accumulator*.



2 Select  ➤ **Save As**, and use **Accumulator_SDLine.iam** for the file name.

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Create the Self-draining Style

We now create a self-draining style using the elbow we authored and published.

1. On the ribbon, click **Environments tab ➤ Begin panel ➤ Tube and Pipe**  .
2. In the **Run Assembly File Name** field of the Create Run dialog box, type **SD Run**.
3. Click **OK**.
4. Click **Pipe Run tab ➤ Manage panel ➤ Tube and Pipe Styles**  .
5. In the style browser of the Tube & Pipe Styles dialog box, expand the **Rigid Pipe with Fittings** node.
6. Right-click the **ASME B36.10M-ASME B16.11 – Steel Threaded Pipe** style, and select **Copy**. You modify this copy to create the self-draining style.
7. Right-click **ASME B36.10M-ASME B16.11 – Steel Threaded Pipe_Copy(1)**, and select **Active** to make the copied style active.
8. Right-click the style, and select **Edit**.

   NOTE Instead of using the context menu, you can use the commands at the top of the dialog box to manipulate the style.

9. On the **General** tab, select the **Self Draining** option. When you select the **Self Draining** style type, the Elbow Custom component is added to the component list. The  icon indicates that the component is required and has not yet been specified from the Content Center.
10. Because the elbow you published earlier is not an iPart, it is available in one size only. In the **Diameter** box, set the **Nominal Diameter** to **3/4 in**.
11. Right-click **Elbow Custom,** and select **Browse**. It accesses components in the Content Center. It can take a few moments.
12. When the Library Browser dialog box activates, select the elbow (**self-draining_elbow**) you published earlier.

13 Click **OK**.

14 In the **Name** field of the Tube & Pipe Styles dialog box, replace the **_Copy(1)** portion of the default style name with **SD Temp**.

15 Click **Save**.

16 Before you close the Tube & Pipe Styles dialog box, ensure that this self-draining style is active. (It is highlighted in **bold** in the style browser.)

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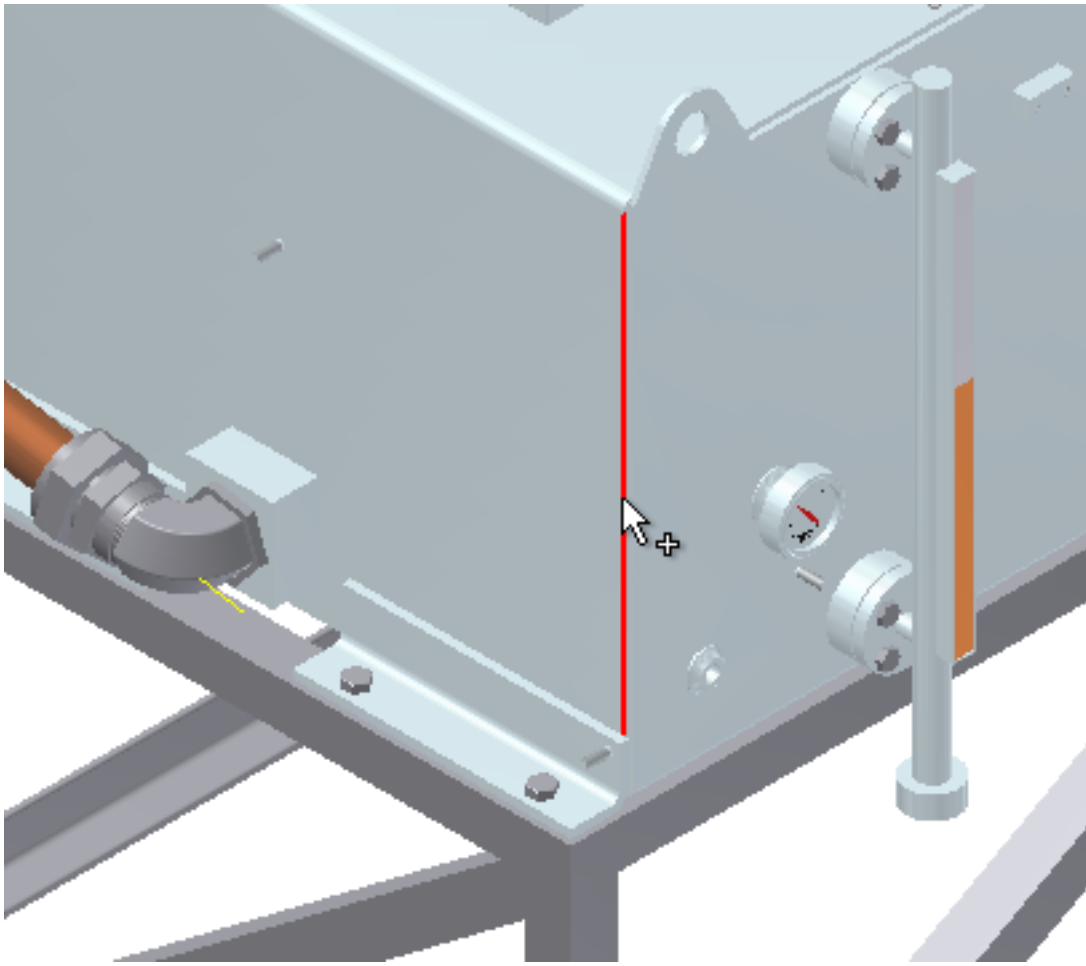
Define Gravity

Next, you specify a direction for gravity. The Gravity command is located at the master Tube & Pipe Runs level.

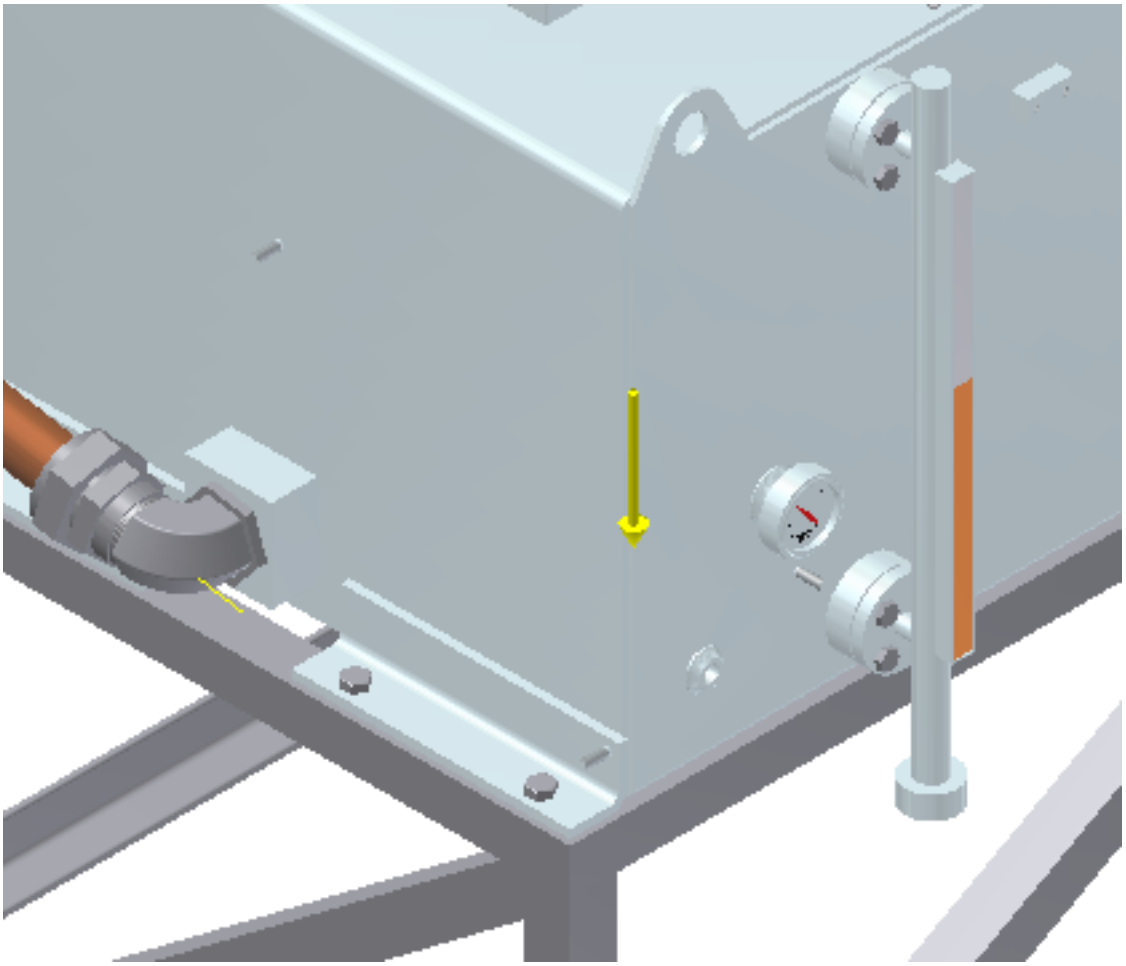
1 Double-click the **Tube & Pipe Runs** node.

2 On the ribbon, click **Tube and Pipe tab ➤ Run panel ➤ Define Gravity**  .

3 Select the edge to set the gravity vector.



4 In the Gravity dialog box, click **Invert Normal**  as needed, so that the vector points down.



5 Click **OK**.

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Return to the Self-draining Run

To continue defining the self-draining line, we must return to the self-draining run level of the assembly.

1 Double-click **SD Run** to reactivate the run.

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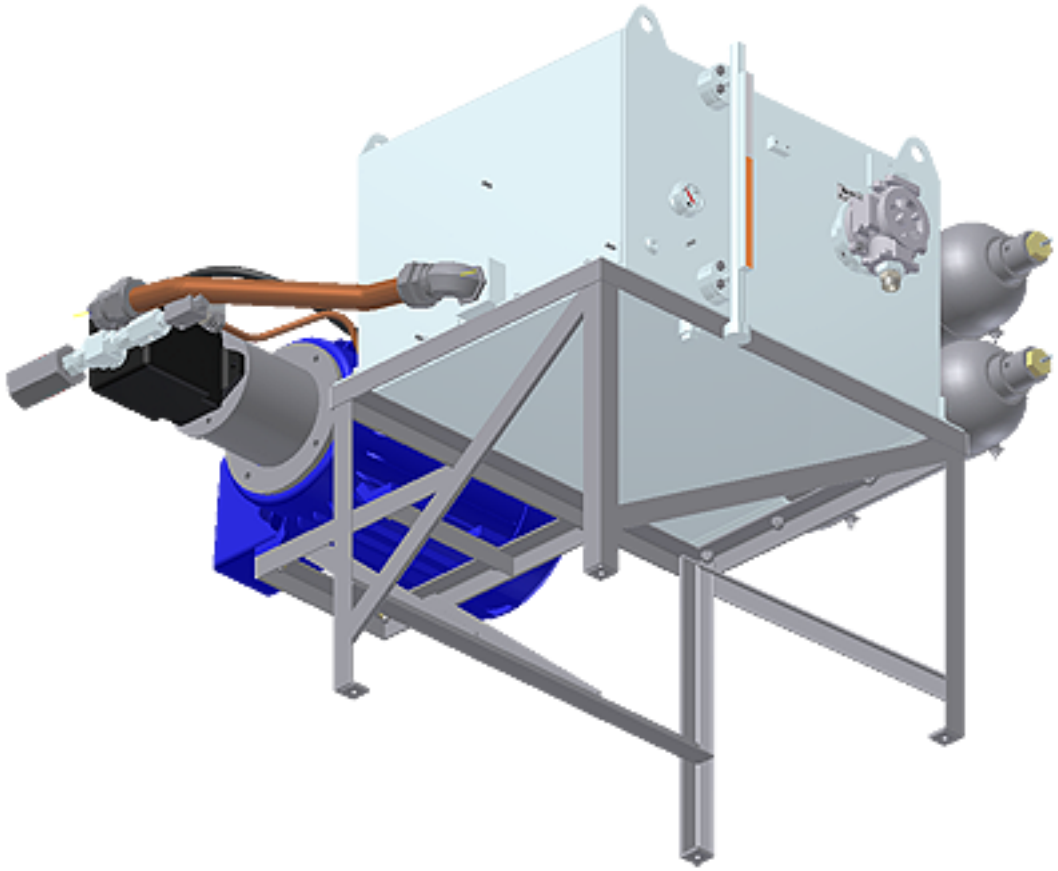
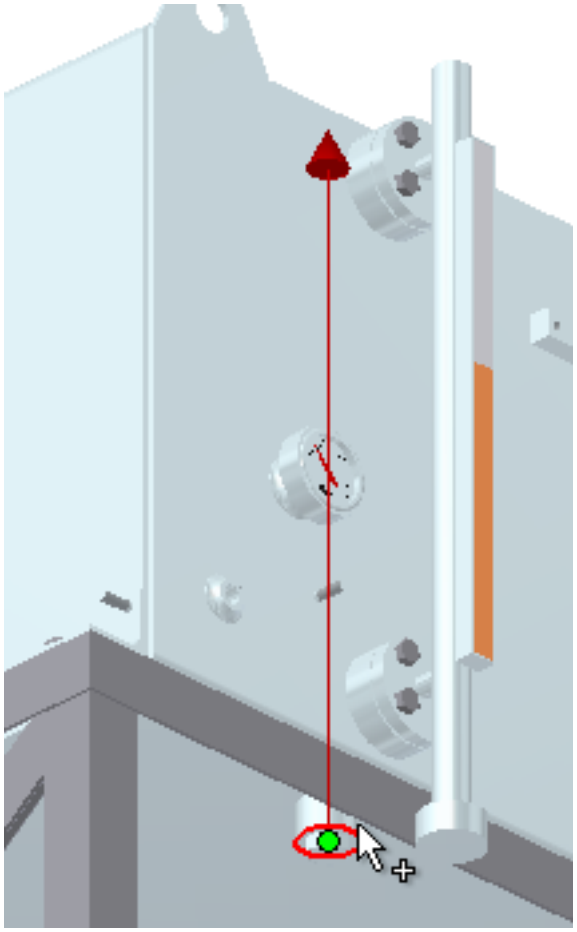
Create the Self-draining Route

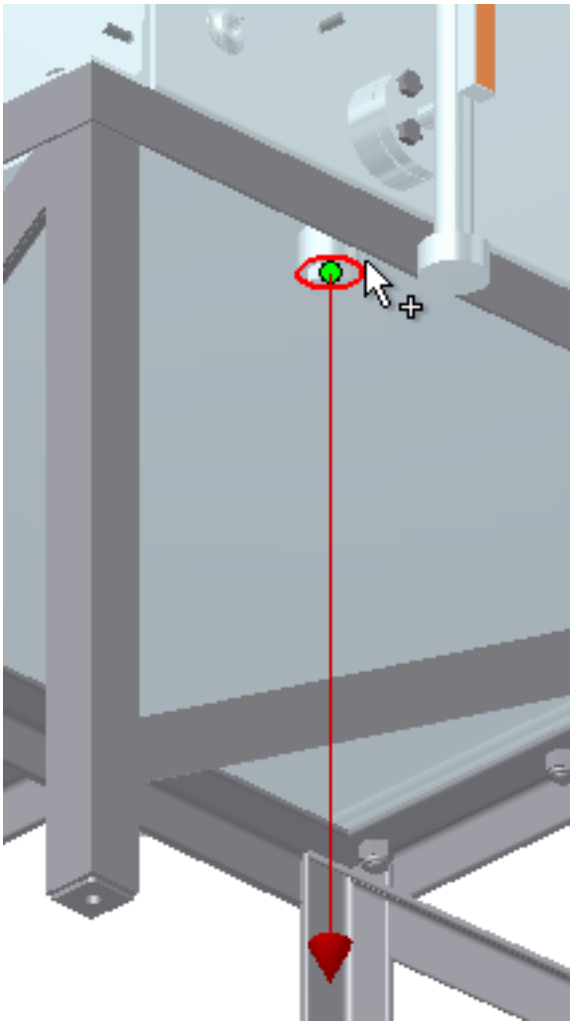
1. On the ribbon, click **Pipe Run tab ➤ Route panel ➤ New Route**  .
2. In the **Route Part File Name** field of the Create Route dialog box, type **SD Route**.
3. Click **OK**.
4. Click **Route tab ➤ Create panel ➤ Route**  .
5. Orbit the model, approximately per the following image.



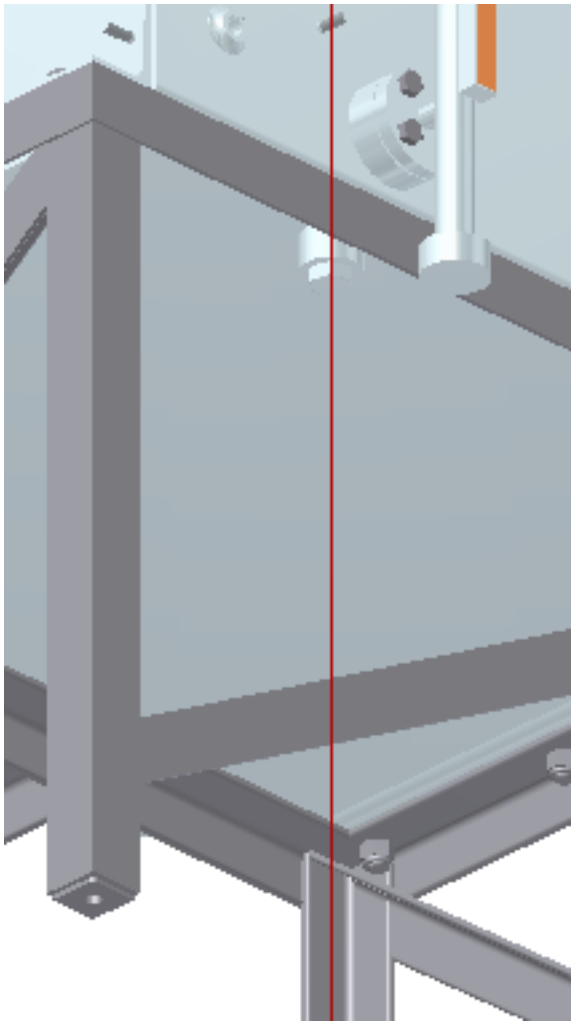
6 Pause the cursor over the circular edge.



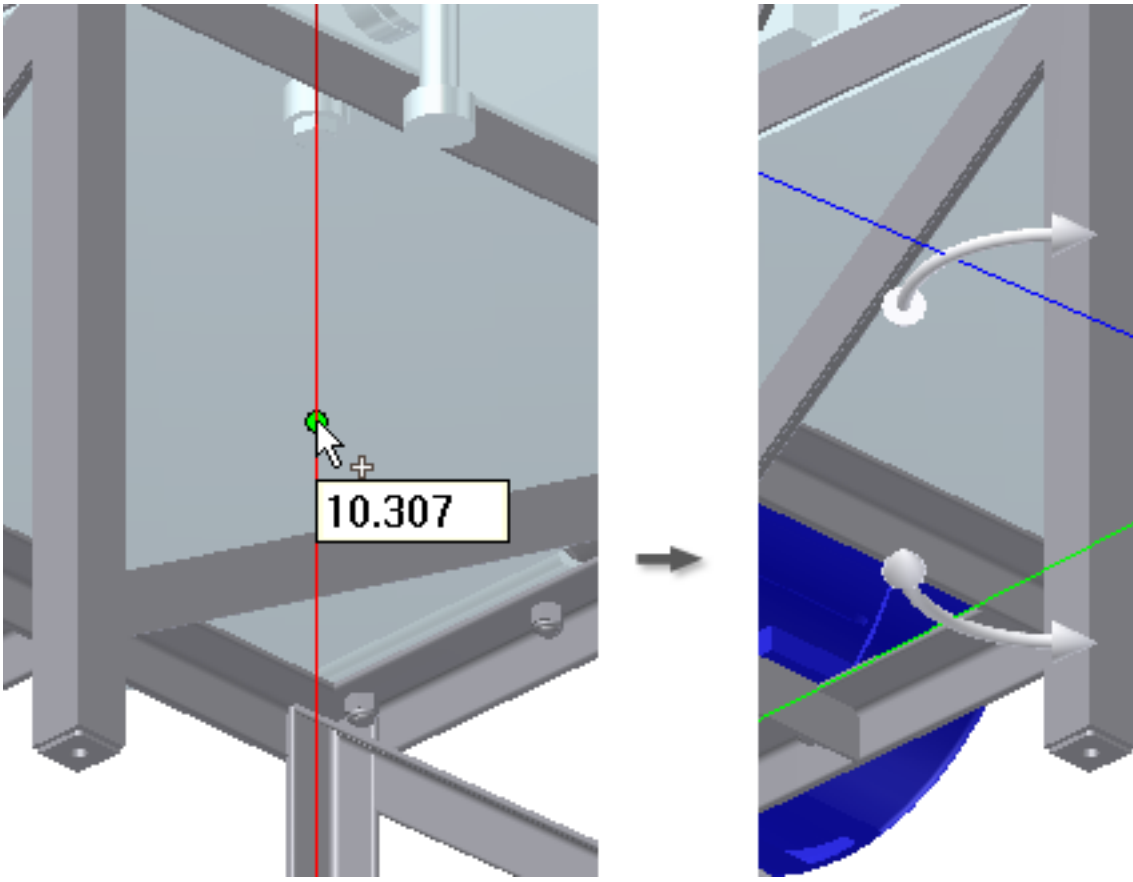
7 Press the spacebar to flip the route vector.



8 Select the circular edge.

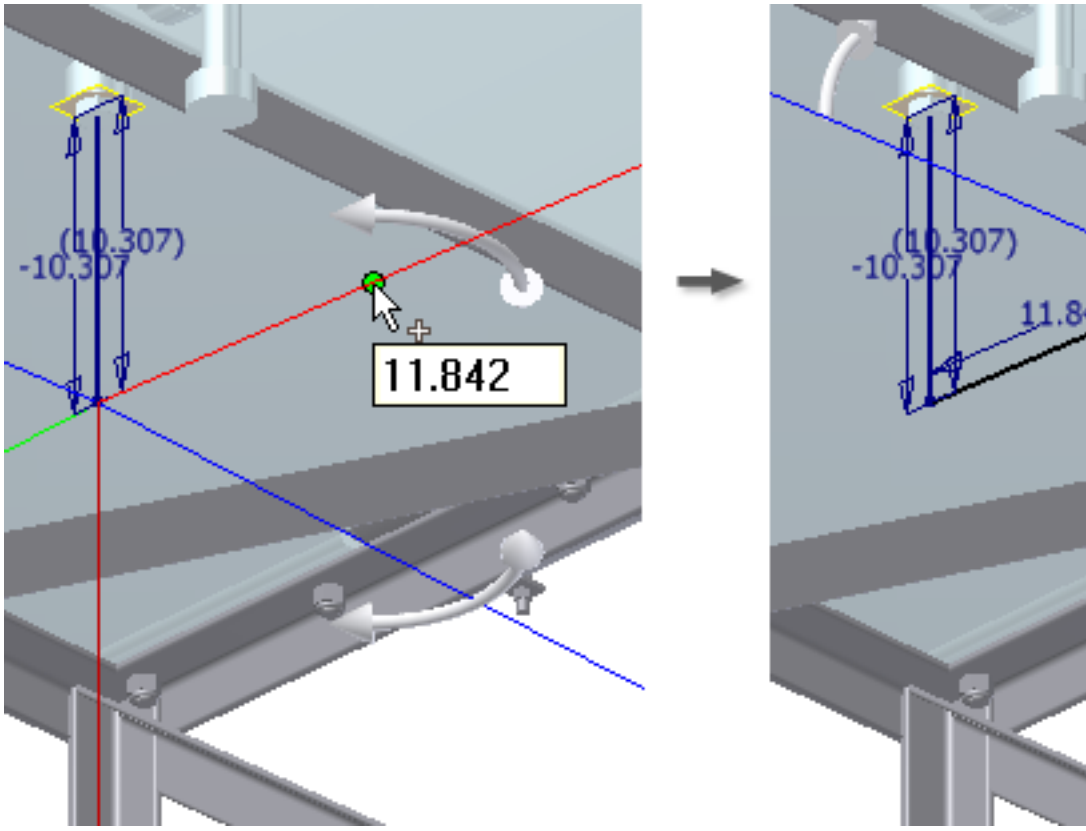


9 Select the route vector, as shown in the following image.



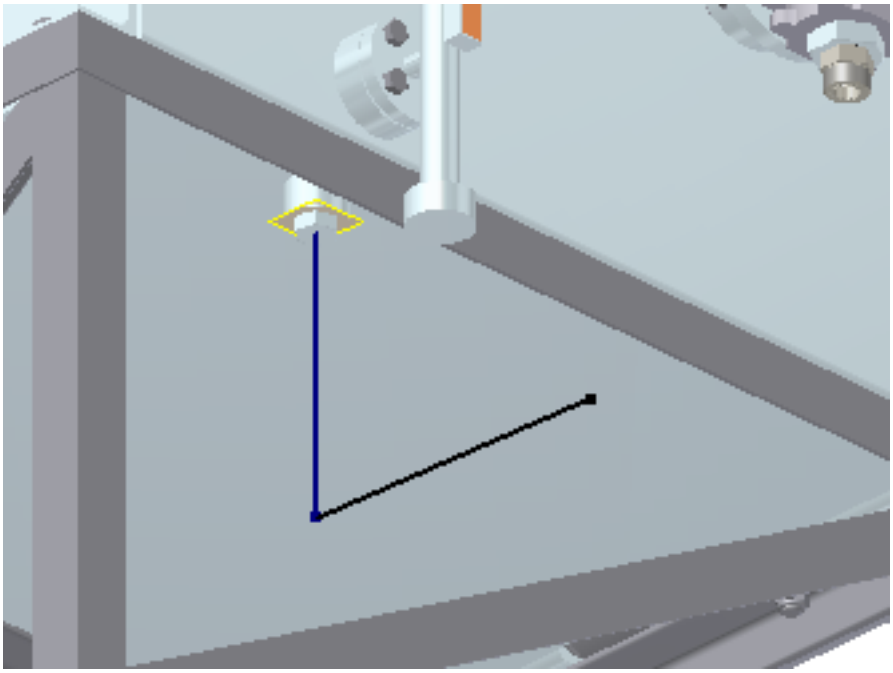
Notice that the green and blue vectors are not at right angles to the red vector. Relative to the red vector, the angle is 88 or 92 degrees, depending on how you measure. The custom fitting you specified in the self-draining style determines this angle.

10 Select the green route vector, as shown.



11 Right-click, and select **Done**.

12 Right-click, and select **Finish Edit** to return to **SD Run**.



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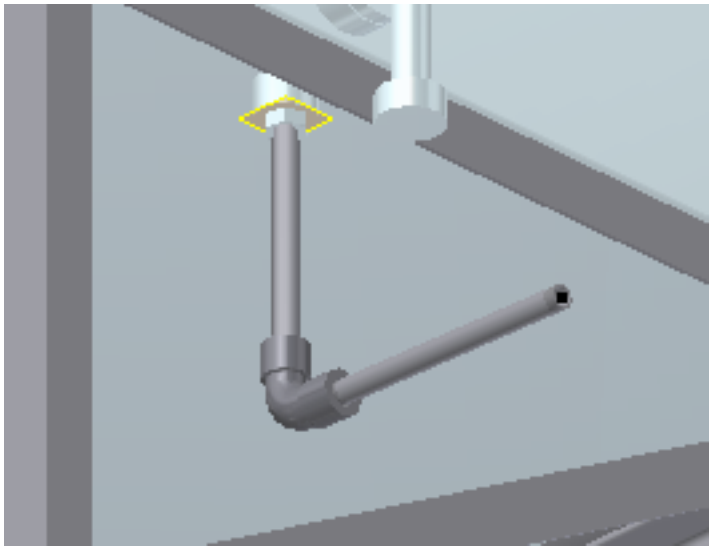
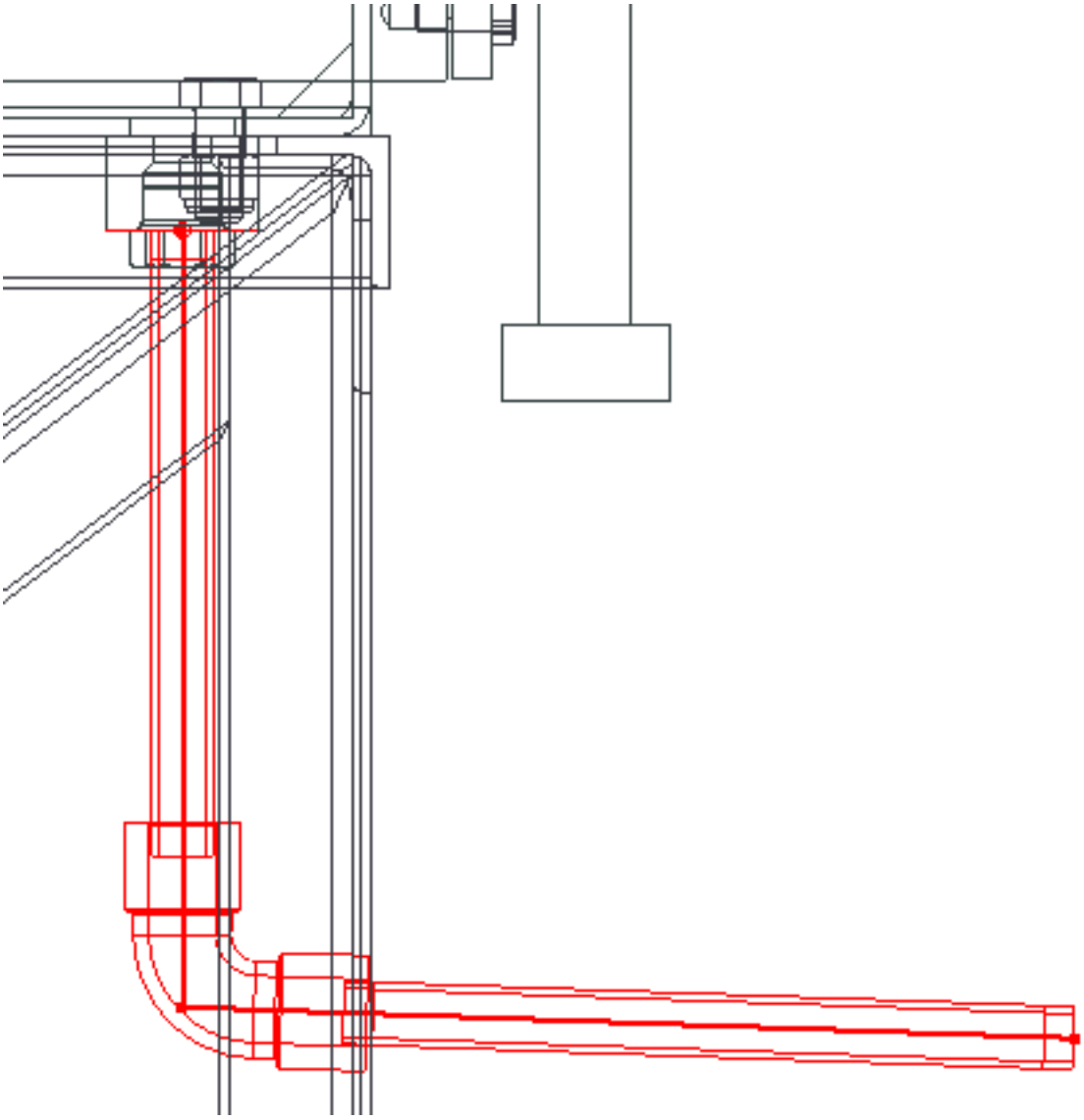
Populate the Route

1 On the ribbon, click **Pipe Run tab ➤ Route panel ➤ Populate Route**  .





The components specified in the self-draining style are added to the route.
Congratulations! You finished the workflow portion of the tutorial.

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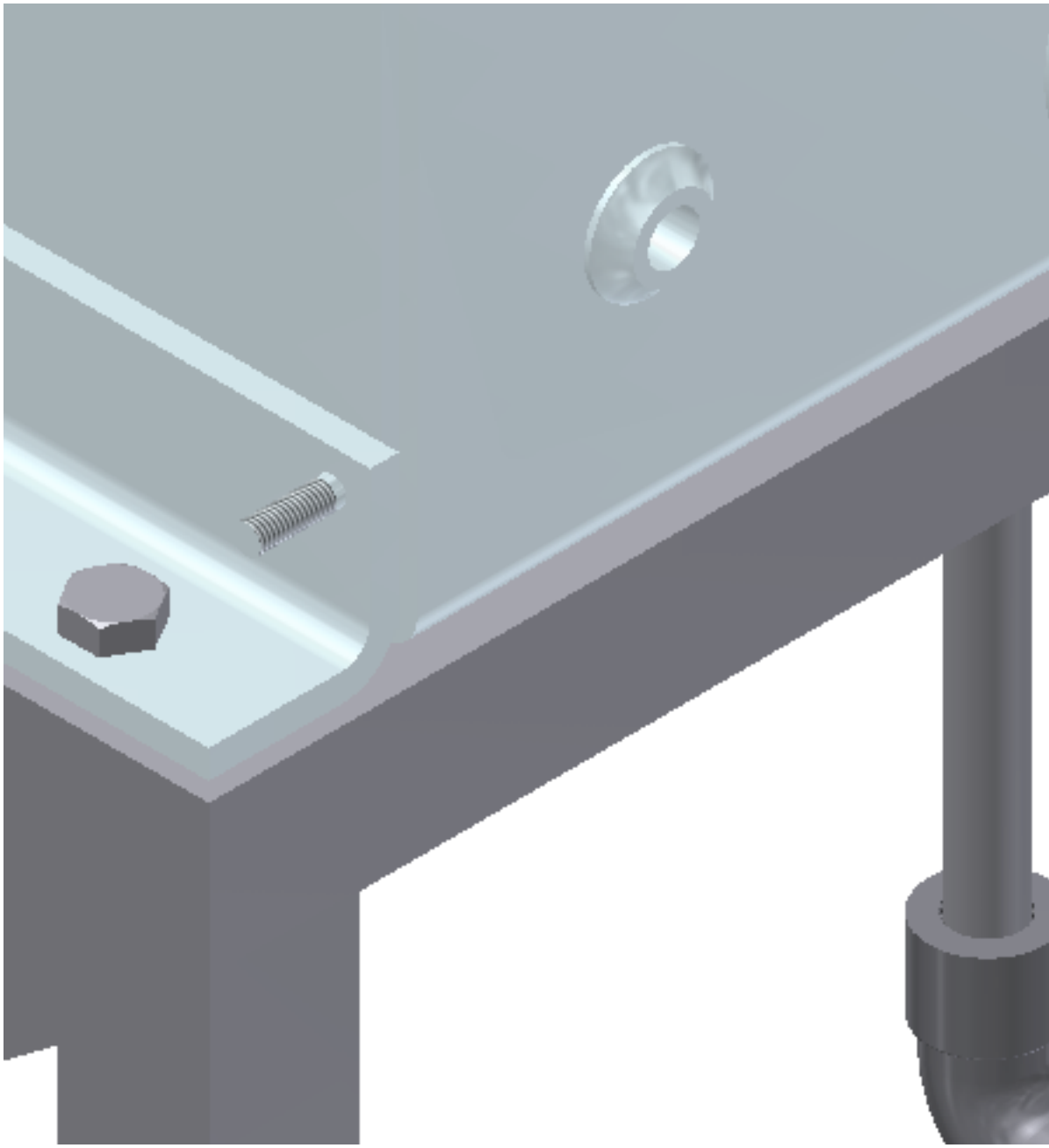
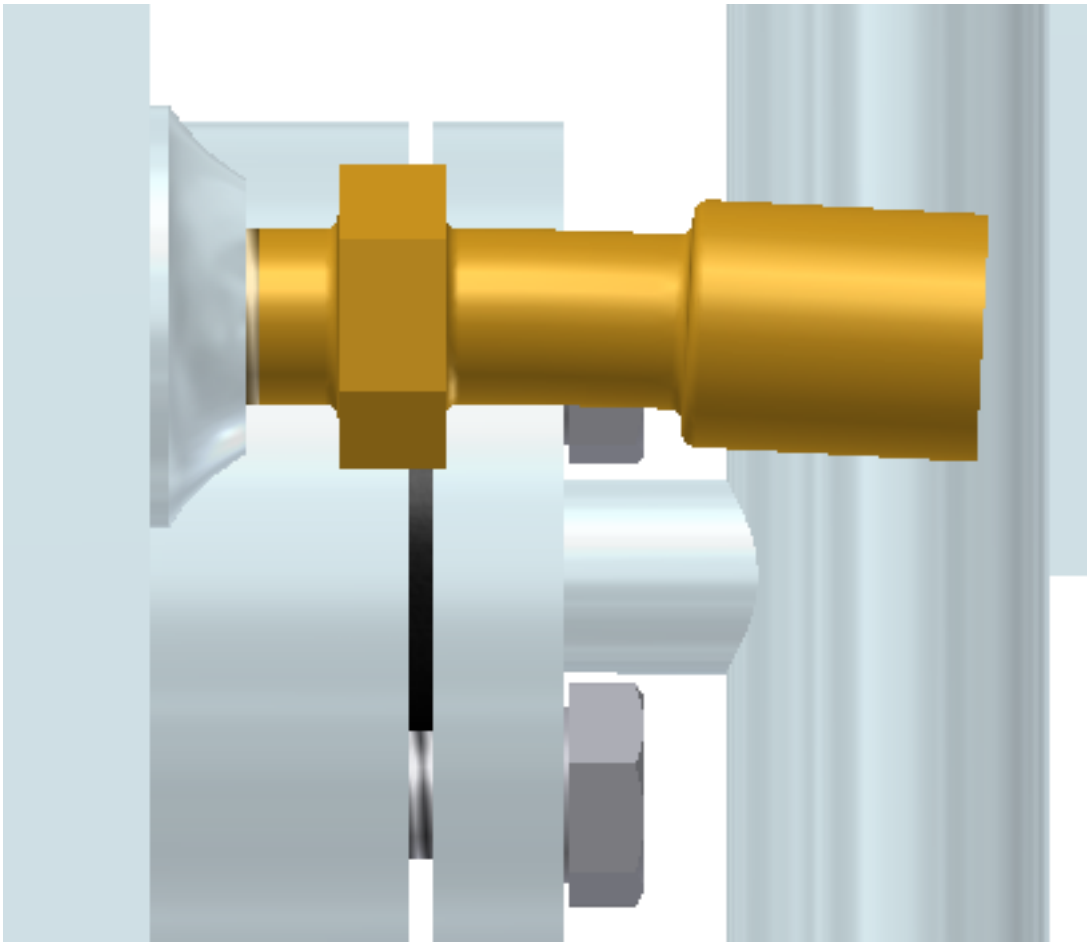
Other Route Starting Locations

Assume that the route must begin from the outlet located on the side of the tank - not from the outlet on the bottom of the tank.



Since this outlet axis is horizontal, the first segment of a self-draining route created directly from this outlet would not be sloped. This positioning would contradict the logic of a self-draining run and the route would be listed as **Sick** in the browser. First you add a fitting that provides a transition from the horizontal outlet axis to the route slope. For example, if the route slope angle is three degrees, first place a custom fitting with a bend of 177 degrees, like the following image.



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Summary

In this tutorial, you learned how to:

- Create and access a read/write library.
- Author and publish a custom fitting to use in a self-draining style.
- Create a self-draining style.
- Create and populate a self-draining route.

Please remember to check the **Self-draining lines** Help topics for further information.

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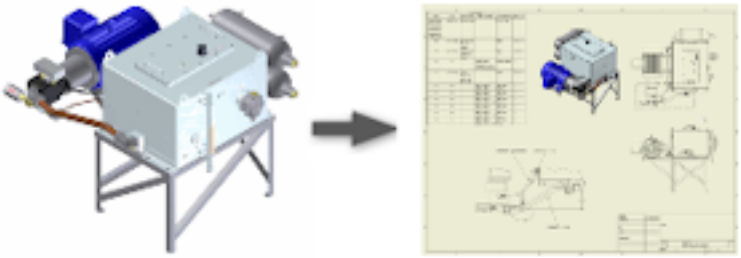
Document Routes and Runs

5

About this tutorial



Document routes and runs.

Category	**Routed Systems**
Time Required	40 minutes
Tutorial Files Used	Accumulator.iam

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

Objectives

- Create a base view for a tube and pipe assembly
- Create a projected view
- Create a detail view
- Add general drawing dimensions
- Add Tube & Pipe-specific properties to the Bill of Materials table
- Add Base QTY and Stock Number to a parts list
- Create sketched symbols with piping style data and annotate drawings

Prerequisites

- Basic knowledge of Tube and Pipe assemblies.
- Basic knowledge of Autodesk Inventor drawings.

Navigation Tips

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

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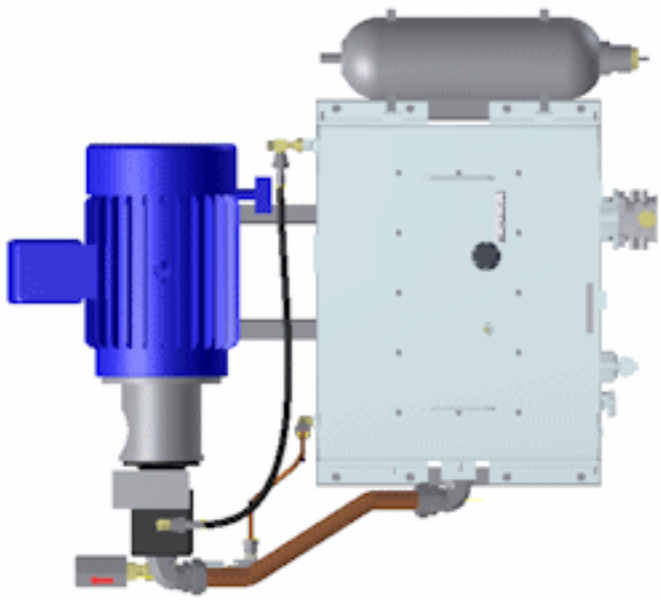
Get Started

To begin, set your active project to **tutorial_files**. Then open a sample assembly.

1. Click  ➤ **Manage** ➤ **Projects**.
2. On the Projects dialog box, double-click the **tutorial_files** project in the projects list to set it as the active project.
3. Click **Save** and **Done** to close the Projects dialog box.
4. Click  ➤ **Open**, and navigate to the Tube and Pipe ➤ Accumulator folder. Select the **Accumulator.iam** file and click Open to open the assembly.
5. Use the View Cube to display the Front view. Then review the tube and pipe components available in the assembly.



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Create a base and projected view

1 To create a drawing, click  ➤ **New**.

2 In the Create New File dialog box, click the Templates folder and select **ANSI (in).idw**, and then click **Create**.

3 Save the drawing document as **MyAccumulator.idw**.

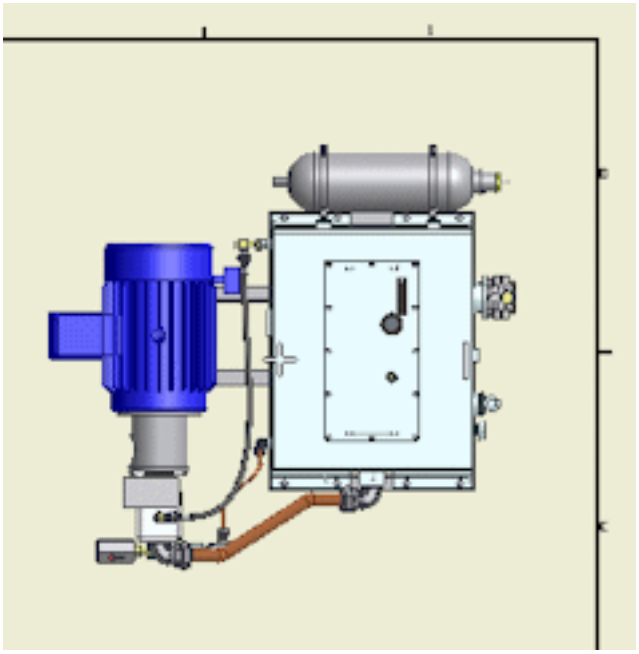
4  On the ribbon, click **Place Views tab** ➤ **Create panel** ➤ **Base**. The Drawing View dialog box displays.

5 Select **Current** in Orientation on the Component tab to use the current view orientation from the assembly file. Keep the default settings for other options. Notice:

- The Accumulator.iam assembly is selected in File.
- The Default view representation and Master level of detail is selected.

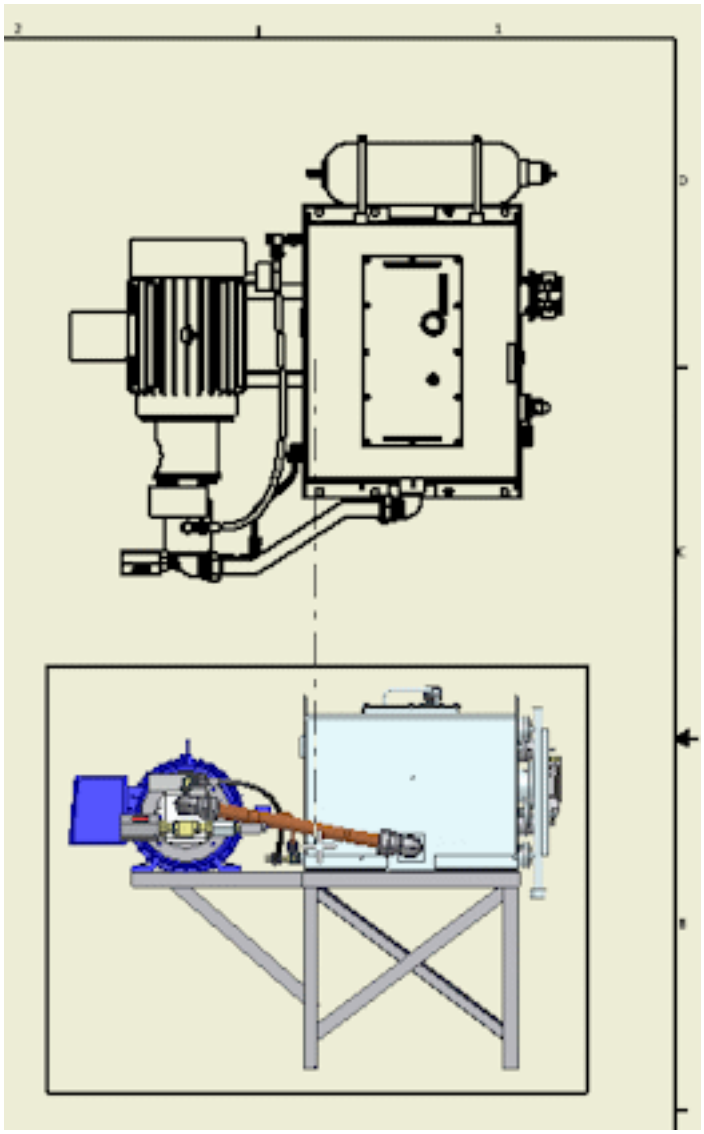
- The view scale is set as 0.08:1.
- The Create Projected Views Immediately after the Base View Creation option is selected.

6 Move the preview to the upper-right quadrant of the drawing sheet. Then click to place the view.

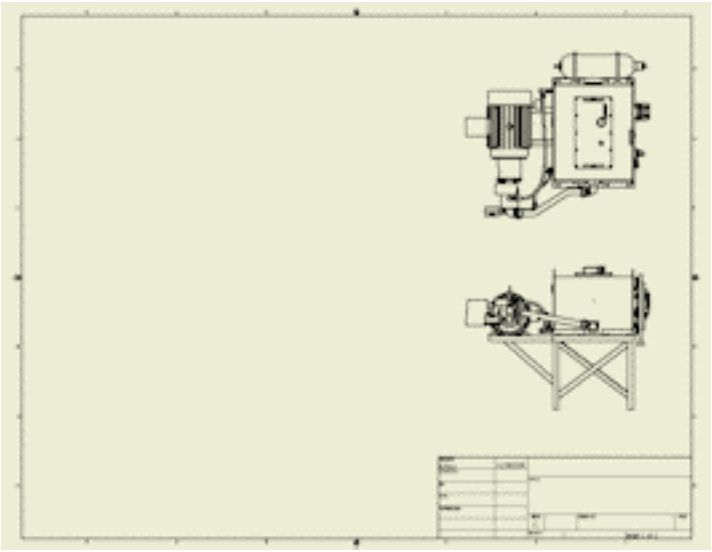


NOTE Do not click OK on the Drawing View dialog box. If you click OK, the view is placed in a random location and the command is finished.

7 Move the cursor down to display a preview of the projected view. Then click to place the projected view.



8 Right-click, and select **Create**. Two views are created in the drawing.



9 Save the drawing document.

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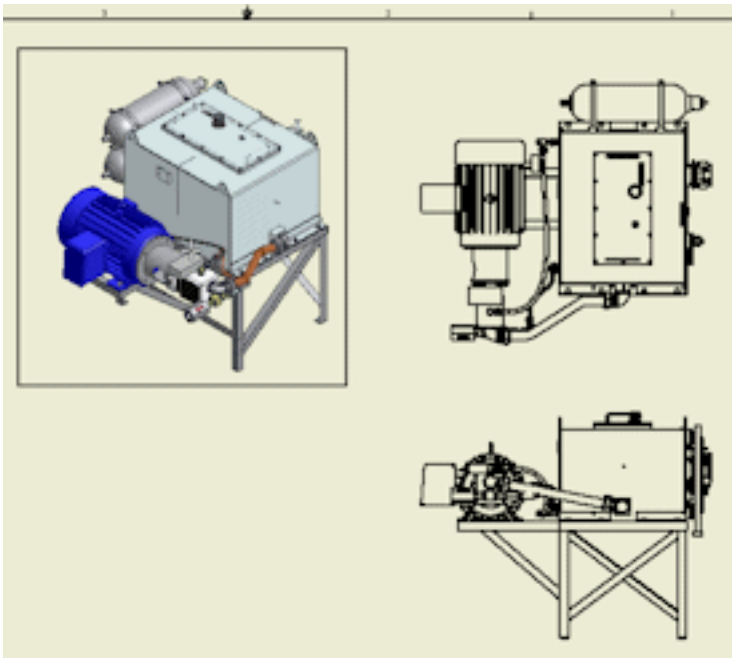
Create an isometric view

1 On the ribbon, click **Place Views tab ➤ Create panel ➤ Projected**.

2 Click VIEW2 (the side view) to use it as a parent view.

3 Move the preview to the upper-middle of the drawing sheet. Then click to place the view.



4 Right-click and select **Create**.

The isometric view is placed.

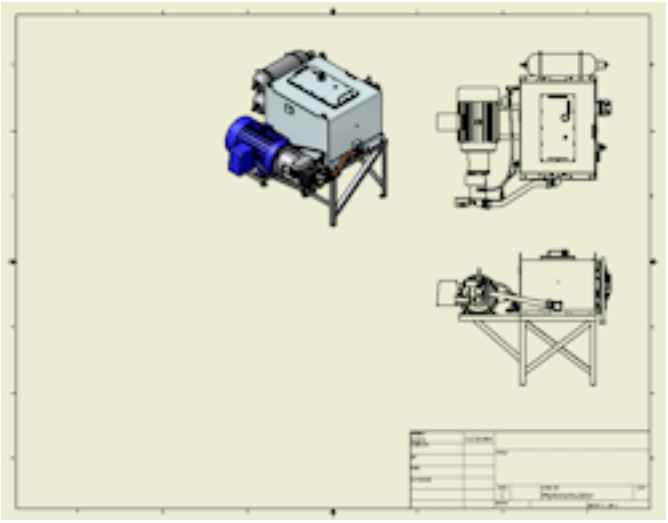
5 To edit the projected view properties, right-click inside the projected view, and select **Edit View**. The Drawing View dialog box displays.

6 On the Component tab of the Drawing View dialog box, select **Shaded**  in Style.

7 Click **OK**.

The projected view updates to display a shaded view of the model.

8 Save the drawing document.



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Add a detail view

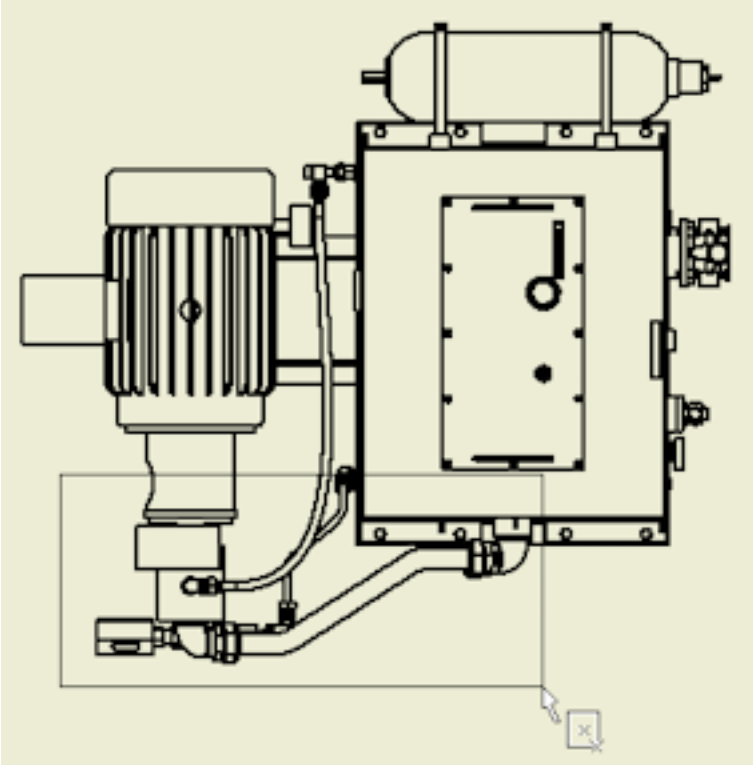
1  On the ribbon, click **Place Views tab ➤ Create panel ➤ Detail**.

2 Click inside VIEW1 to use it as a parent view.

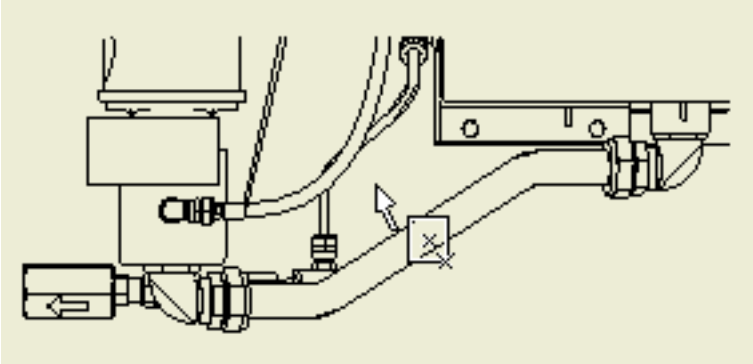
3 In the Detail View dialog box:

- Click **Toggle Label Visibility**  to hide the view label.
- Click **Rectangular**  in Fence Shape.

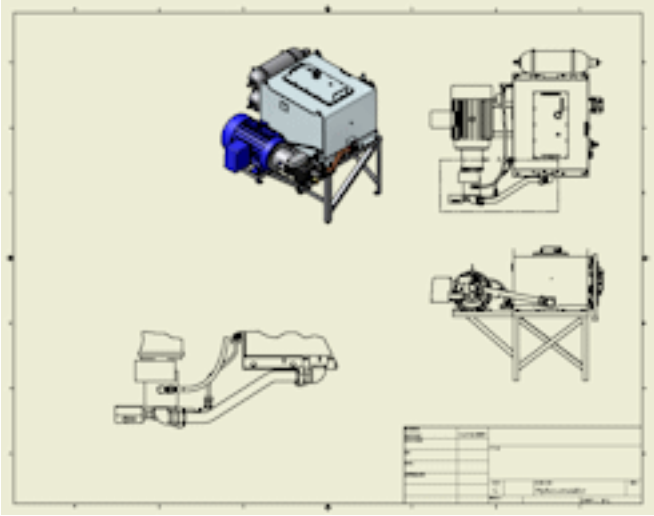
4 Place the cursor in the middle of the desired detail area. Then click, and drag the cursor to include a part of the model as shown in the following image.



5 Click to set the portion of the view to include in the detail view. The detail view preview is displayed and you can drag it.



6 Move the detail view to the desired position, and then click to place the view.



TIP Adjust the size and position of the detail view fence after you place the detail view. Move the cursor over the detail fence in VIEW1, and then drag the fence center or a grip point to the desired position.

7 Save the drawing document.

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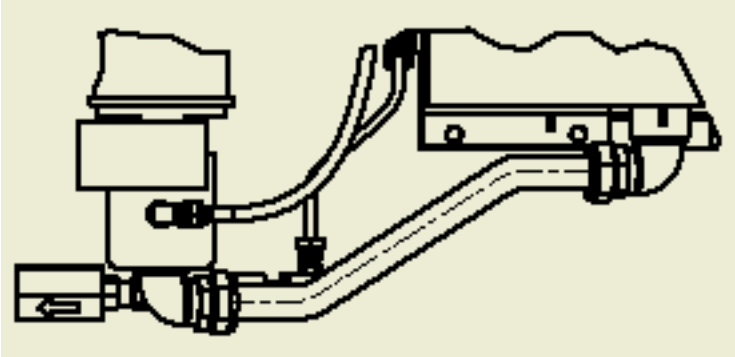
Recover route centerlines

1 In the Model browser, under Sheet 1, navigate to **VIEW1:Accumulator.iam➤A:Accumulator.iam➤Accumulator.iam➤Tube & Pipe Runs**.

2 Expand **Tube & Pipe Runs**, right-click **Pipe Run 1**, and select **Include Route Centerlines**.

The route centerline is recovered in the detail view.



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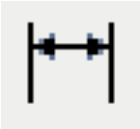
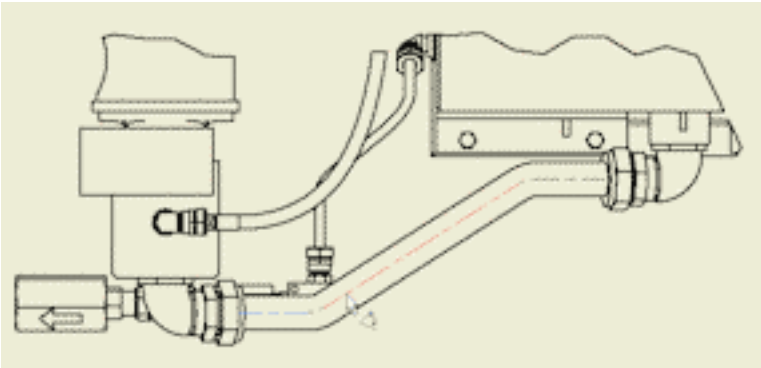
Add general dimensions

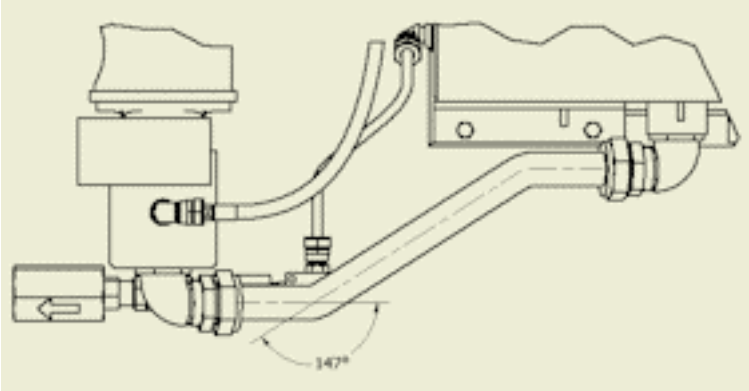


1 On the ribbon, click **Annotate tab ➤ Dimension panel ➤ Dimension**.

2 Zoom in the detail view. Then click the first segment and the second segment of the centerline as shown in the image.

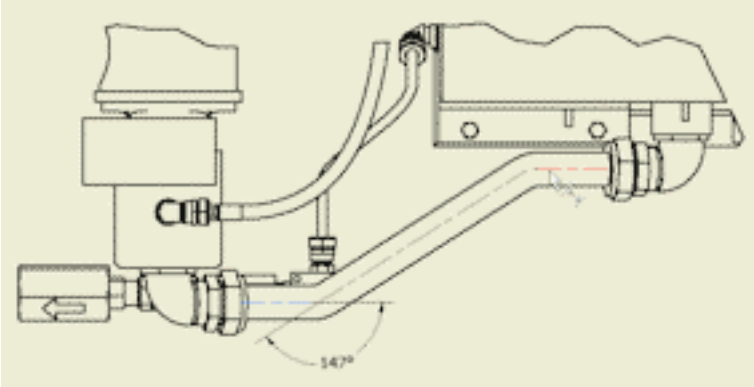


3 Drag the angular dimension to the desired position and click to place it.

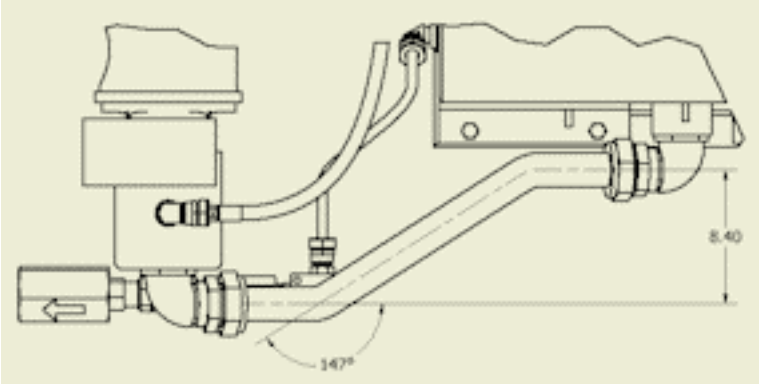


4 In the Edit Dimension dialog box, clear the **Edit dimension when created** box and click OK.

5 Click the first and the last segment of the centerline as shown in the image.



6 Drag the linear dimension to the desired position and click to place it.



7 Right-click and select **Done** to exit the command.

8 Save the drawing document.

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Enable the Parts Only view

1 Right-click **VIEW1:Accumulator.iam ➤ Accumulator.iam** in the Model browser.

2 Select **Bill of Materials** on the menu.

The Bill of Materials dialog box displays. By default, the Structured view is enabled and the Parts Only view is disabled.

3 In the Bill of Materials dialog box, click the **Parts Only (Disabled)** tab.

4  On the toolbar, click the arrow next to **View Options**, and then select **Enable BOM View**.

All components in the tube and pipe assembly are shown in a flat list. Subassemblies are not displayed.

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Add Tube and Pipe-specific properties to the BOM

1. On the Model Data tab of the Bill of Materials dialog box, verify that the Stock Number column is included in the table.
2. Add the Base QTY column.

    Click **Choose Columns** on the toolbar. Drag the **Base QTY** column from the Customization list to the table.

   NOTE You can also remove unwanted columns by dragging them to the Customization list.
3. Close the Customization list.
4. Review data available in the BOM table. Optionally, edit the BOM data.
5. Click **Done**.

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Create a parts list

1.  On the ribbon, click **Annotate tab ➤ Table panel ➤ Parts List**.
2. In the Parts List dialog box:
   - Select the parts list source. Click the base view (VIEW1) in the drawing or in the Model browser. Notice that the **Accumulator.iam** file is selected as the source file.
   - Select **Parts Only** in BOM Settings.
   - Click **OK** to close the Parts List dialog box.
3. A preview of the parts list displays. Place the parts list at the top-left corner of the drawing.

   The parts list table is too long and does not fit into the drawing. In addition, tube and pipe-specific columns are not included in the parts list. You must filter the parts list and add parts list columns.

4 To edit the parts list, right-click the parts list in the drawing and select **Edit Parts List**.

5  In the Parts List dialog box, click **Filter Settings**.

6 In the Filter Settings dialog box, define a parts list filter:

- Select **Standard Content** from the Define Filter Item list.
- Select **Standard Content Only**.
-  Click **Add Filter**. The Standard Content Only filter is added to the list of filters to apply.
- Click **OK** to close the Filter Settings dialog box.

7  In the Parts List dialog box, click **Column Chooser**.

The Parts List Column Chooser dialog box displays. It shows a list of available properties and properties included in the parts list (Selected Properties).

NOTE The parts list style determines properties displayed by default in a parts list. The current parts list style does not include the STOCK NUMBER and BASE QTY property.

8 In the Parts List Column Chooser dialog box:

- Select **BASE QTY** in Available Properties, and click **Add**. The BASE QTY property is moved to the Selected Properties list.
- Select **STOCK NUMBER** in Available Properties, and click **Add**. The STOCK NUMBER property is moved to the Selected Properties list.
- Click **OK** to close the Parts List Column Chooser dialog box.

  Both BASE QTY and STOCK NUMBER columns are displayed in the Parts List dialog box.

9  In the Parts List dialog box, click **Group Settings**.

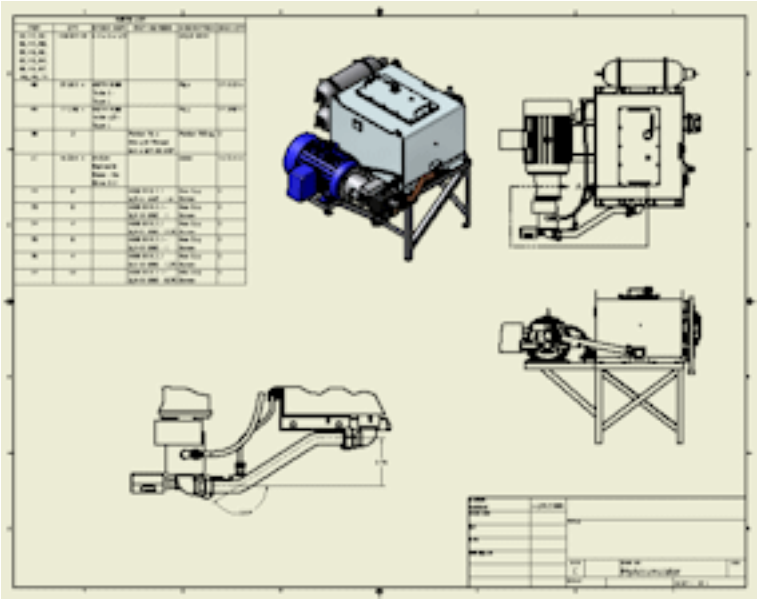
10 In the Group Settings dialog box, define part grouping based on the Stock Number property:

- Select the **Group** option.
- Set **STOCK NUMBER** as the **First Key**.
- Clear the **Display Group Participants** box.
- Click **OK** to close the Group Settings dialog box.

11 In the Parts List dialog box, click **OK**.

The parts list automatically updates.

12 Drag the parts list lines to refine the parts list appearance in the drawing.



13 Save the drawing document.

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Create sketched symbols with piping style data

1 Expand **Drawing Resources** in the Model browser.

2 Right-click **Sketched Symbols** and select **Define New Symbol**.

The drawing sketch environment is activated.

3 On the ribbon, click **Sketch** tab ➤ **Draw panel** ➤ **Text**, and then click a location in the graphic window.

The Format Text dialog box is displayed.

4 Review and keep the default text style. Then select **_Piping Style** from the Type list.

The Property list is populated with properties related to tube and pipe styles.

5 Select **Pipe Type** from the Property list and click **Add Text Parameter**  . The Pipe Type property is added to the text field.

6 Select **Name** from the Property list, and click **Add Text Parameter** to add the property to the text field.

7 Add a hyphen (surrounded with two spaces) between the properties in the text field to separate the properties.

8 Accept the other defaults and click **OK** to close the Format text dialog box.

A string with two selected piping style properties is previewed in the sketch.

9 Right-click in the graphics window and select **Done**.

10 Right-click again and select **Save Sketched Symbol**. Enter **My Symbol** in the Sketched Symbol dialog box, and click **Save**.

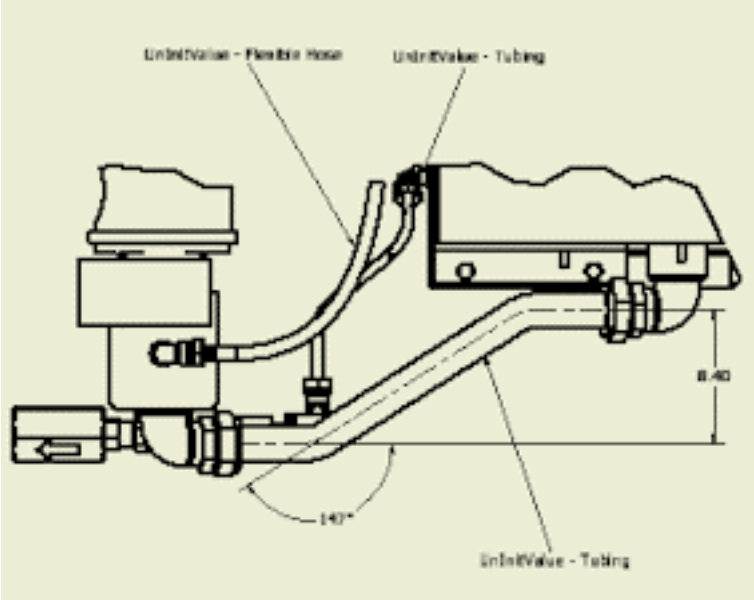
The new sketched symbol is added to the Sketched Symbol list in Drawing Resources.

11  To place the symbol, click **Annotate tab ➤ Symbols panel ➤ User**.

12 Select **My Symbol** in the Symbols dialog box, and then click **OK**.

13 Click an edge of a tube or pipe in the detail view, and click a point for the symbol leader. Then right-click and select **Continue**. A sketched symbol is placed in the drawing.

14 Repeat the previous step for other tube or pipe parts. When finished, right-click and select **Done**.



15 Save the drawing document.

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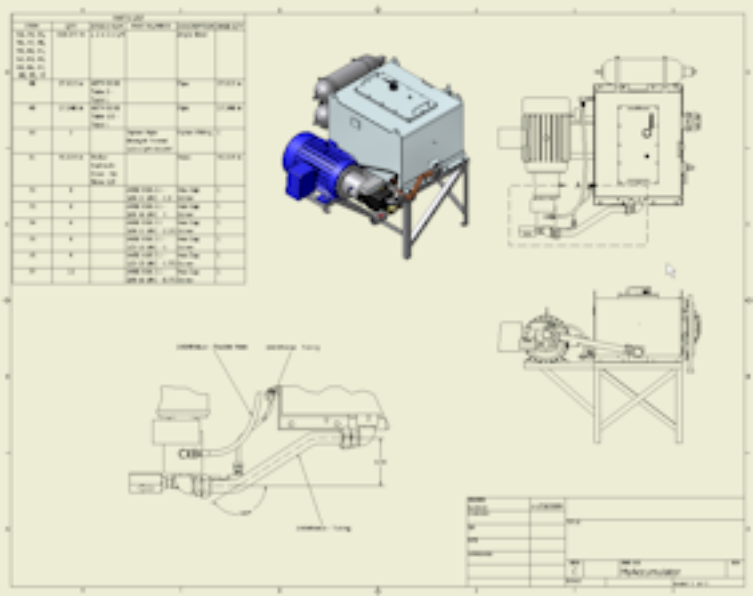
Summary



In this tutorial, you learned how to:

- Create a base view, projected view, and detail view for a tube and pipe assembly.
- Recover a route centerline and add dimensions to it.
- Add Tube and Pipe-specific properties to the Bill of Materials.
- Create and customize a parts list.
- Create a sketched symbol with piping style data.

What Next? - As a next step, complete other Tube and Pipe tutorials or Drawing tutorials.

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Cable and Harness 6

About this tutorial

Create and manipulate three-dimensional wire harnesses in an assembly.

Category	**Routed Systems**
Time Required	60 minutes
Tutorial File Used	Enclosure_Assembly.iam ribbon_cable_connector.ipt

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

Take this working tour of the basics in Cable and Harness in Autodesk Inventor. Then proceed to the successive tutorials to go deeper into the details of the functionality.

Objectives

- Use various Cable and Harness commands on the ribbon.
- Create and work with :
  - Pins on a connector
  - Wires between connectors
  - Ribbon cables and folds
  - Segments

- Route wires.
- Place additional connectors.

Prerequisites

- Know how to set the active project, navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic “Fundamentals” and the New User tutorials for further information.

Navigation Tips

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

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Explore Cable and Harness

Open the tutorial file.

1. Click  ➤ **Open**.
2. Set the Project File to **tutorial files.ipj**.
   - For Microsoft Windows XP, the default file location is *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness*
   - For Microsoft Windows Vista, the default file location is *Public\Public Documents\Inventor <version>\Tutorial Files\Cable & Harness*
3. Open the tutorial file **Enclosure_Assembly.iam**.
4. Click  ➤ **Save As**. In the Save as dialog box, save the file with the name **Enclosure_Assembly_basics.iam**. You need the unchanged source file for the next tutorial.

NOTE In this tutorial, you save the changes you make to your tutorial files. To replace the modified files with the original source files, reinstall the source files from your installation software and copy them with the appropriate name. Alternatively, back up the source **Tutorial Files** directory to another location and access the back-up files as needed.

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Add Pins to Connector

Add pins to a connector already present in **Enclosure_Assembly_basics.iam**.

1. In the assembly browser, double-click the part named **360124** to open the part for edit.
2. In the model browser, delete work points **1** through **9**. You re-create these points in the next steps.
3. On the **Model** tab, Harness panel, click **Place Pin**. 
4. Rotate and zoom the view to see the part as shown. Turn **Visibility** off for any parts that obstruct your view.
5. In the graphics window, move the cursor over the part geometry to highlight valid points for your selection. Highlight the circular edge shown, and then click to select the center point.

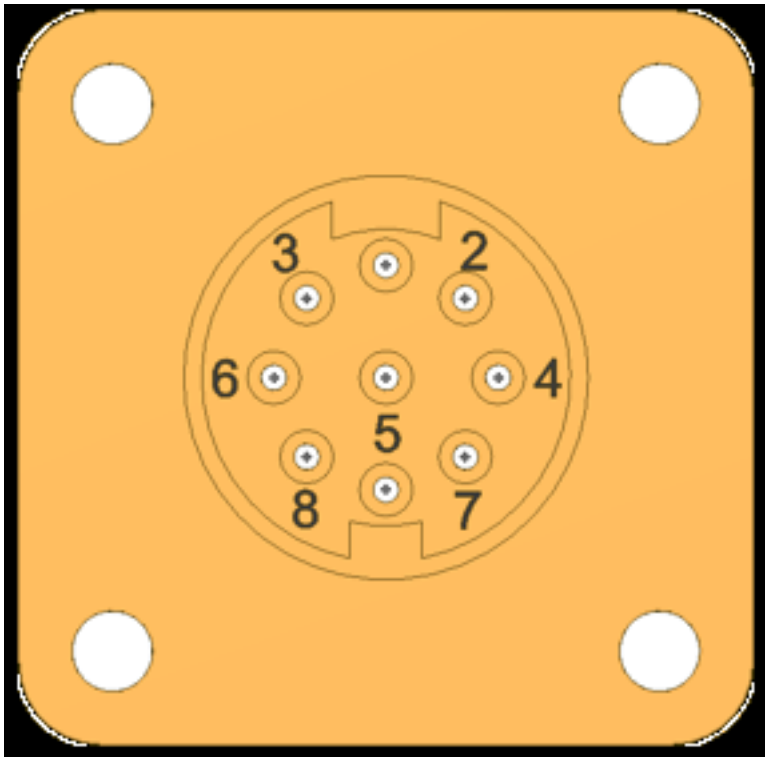
6 On the Place Pin dialog box, use the default pin name of **1**.

7 Click **Harness Properties**  to display the Connector Pin Properties dialog box.

8 Click the **Custom** tab to view where common properties are added. Without changing the properties, click **Cancel**.

9 Click the green check mark. The work point that represents pin 1 appears.

10 Repeat the necessary steps to create pins 2 through 9.



11 Right-click, and select **Done** to exit the **Place Pin** command.

Save your file.

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Add a RefDes Placeholder to the Connector

A reference designator, or RefDes placeholder, is an identifier that maps the part to the schematic design. Typically, you add a placeholder such as U?. A number that represents the associated occurrence in the assembly replaces the?. For example, if you have three occurrences of a connector in your

assembly, each occurrence must have a unique identifier such as U1, U2, and U3.

1. With nothing selected, on the **Model** tab, **Harness** section, select **Harness Properties**. Alternatively, in the browser, right-click the part and select **Harness Properties**.
2. In the Part properties dialog box, **General** tab, enter a **reference designator** (RefDes). In this case, enter the placeholder `U?`.
3. Click **OK**, and then click **OK** on the message box.
4. Click **OK** to accept the RefDes and close the dialog box.
5. Click **Return** to finish the edit. Alternatively, right-click and select **Finish Edit**.
6. Save your file. Click **OK** in the Save message dialog box to save the changes to the appropriate files.

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Configure Additional Connectors

Use the skills you just learned to add pins and RefDes placeholders to two other parts.

1. Edit the part **360575:1**, and add pins at each hole. Name the pins to match their number label. Assign the RefDes placeholder `U?`.

2 In the browser, expand the **PCB** subassembly to locate and edit part **LTP**. Add the pins as shown. Pin 1 is the pin closest to part 360124. Assign the RefDes placeholder `J?`.

3 In the browser, edit part **360575:2** and notice how this occurrence inherits the pins set in 360575:1. After you pin the part once, all occurrences inherit the pins. **Return** from the edit mode, and save your file.

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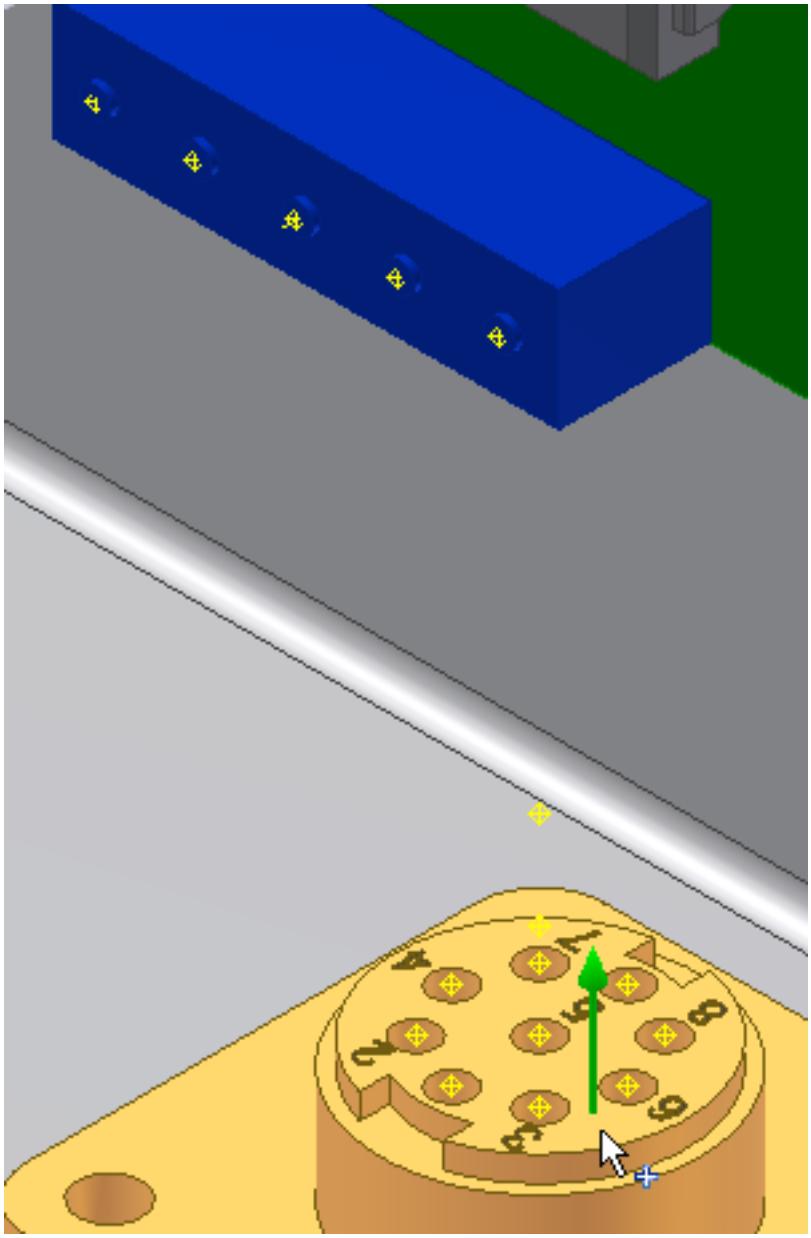
Author Connectors

Use connector authoring to customize your connectors with settings specific to Cable and Harness. Connectors must be authored to be recognized for use with ribbon cables. Also, the natural curvature display option functions only between authored connectors.

1. If it is not already open, navigate to and open **Enclosure_Assembly_basics.iam**.
2. In the assembly browser, double-click part **360124**.
3. On the **Manage** tab, **Author** panel, click **Connector Authoring**.

4. You can define various characteristics of the connector, but for this tutorial you have to define only the outward direction of the connector. Click the face of **part 360124** as shown. Ensure that the **Outward Direction** arrow is as shown.



5 Click **OK** to accept the changes.

6 On the Authoring Result dialog box, click **OK**. Click **Return** to finish editing the part.

7 Perform the same steps to author connectors from both the **LTP** and **360575** parts. In each case, verify the **Outward Direction** arrow points into the interior of the Enclosure Assembly.

8 Save your file and the associated parts.

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Create a Harness Assembly

Create a harness assembly. Later, harness objects such as wires and cables are added to the assembly.

1 If it is not already open, navigate to and open your tutorial file.

2 Click **Assemble tab ➤ Begin panel ➤ Cable and Harness** 
.

3 On the Create Harness dialog box, enter **Harness Assembly1.iam** as the **Harness Assembly File Name**, and click **OK**. The harness assembly is added to the browser, and the Cable and Harness environment is activated. You are in edit mode for Harness Assembly1.

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Assign Reference Designators to Each Occurrence

Now that the harness assembly is added, set a reference designator for each electrical part occurrence. The reference designators are specific to the harness assembly.

1 In the browser, right-click part **360124** and select **Harness Properties**.

2 On the Part Properties dialog box, set a RefDes of `U3` for the occurrence, and click **OK**.

3 Set a RefDes of `U7` for part 360575:1.

4 Set a RefDes of `U8` for part 360575:2.

5 In the browser, expand the **PCB** subassembly to locate part **LTP**, and set a RefDes of `J12`.

NOTE The unique values override the generic RefDes placeholders at the occurrence level. The designators are specific to the harness assembly. If you create additional harness assemblies, you can follow the same steps, but use different designators.

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Create Harness Objects

Create wires and cables in the harness assembly.

Before you begin creating harness objects, such as wires and cables, you can set several default configuration options for forward creation. The defaults are set on the Harness Settings dialog box.

1 Activate **Harness Assembly1**, if not already active.

2 Right-click **Harness Assembly1**, and select **Harness Settings**.

3 On the **General** tab, examine the settings of interest and ensure the **Use Surfaces for All Segments and Wires** is selected (it is selected by default). Choose this setting for faster processing times while creating and editing. To represent segments and wires with solids, clear the check box.

4 On the **Wires/Cables** tab, under **Natural Curvature**, select **Without Natural Curvature**  to display the wires as straight lines for the harness assembly.

5 Click the other tabs and examine the various settings.

6 Click **OK**.

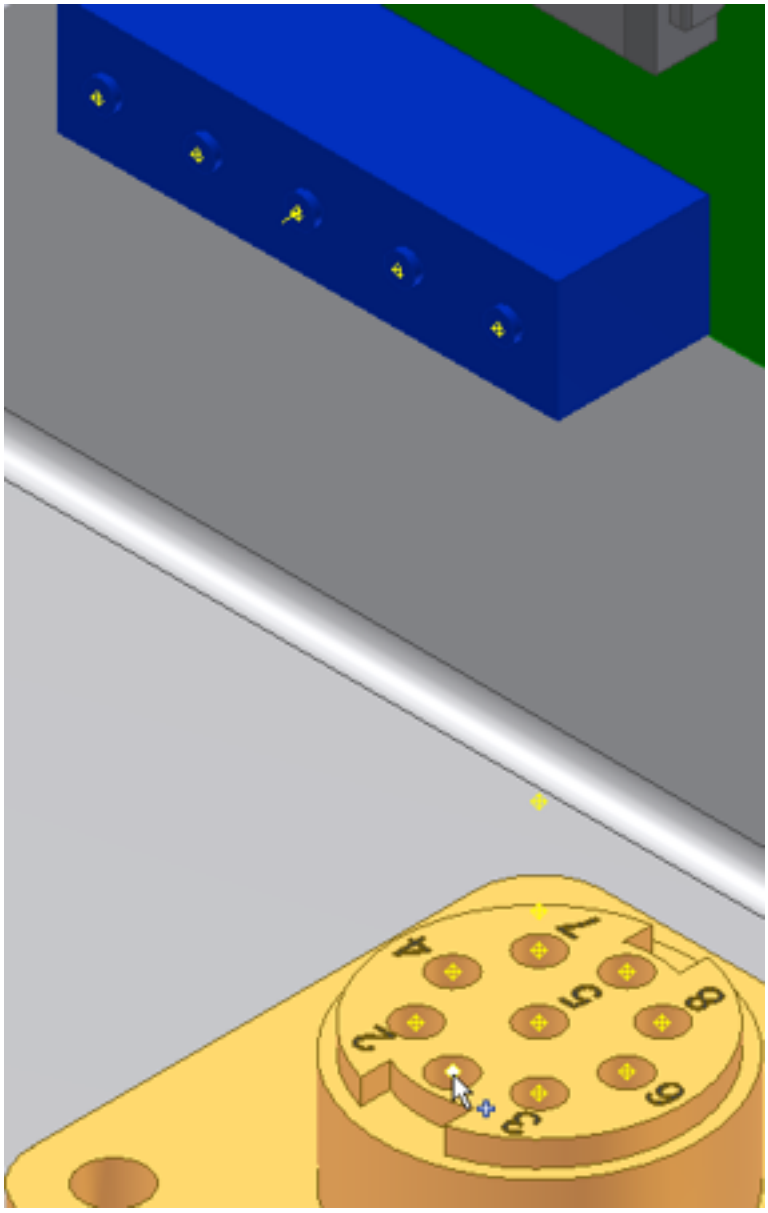
Manually insert wires and cables.

1 On the **Cable and Harness** tab, **Create** panel, click **Create Wire**.

2 On the Create Wire dialog box, accept the default wire ID (**Wire1**). The Wire ID is **Wire** plus an incremental number (the numbers start at 1).

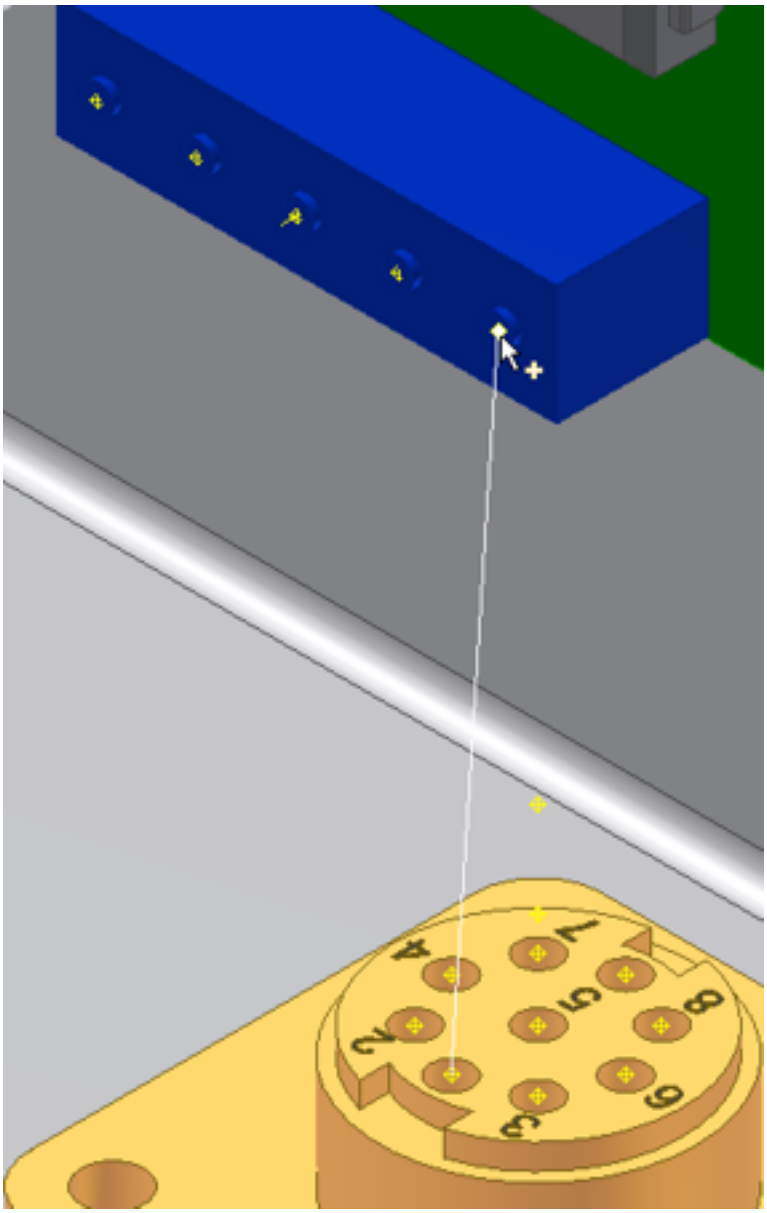
3 From the list of wires, select the wire to insert:
   - For category, select **Generic**.
   - For name, select **22AWG-GRN**.

4 To start the wire, select **Pin 1**. Move the cursor over the pins on part 360124. When the tooltip displays as **U3 Pin 1**, click the work point. If you have Dynamic Prompts turned on, the tooltip provides additional information.

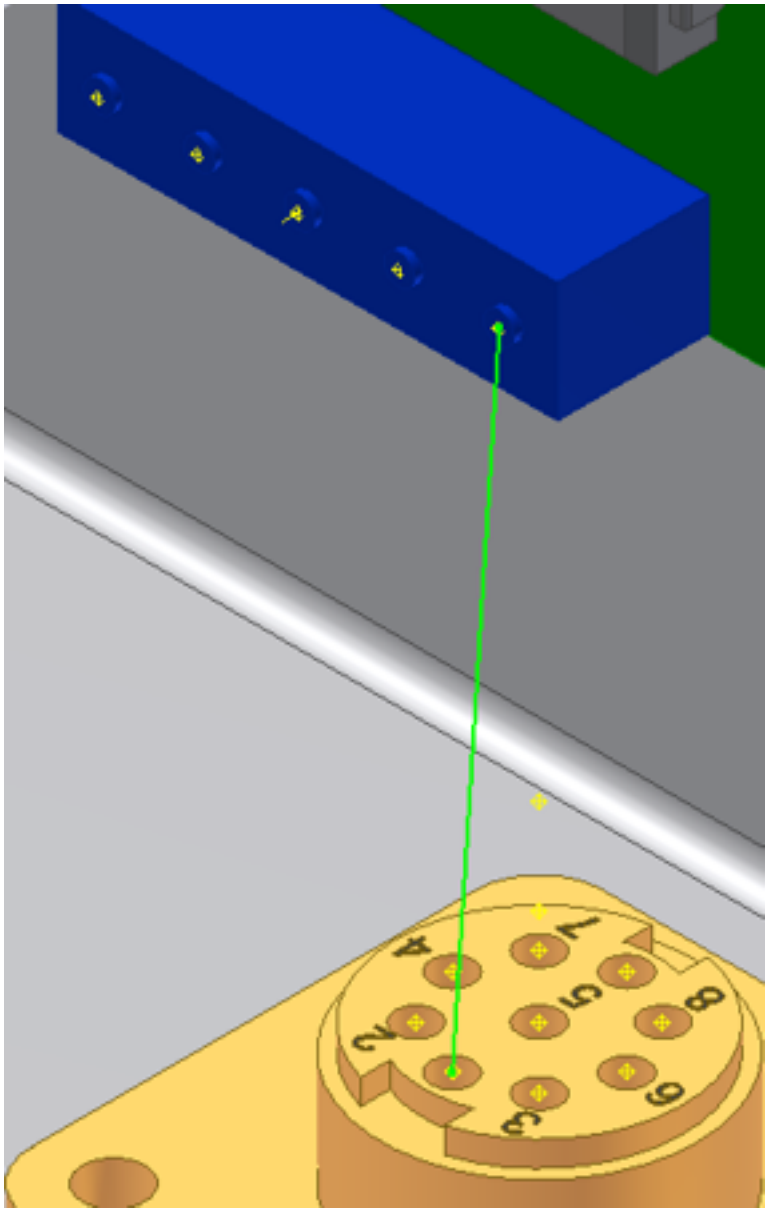


NOTE To turn off Dynamic Prompts, on the **Tools** tab, click **Application Options** . On the **General** tab, clear the check box for **Show command prompting** (Dynamic Prompts).

5 To end the wire, select **Pin 2**. Move the cursor over the pins on part **LTP**. When the tooltip displays as **J12 Pin 1**, click the work point.



6 To create the wire, click **Apply**. The wire is created as centerline or rendered, depending on your display setting. By default, the display is set to centerline for wires and cables.

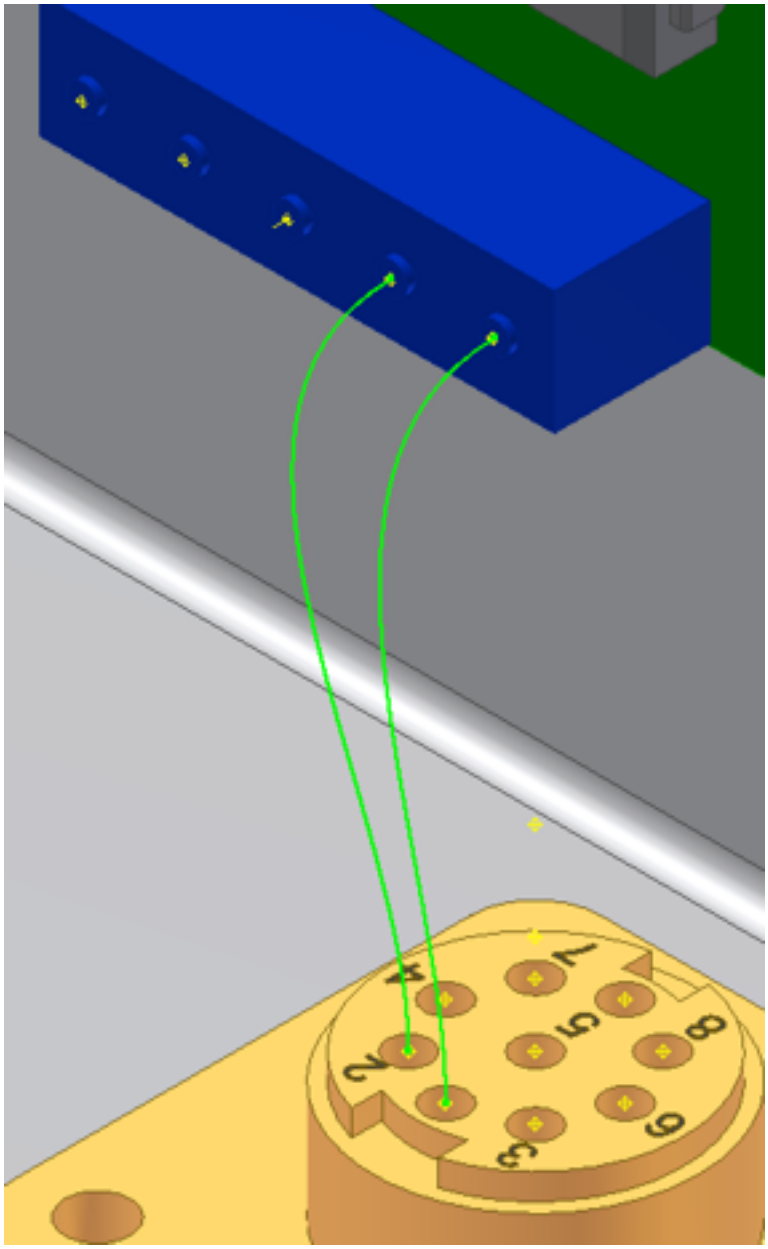


7 On the Create Wire dialog box, click **Cancel** to exit.

8 Before you add the second wire, access the Harness Settings dialog box for **Harness Assembly1**. Change the natural curvature setting for wires and cables to **With Natural Curvature**. 

9 Add one more wire connecting **U3 Pin 2** to **J12 Pin 2**.

NOTE If the wires are still depicted as straight lines, go back and reauthor the connectors. Natural curvature only displays between authored connectors, since the authoring process defines the Outward Direction.


4
7
5
8
2
6
3

10 Click **Finish Cable and Harness** to exit the Cable and Harness environment. You can view the added wires in the browser when you expand the **Wires** folder under **Harness Assembly1**.

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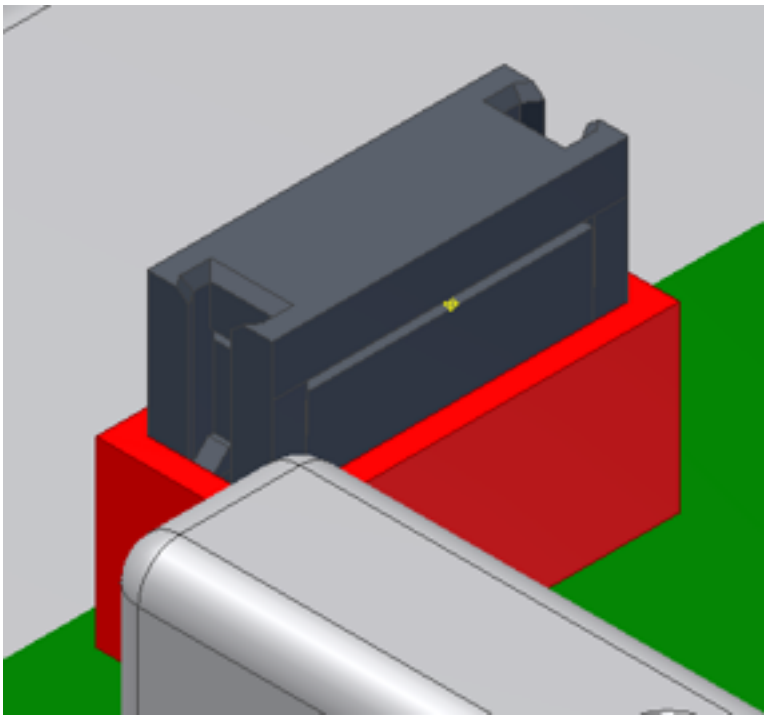
Create and Edit a Ribbon Cable

Create and edit a ribbon cable between two connectors.

1 On the **Assemble** tab, click **Place Component**. Place one instance of **ribbon_cable_connector.ipt**. This connector has already been authored, so it is a valid selection for a ribbon cable.

2 Use **Mate** constraints to constrain the ribbon cable connector to the female connector. Ensure that the final position of the ribbon cable connector is as shown.

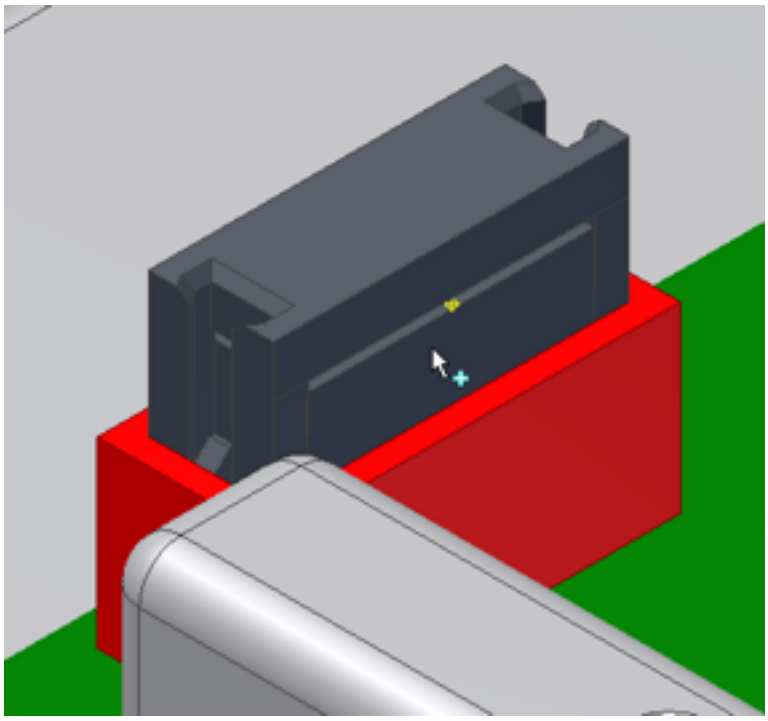


3 Activate **Ribbon Cable** in the browser.

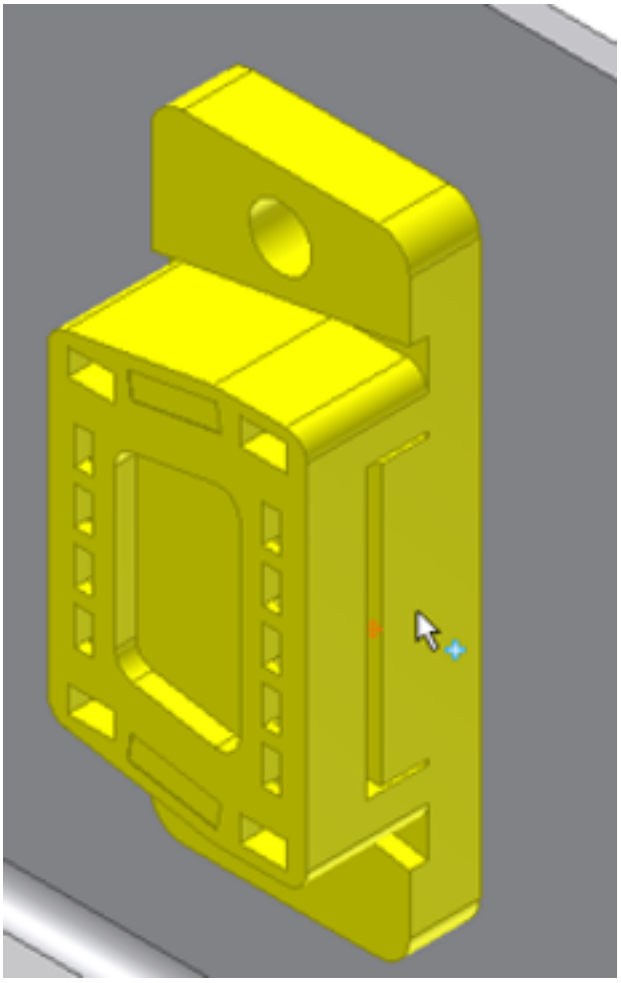
4 On the **Cable and Harness** tab, click **Create Ribbon Cable**.

5 Ensure that **Name** is set to **28AWG_10con** in the dialog box.

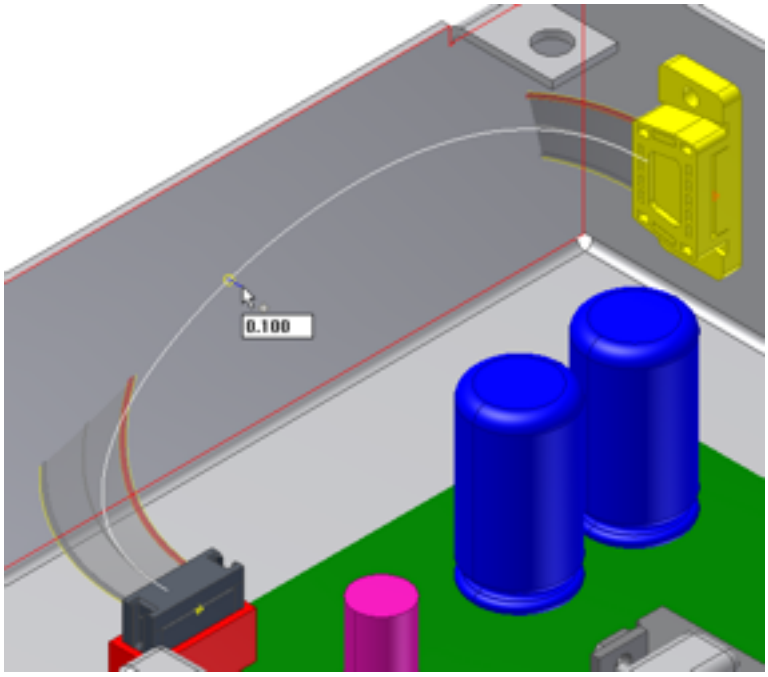
6 Select the start connector as shown. Ensure that **Start Pin** is set to **1**.



7 Select the end connector. Ensure that **Start Pin** is set to **1**.



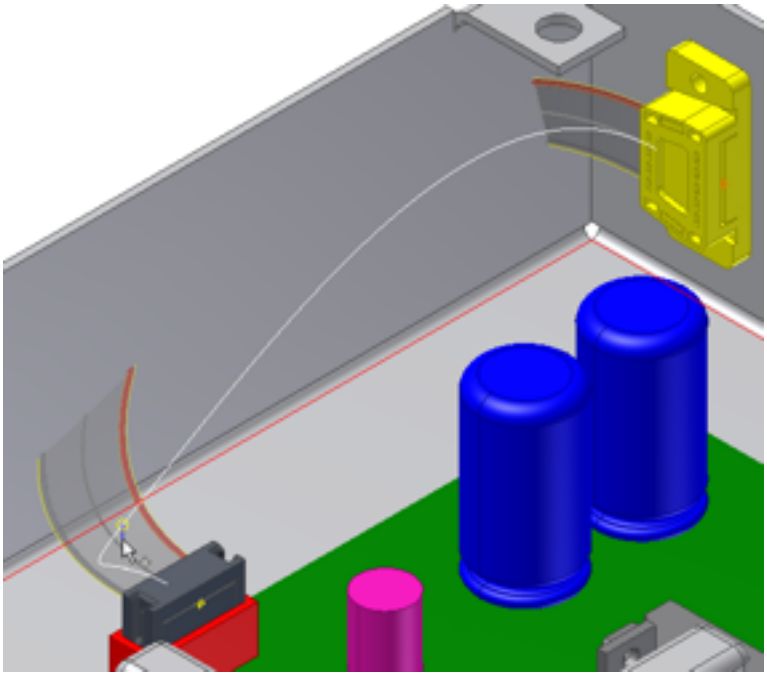
8 Click **OK**. The system draws a spline between the two connectors and shows a preview of the ribbon cable. You remain in spline creation mode.



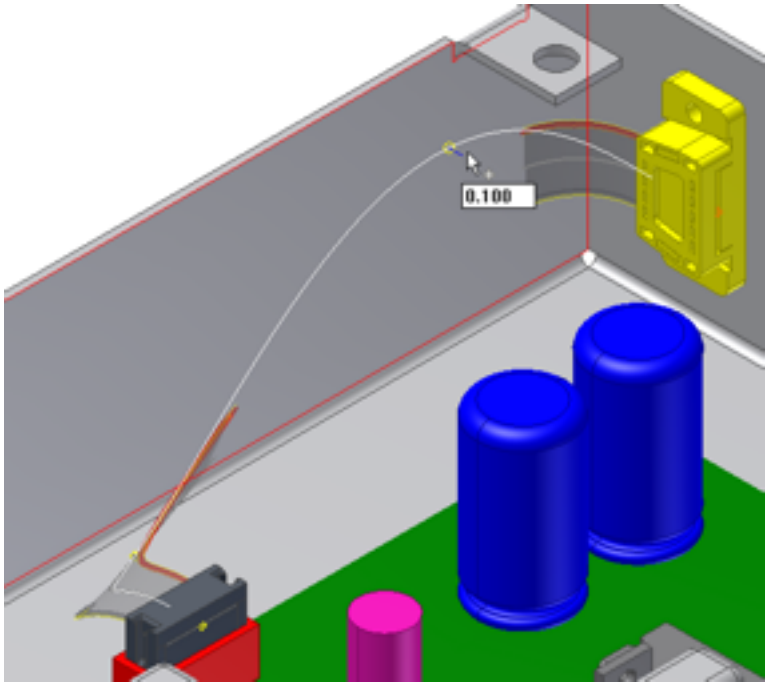
NOTE

- If the location and orientation of the connectors prevents the ribbon cable from joining the connectors, the ribbon cable outline does not preview.
- To control how the ribbon cable is rendered, click **Cable and Harness tab ➤ Visibility panel ➤ Rendered Display** or **Centerline Display**. This example uses **Rendered Display**.

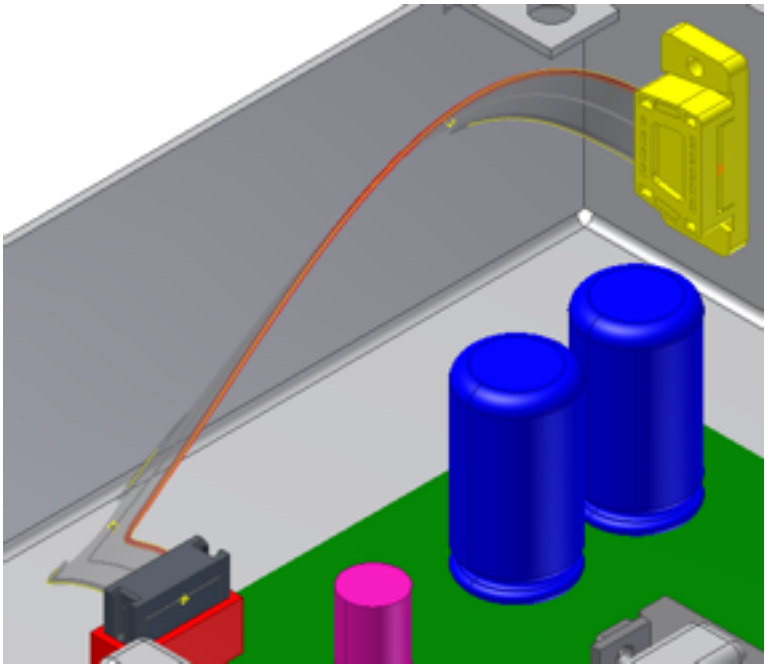
9 Add a point to the spline that is used to specify a fold location. Select the housing face, near the start connector, to locate the first point.



10 Add a point to the spline that is used as a control point to edit the cable twist. Select the housing face, near the end connector, to locate the second point.

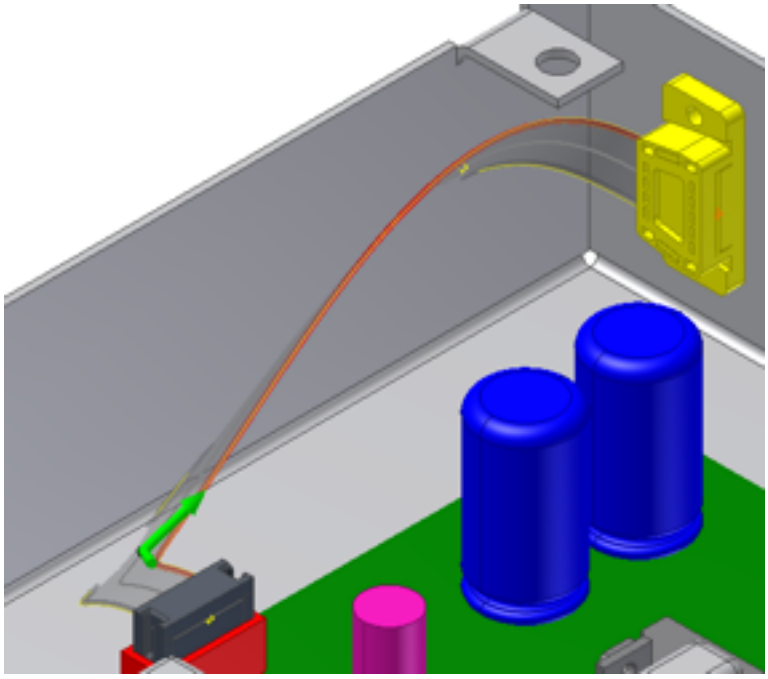


11 Right-click, and select **Finish**.



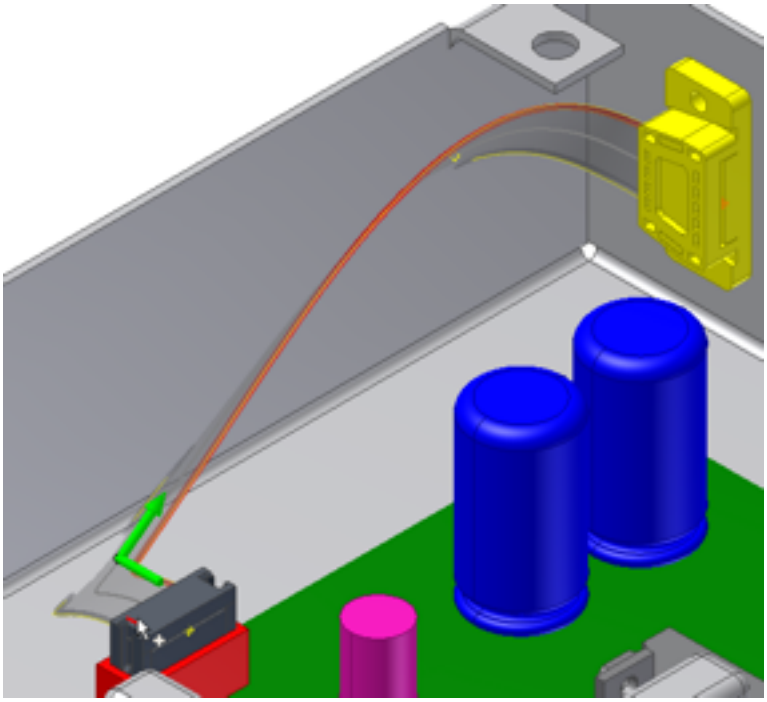
12 Right-click the first spline point you created, and select **Create Fold**.

An indicator attached to the point provides feedback about the orientation of the fold. Refer to this indicator to align the fold relative to model geometry.



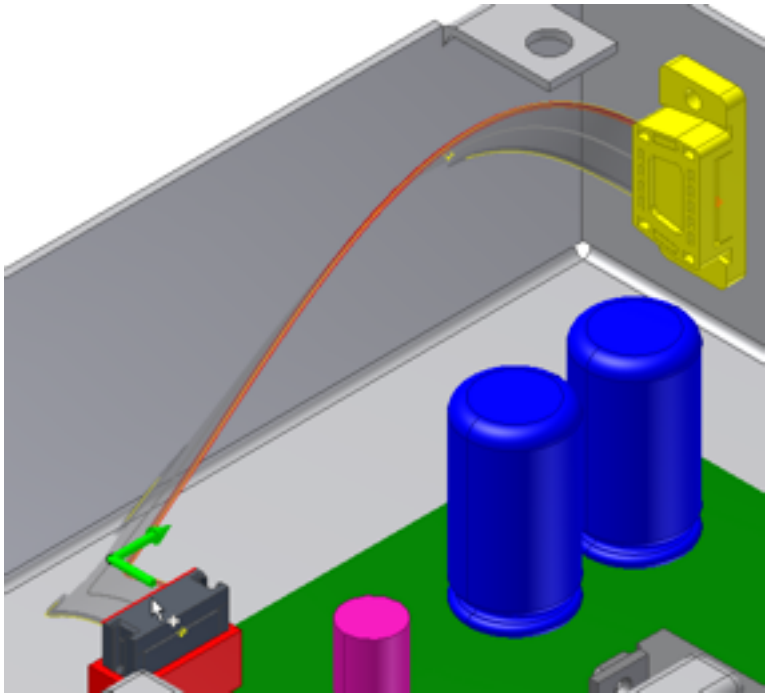
13 In the Create Fold dialog box, select **Shaft**.

This control specifies the orientation of the shaft portion of the indicator. Select a straight edge or a flat face on the connector. If you select an edge, the shaft aligns parallel to the edge. If you select a flat face, the shaft aligns normal to the face.



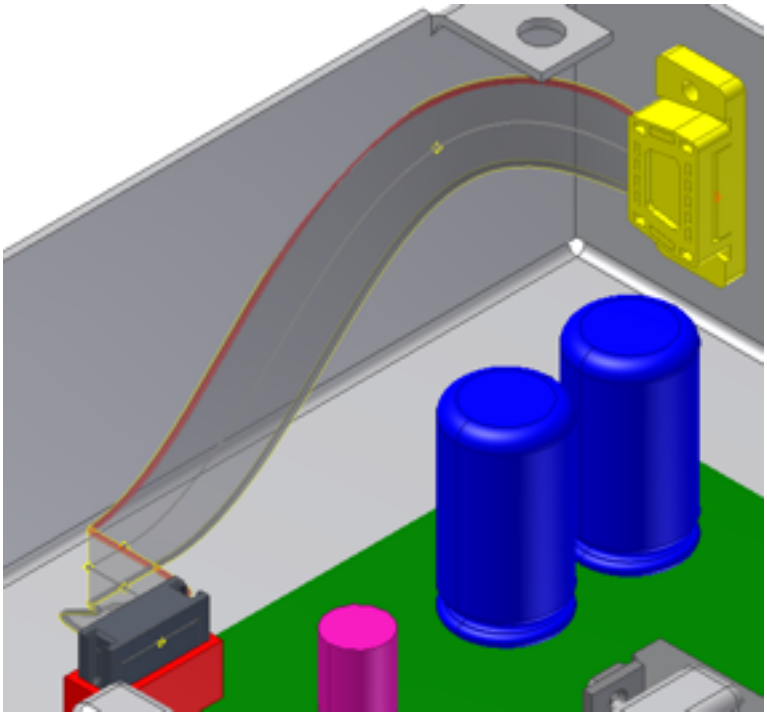
14 Select the arrowhead button. .

This control specifies the orientation of the arrowhead portion of the indicator. Select an edge or a face. Verify that the indicator matches the following image.



15 Ensure that **Single Fold** is selected.

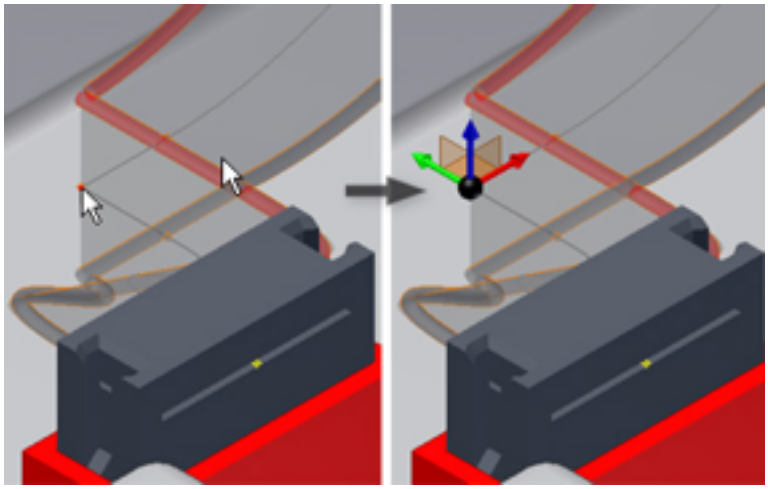
16 Click **OK**. A single fold added to the ribbon cable creates a right-angle direction change.



NOTE Two points are added automatically to the spline to define the fold geometry fully. The added points are not editable. Their positions are determined automatically as a by-product of the size of the ribbon cable and orientation of the fold.

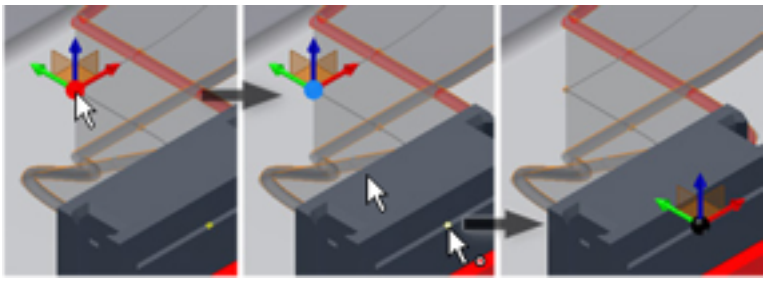
17 Depending on the location of the first spline point, the fold sometimes does not align laterally with the center of the pin row. In this example, the first spline point is displaced to one side of the row. If the lateral offset of the fold is not acceptable, align the spline point to the center of the pin row.

18 Right-click the fold point and select **3D Move/Rotate**.

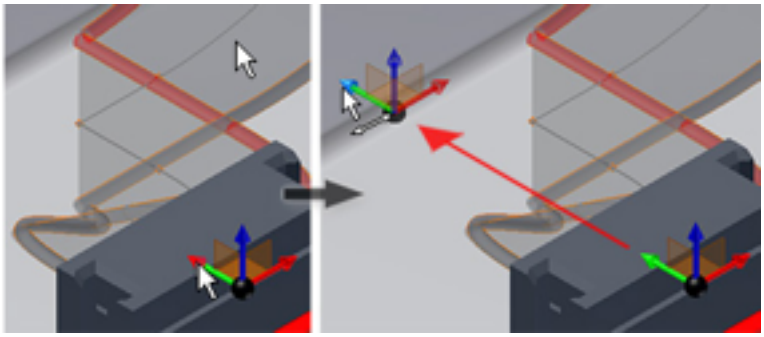


19 In the 3D Move/Rotate dialog box, click **Redefine alignment or position**.

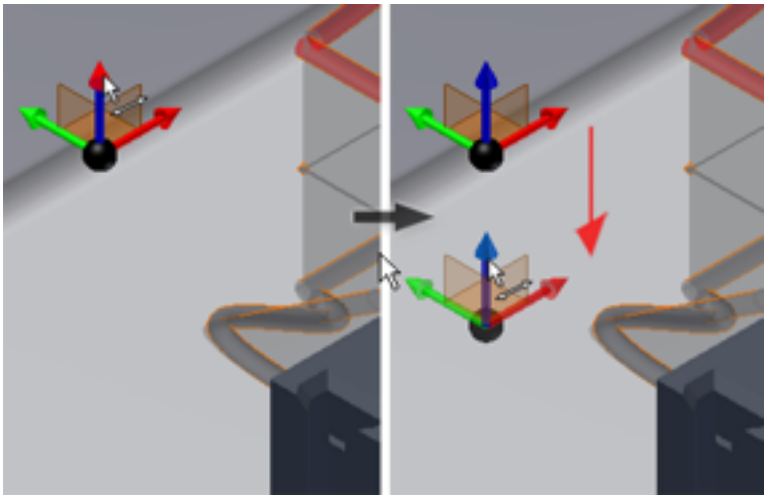
20 Select the triad sphere, and then select the work point on the connector. The triad relocates to the work point.



21 Select the **green** arrowhead, and drag the triad approximately 0.9 in along the Y axis. Alternatively, you can click the green arrowhead, and enter `0.9` in the **Y** control of the 3D Move/Rotate dialog box.

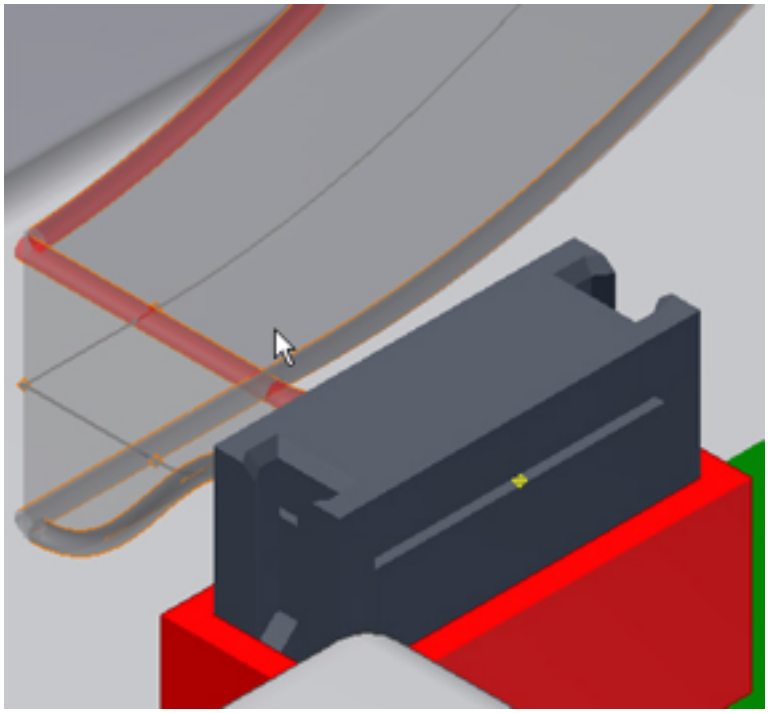


22 Select the **blue** arrowhead, and drag the triad approximately -0.3 in (negative 0.3 in) from its current location.

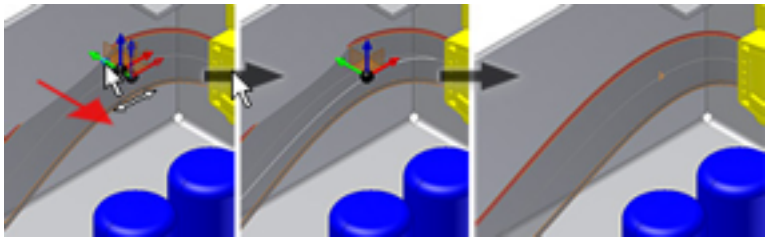


23 Click **Apply.** The ribbon cable fold adjusts and the 3D Move/Rotate dialog box remains open. If you click **OK**, the fold adjusts and the dialog box closes.

After you click **Apply** or **OK**, use **Undo** to cancel the 3D Move/Rotate result.

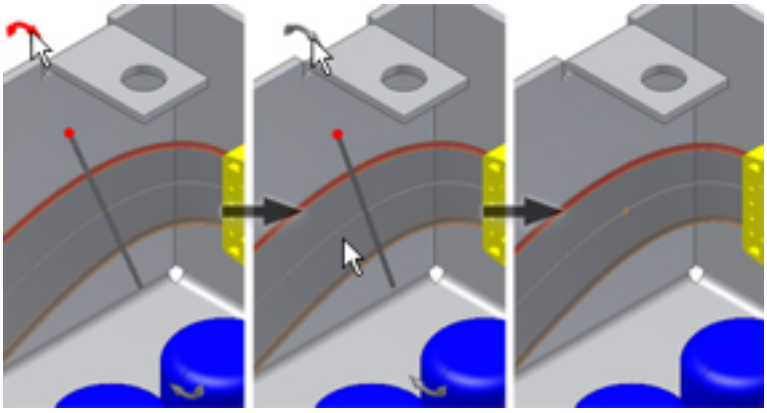


24 In this example, the ribbon cable interferes with the housing around the second intermediate spline point. Use **3D Move/Rotate** to move the point so that the ribbon cable clears the housing.



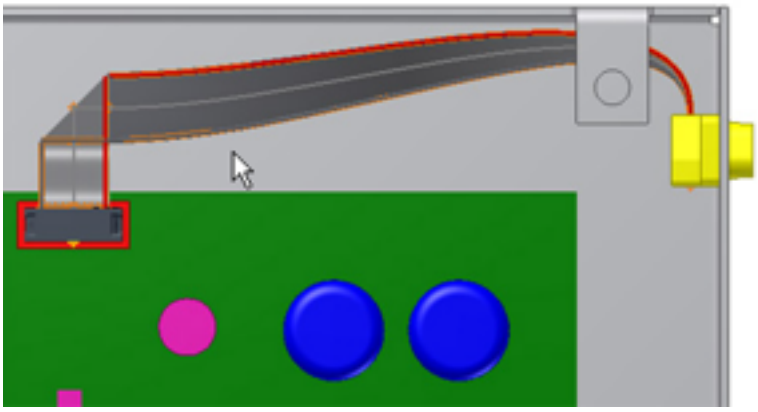
25 You can edit the twist of any intermediate spline point, unless a fold consumes the point. Right-click the spline point, and select **Edit Twist**. Drag the twist arrows.

To specify a precise angle instead, right-click the twist arrows and select **Enter Angle** . In this example, the twist control is rotated ten degrees.



NOTE Use the Plus and Minus keys (+ and –) to adjust the size of the twist control.

26 Right-click, and select **Apply**.



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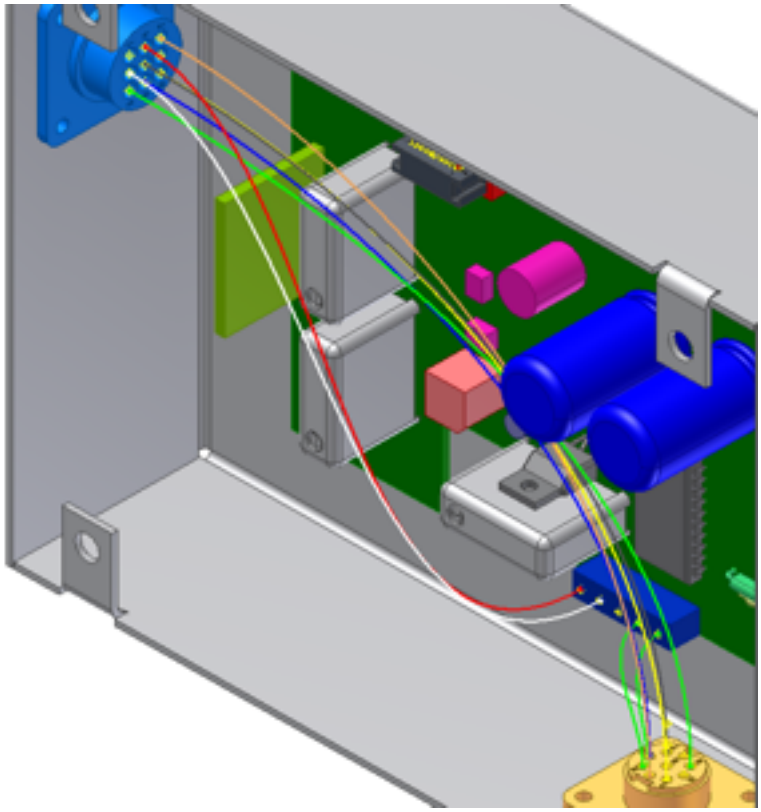
Create Segments and Route Wires

Use segments, and automatically route wires. In this section, you create and import additional wires between connectors. As more wires are added to your

design, it is easy to see the need for wire routes to avoid interference between the wires and components.

1. With **Harness Assembly1** active, on the **Cable and Harness** tab, click **Import Harness Data**  .
2. In the Import Harness Data dialog box, browse to the *WireListImport.csv* file for the Harness Data File.
3. For the Configuration File, browse to the *WireListConfiguration.cfg* file.
4. Click **OK**.
5. Another Import Harness Data dialog box is shown. Browse the data to determine how the harness is designed. The Harness Data and Configuration files controls the information in the dialog box.
6. Click **OK**.
7. View the Imported Harness Data log, or click **Close**.

   Seven discrete wires are imported into the harness assembly and connect part 360124:1 to 360575:1.

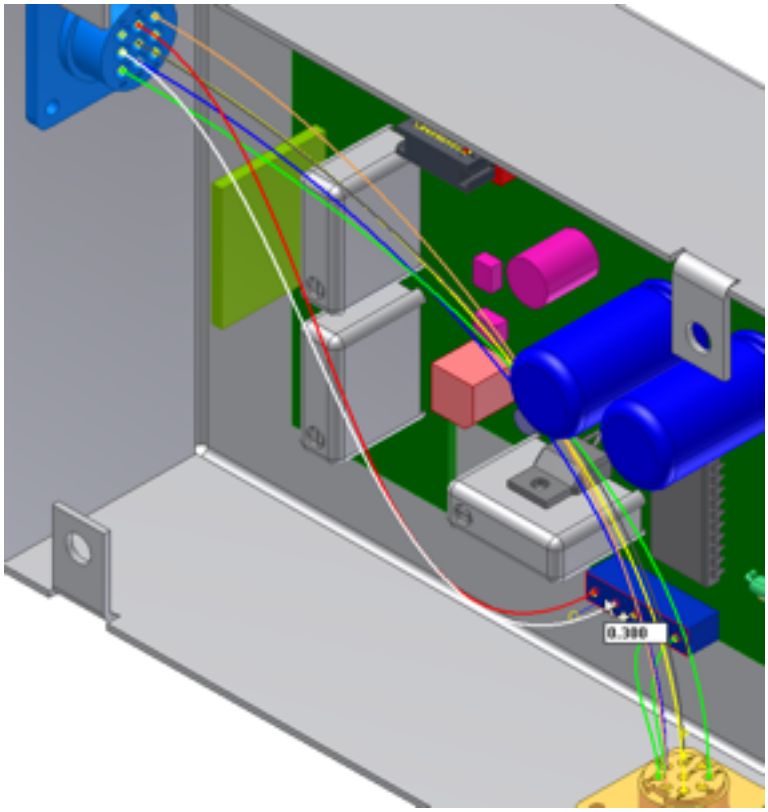


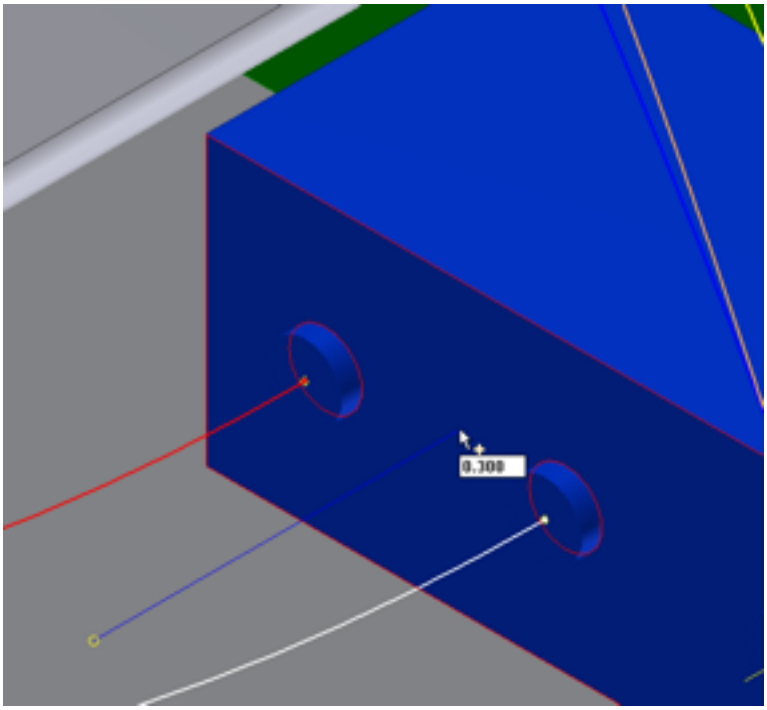
8 Click **Create Segment**.

9 Establish spline points that define the segment. The offset value defines the perpendicular offset from the point you click. The default offset of 0.100 inches is too low. Right-click, and select **Edit Offset**. Change the value to `0.300 inches`, and click **OK**.

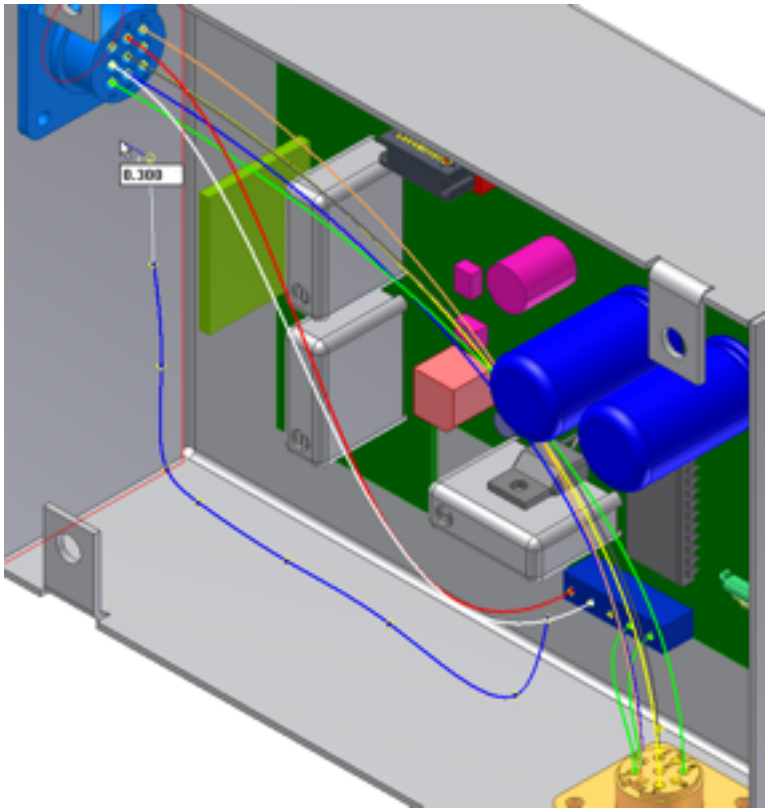
10 Click the front face of the LTP part between the red and white wires.

A spline point is created at an offset of 0.300 inches from the LTP face.

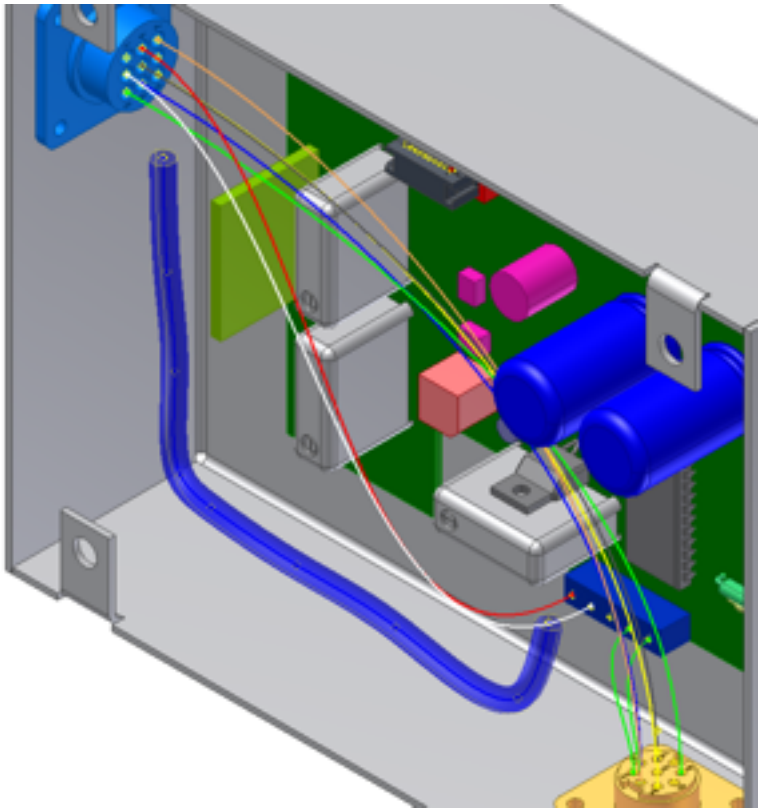

0.300



11 Click additional points along the enclosure walls. Ensure that the offset is shown to the interior of the enclosure. If the blue offset line is not visible, the point is created on the exterior of the enclosure.

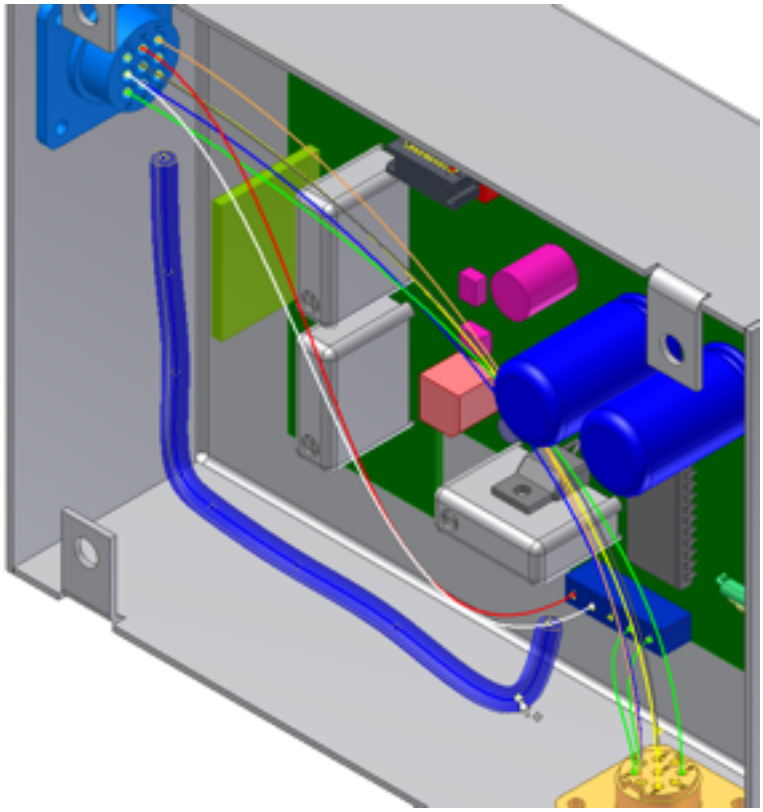


12 With enough points defined, right-click and select **Continue**. The segment is created around the spline points and the **Create Segment** command is still active.

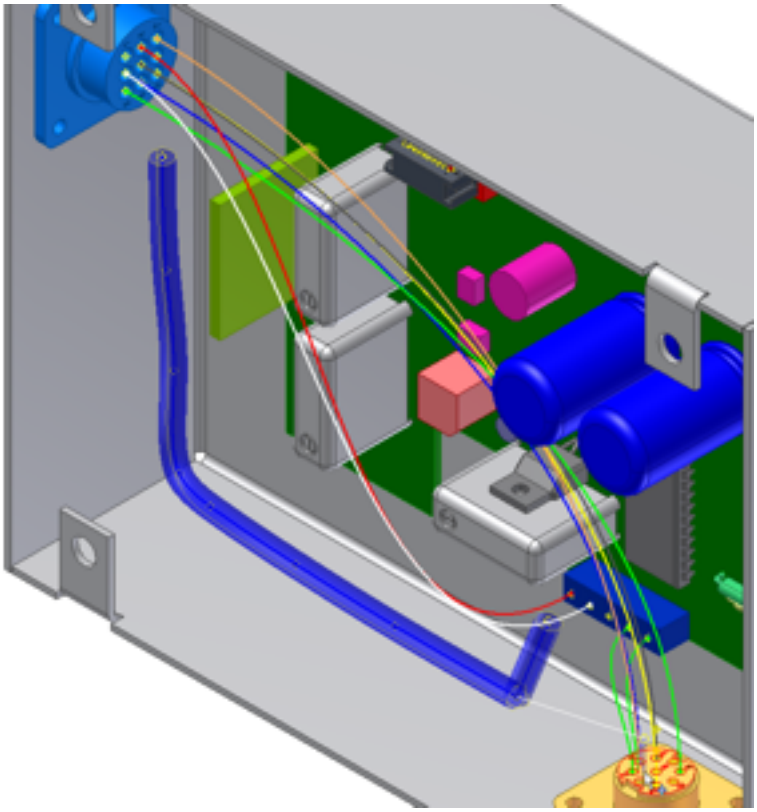


NOTE Use the **3D Move/Rotate** command to reposition the segment spline points after the segment is created.

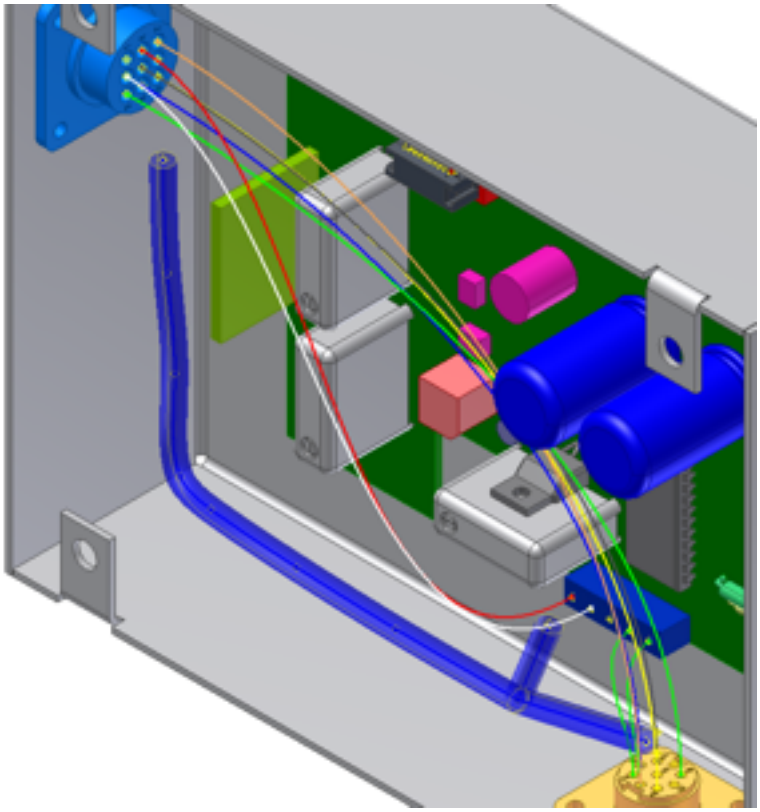
13 Create an additional segment. Click the segment spline point as shown.



14 Click the face of part 360124:1 as shown.

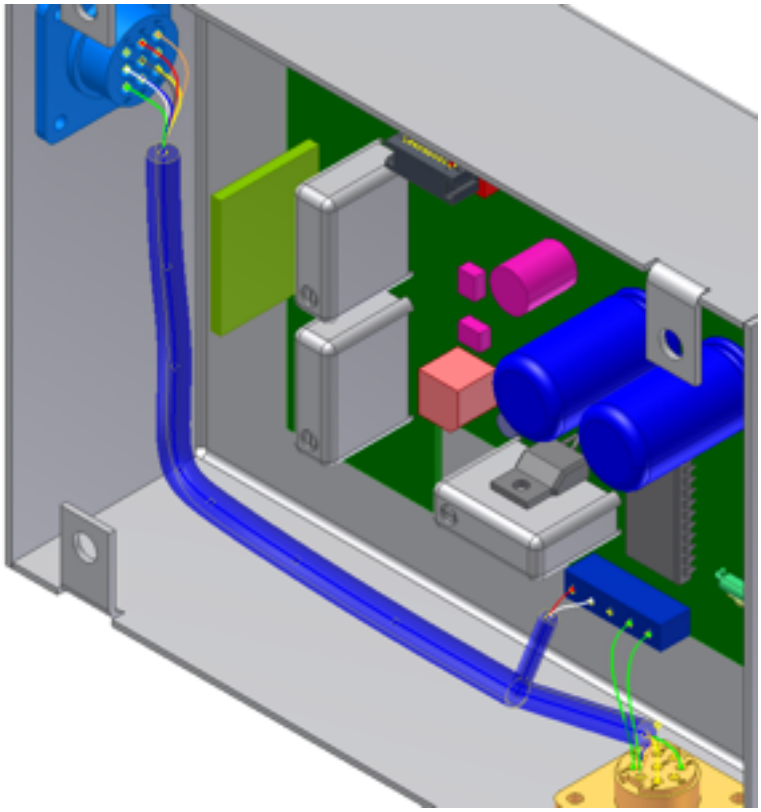


15 Right-click, and select **Finish**. The additional segment is created.



16 Click the **Automatic Route** command. 

17 Select all seven wires that you previously imported, and click **OK**. The wires are automatically routed through the nearest segment.



18 Exit **Harness Assembly1**, and save your file and the associated parts.

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Summary

In this tutorial, you:

- Created and assigned pins on a connector.
- Created and manipulated wires between connectors.
- Created and manipulated ribbon cables and folds.
- Created segments and routed wires.

There are many other features available in Cable and Harness. Proceed to the following successive tutorials to go deeper into the details of the functionality.

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Place Electrical Components

7

About this tutorial

Place electrical components and work with properties in an assembly.

Category	**Routed Systems**
Time Required	20 minutes
Tutorial Files Used	Enclosure_Assembly.iam

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

This tutorial is the first in the set of successive tutorials. You work with an assembly file provided.

Objectives

- Place pins and define pin-level properties.
- Add RefDes placeholders.
- Add custom properties to parts.

Prerequisites

- Know how to set the active project, navigate the model space with the various view commands.

- See the Autodesk Inventor Help topic “Fundamentals” and the New User tutorials for further information.

Navigation Tips

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

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Open Assembly

The assembly used in this tutorial is also used in other Cable & Harness tutorials. Be sure to save changes made in this assembly to continue with further Cable & Harness tutorials.

Back up the Cable & Harness directory in the Tutorials Files directory before beginning the tutorial.

Open the assembly

1. Click  ➤ **Open**.
2. Set the Project File to **tutorial files.ipj**.
   - For Microsoft Windows XP, the default file location is *Program Files\Autodesk\Inventor <version>\Tutorial Files*
   - For Microsoft Windows Vista, the default file location is *Public\Public Documents\Inventor <version>\Tutorial Files*
3. Open the tutorial file **Enclosure_Assembly.iam** located in the Cable & Harness subfolder.

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Place Pins

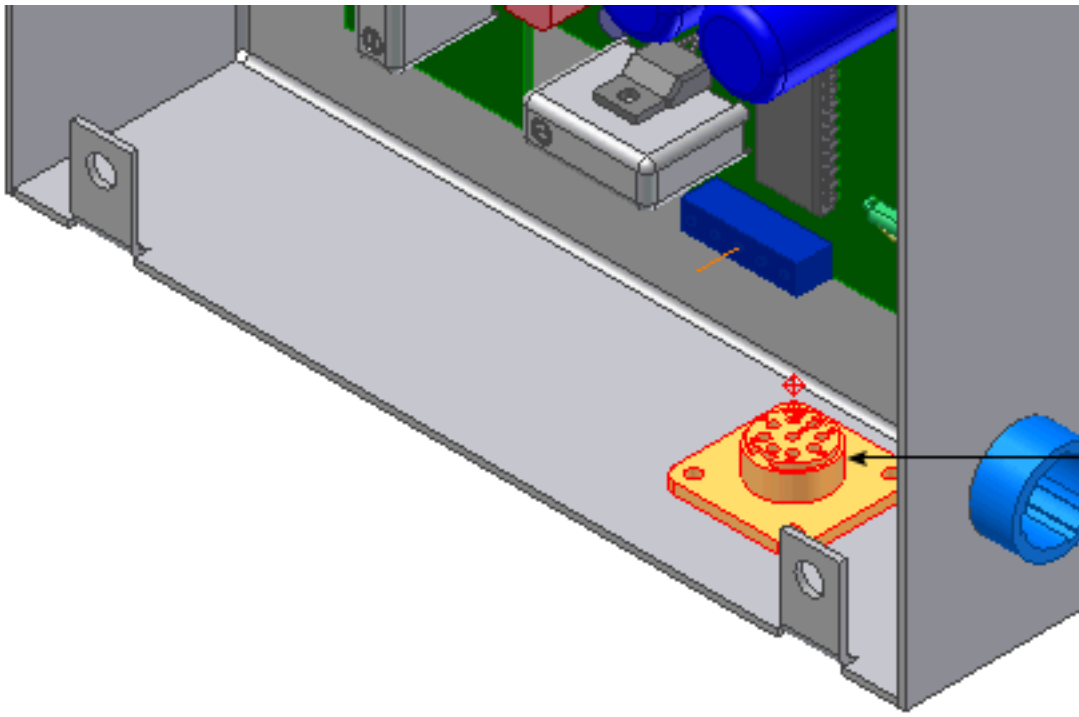
Edit the part in place to add individual pins and a placeholder reference designator to a part. The connectors are already placed in the assembly.

Each pin name within the selected part is unique. The pin name specified is also the name of the special work point feature listed in the browser.

Add pins in the assembly

1. In the browser, pause the cursor over the part named 360124.

   The part highlights in the graphics window so that you can see the part to edit.



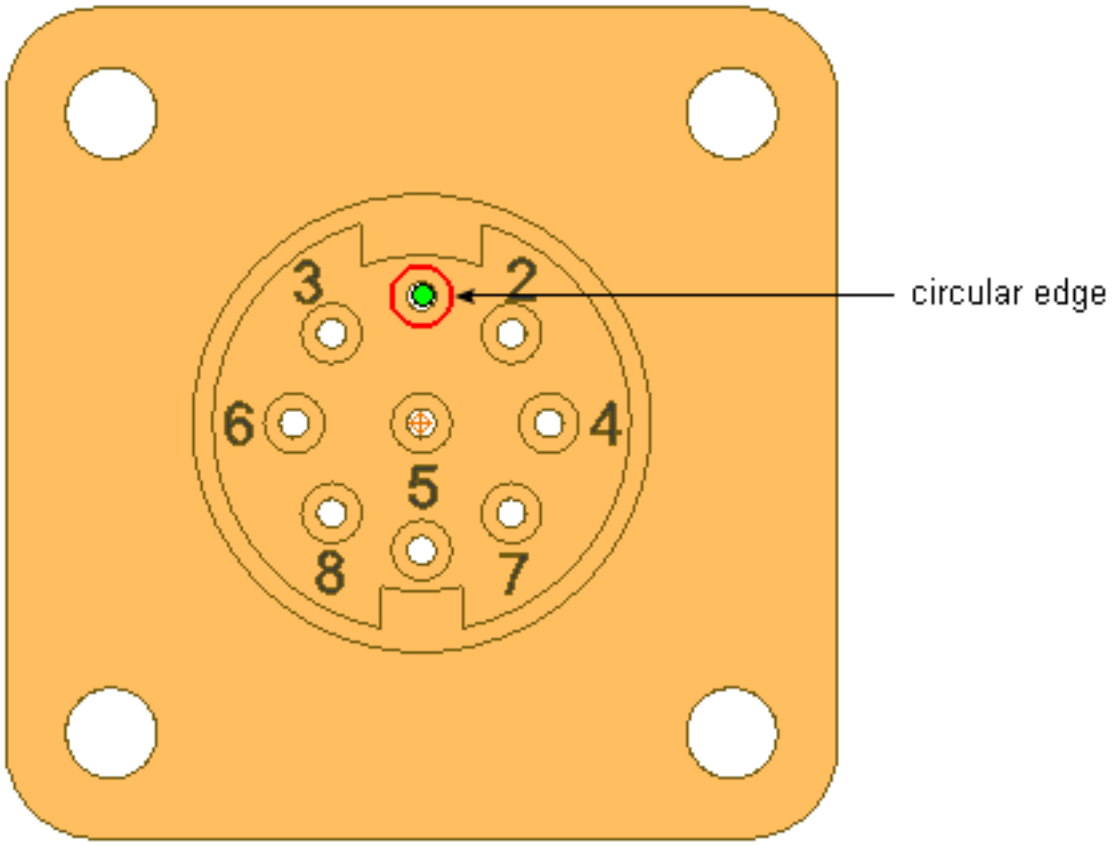
2. In the browser, double-click the part name to edit the part.

3 In the Model browser, delete work points 1 through 9. You re-create these points in the next steps.

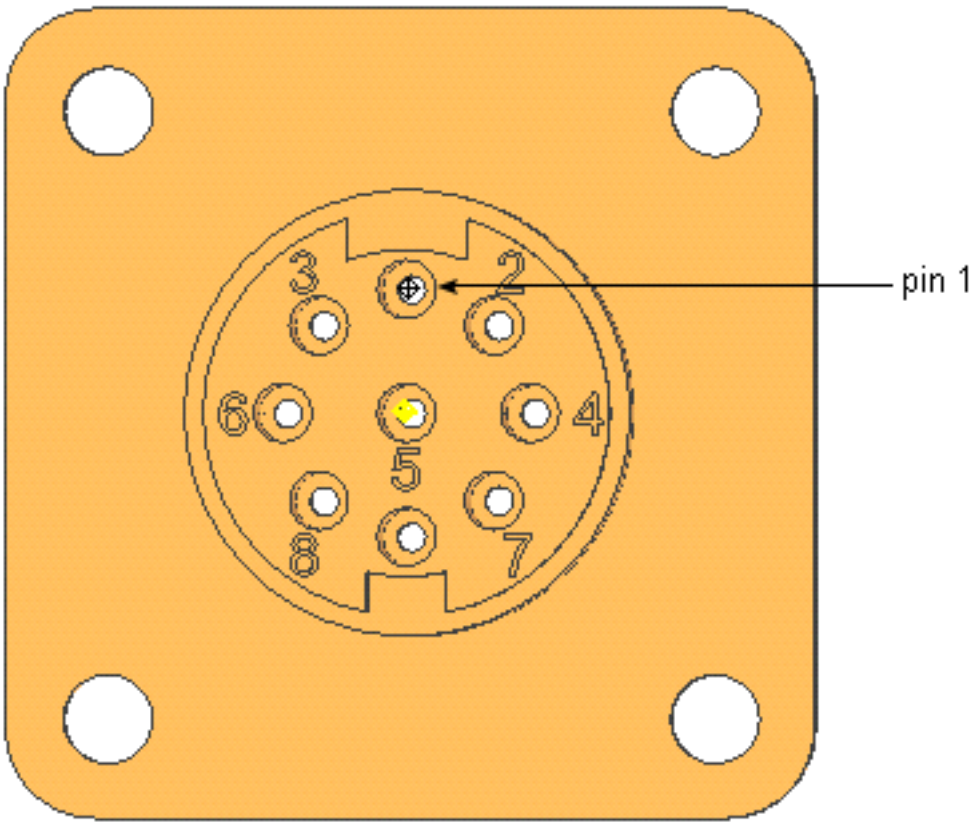
4  On the ribbon, click **3D Model tab ➤ Harness panel ➤ Pin**.

5 Rotate and zoom the view to see the part as shown in the following image. Turn visibility off for any parts obstructing your view.

6 Click the circular edge shown to select the center point.



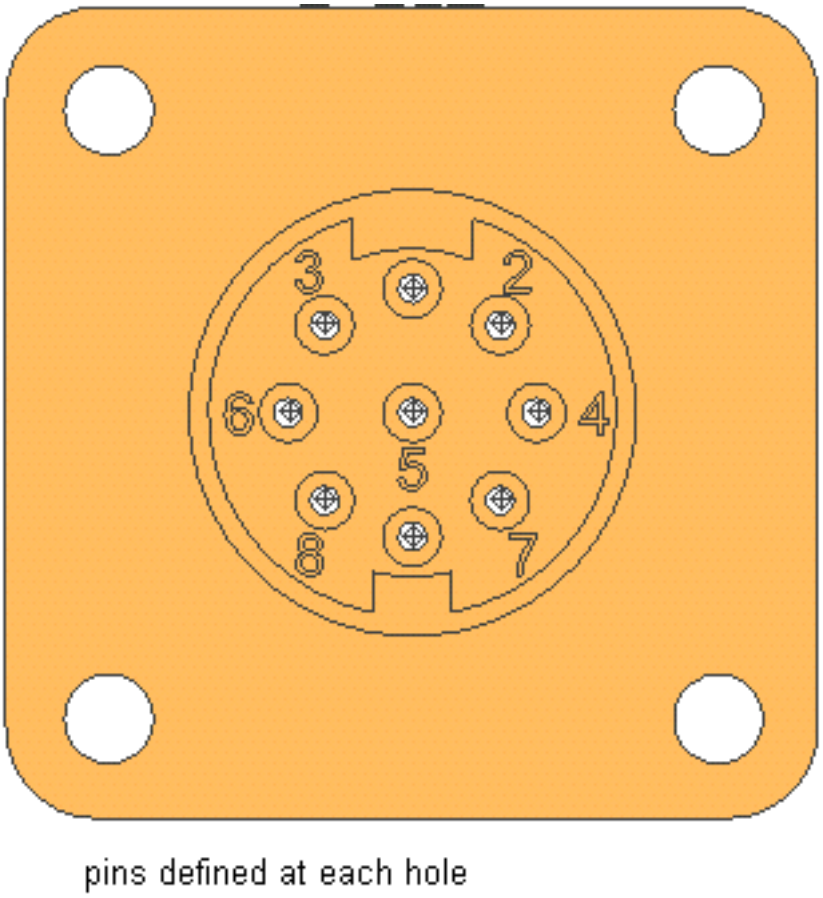
7 On the Pin Name dialog box, use the default pin name of **1** and click  . The work point representing the pin is added for pin 1.



8 Define a pin for hole 2:

- Select the circular edge.
- Accept the Pin Name of **2**.
- Click  .

9 Repeat the steps to define a pin at each labeled hole and hole 9 located between holes 7 and 8. Name the pins to match their number label. For example, enter a Pin Name of 3 for the hole labeled as 3, and so on.



10 To exit Place Pin mode, right-click, and then select **Done**.

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Add RefDes Placeholder

Add a placeholder RefDes to the electrical part. In a later exercise, you set the RefDes value for the occurrences in the assembly.

1.  With nothing selected, select **3D Model tab ➤ Harness panel ➤ Properties**.

   Alternatively, in the browser, right-click the part and select **Harness Properties**.
2. On the **General** tab of the Part Properties dialog box, enter **U?** in the RefDes (reference designator) field.
3. Click **OK** in the Part Properties dialog box.
4. If the Cable & Harness message box displays, select **Do not show the message box again in this session** and click **OK**. After the message box closes, click **OK** on the Part Properties dialog box again.

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Add Custom Properties to Parts

Each custom property requires a data type. Each data type, except for text, has a default unit associated with it.

1. In the browser, right-click **360124:1**, and then select **Harness Properties**. Make sure that you are in-place editing 360124 to access the harness properties.
2. Click the **Custom** tab, and enter the values as follows:

   Name:`Vendor`

   Type:`Text`

   Value: `ACME`
3. Click **Add**, and then click **OK**.

   The property is added to the list, and the dialog box closes.

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Author Connector

Authoring the connector completes the definition of the connector for use with Cable & Harness.

1.  Click **Manage tab ➤ Author panel ➤ Connector**. The Connector command is located under the Component flyout.
2. Click the top face of 360124:1to define the Outward Direction. The Outward Direction controls how wires exit the connector when using natural curvature.
3. Notice the RefDes value entered earlier is automatically populated in the Connector Authoring dialog box.
4. Click **OK** to author the connector.
5. Click **OK** to close the message box. In this tutorial, we do not add the connector to Content Center.

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Configure Additional Connectors

Use the skills you learned to add pins and RefDes placeholders to two other parts, and then author the connectors.

1. Edit the part **360575:1**. Use the  Pin command to add pins at each hole. Name the pins to match their number label.

2 Use the  Harness Properties command to assign the RefDes placeholder `U?` and add the custom Vendor property with value ACME.

3 Use the  Connector Authoring command to set the outward direction for the connector. Select the top face to set the outward direction.

4 In the browser, edit part **360575:2** and notice how this occurrence inherits the pins set in 360575:1. After you pin the part once, all occurrences inherit the pins.

5 In the browser, expand the **PCB** subassembly to locate and edit part **LTP**. Use the  Pin command to add the pins as shown. Pin 1 is the pin closest to part 360124.

6 Use the  Harness Properties command to assign the RefDes placeholder `J?` and add the custom Vendor property with value ACME.

7 Use the  Connector Authoring command to set the outward direction for the connector. Select the top face to set the outward direction.

8 Return to the top-level assembly and save the assembly for use in the next tutorial.

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Summary

In this tutorial, you:

- Placed pins and defined pin-level properties.
- Added RefDes placeholders.
- Added custom properties to parts.

Use the saved assembly from this tutorial to work with harness assemblies in the next tutorial.

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Harness Assemblies

8

About this tutorial

Create the harness assembly to begin designing a harness in your assembly model.

Category	**Routed Systems**
Time Required	10 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial.

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

Objectives

- Create a harness assembly.
- Review the Cable & Harness commands.
- Assign reference designators (RefDes) to connector occurrences.

Prerequisites

- Perform the previous tutorial, Place Electrical Components, and save the tutorial file for use in this tutorial.
- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic “Fundamentals” and the New User tutorials for further information.

Navigation Tips

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

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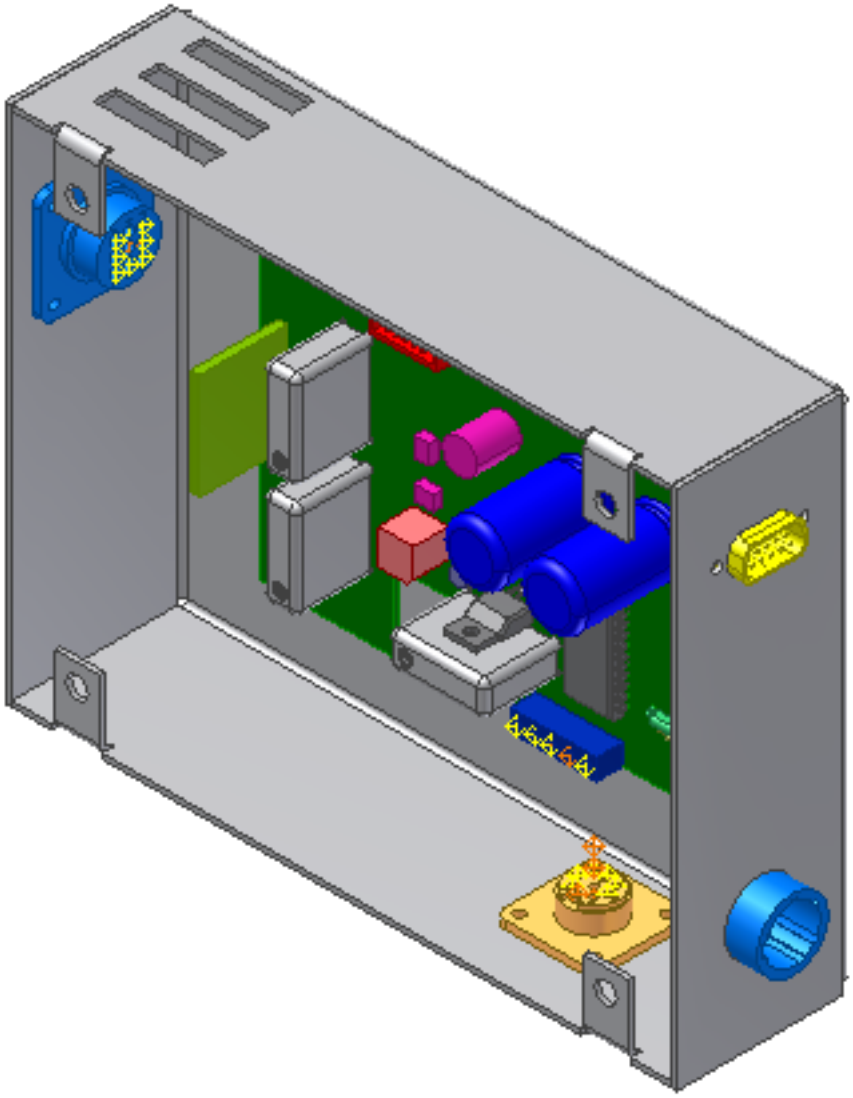
Create Harness Assemblies

Open your assembly and a harness assembly. The harness assembly holds the cable and wire harness data.

Create a harness assembly

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.



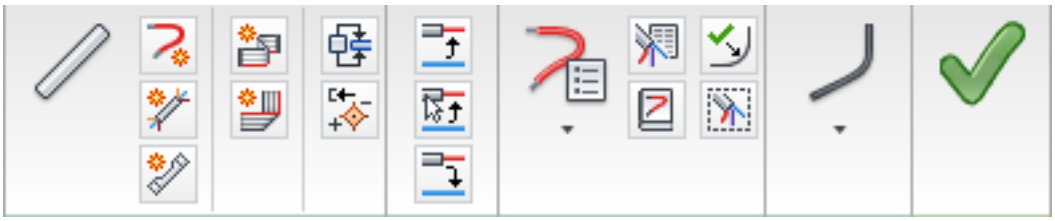
2  On the ribbon, click **Assemble tab ➤ Begin panel ➤ Cable and Harness**.

3 On the Create Harness dialog box, enter **Harness Assembly1.iam** for the name for the harness assembly, and accept the default location.

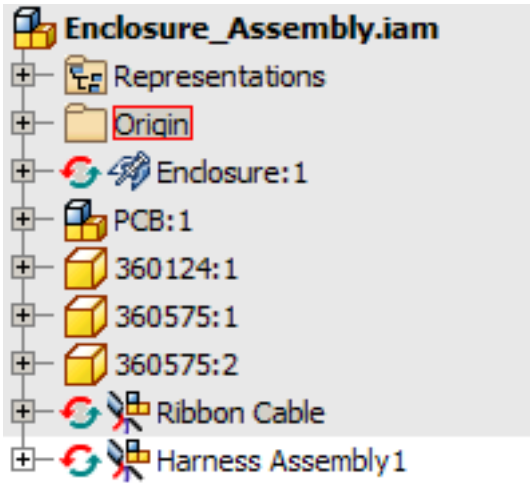
4 Click **OK**.

The system adds the harness assembly to the browser and activates the cable and harness environment.

5 The Cable & Harness tab is displayed on the Ribbon. Pause the cursor over the commands in the Ribbon to view the tooltips.



6 Locate the new harness assembly in the browser hierarchy.



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Assign Reference Designators to Each Occurrence

Now that the harness assembly is added, set a reference designator for each electrical part occurrence. The reference designators are specific to the harness assembly.

1 Make sure that you are editing **Harness Assembly1**.

2 In the browser, right-click part **360124:1** and select **Harness Properties**.

3 On the Part Properties dialog box, set a RefDes of **U3** for the occurrence, and click **OK**.

4 Repeat to set a RefDes of **U7** for part **360575:1**.

5 Set a RefDes of **U8** for part **360575:2**.

6 In the browser, expand the **PCB** subassembly to locate part **LTP**, and set a RefDes of **J12**.

7 Return to the main assembly.

8 Save the assembly for use in the next tutorial.

NOTE The unique values override the generic RefDes placeholders at the occurrence level. The designators are specific to the harness assembly. If you create additional harness assemblies, you can follow the same steps, but use different designators.

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Summary

In this tutorial, you:

- Create harness assemblies.
- Become familiar with the Cable and Harness tab, display tools, and browser.
- Set occurrence level harness properties.

Use your saved file from this tutorial to work with Cable and Harness libraries in the next tutorial.

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Cable and Harness Library

9

About this tutorial

Locate the cable and harness library data, and then work with library definitions. Explore the import and export of library data.

Category	**Routed Systems**
Time Required	30 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial. Cable&HarnessDefaultLibrary.iwl LibraryGXLWires.csv WireLibrary.cfg

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

Objectives

- Locate the cable and harness library.
- Work with library definitions.
- Import and export library data.

Prerequisites

- Perform the previous tutorial, Harness Assemblies, and save the tutorial file for use in this tutorial.
- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic “Fundamentals” and the New User tutorials for further information.

Navigation Tips

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

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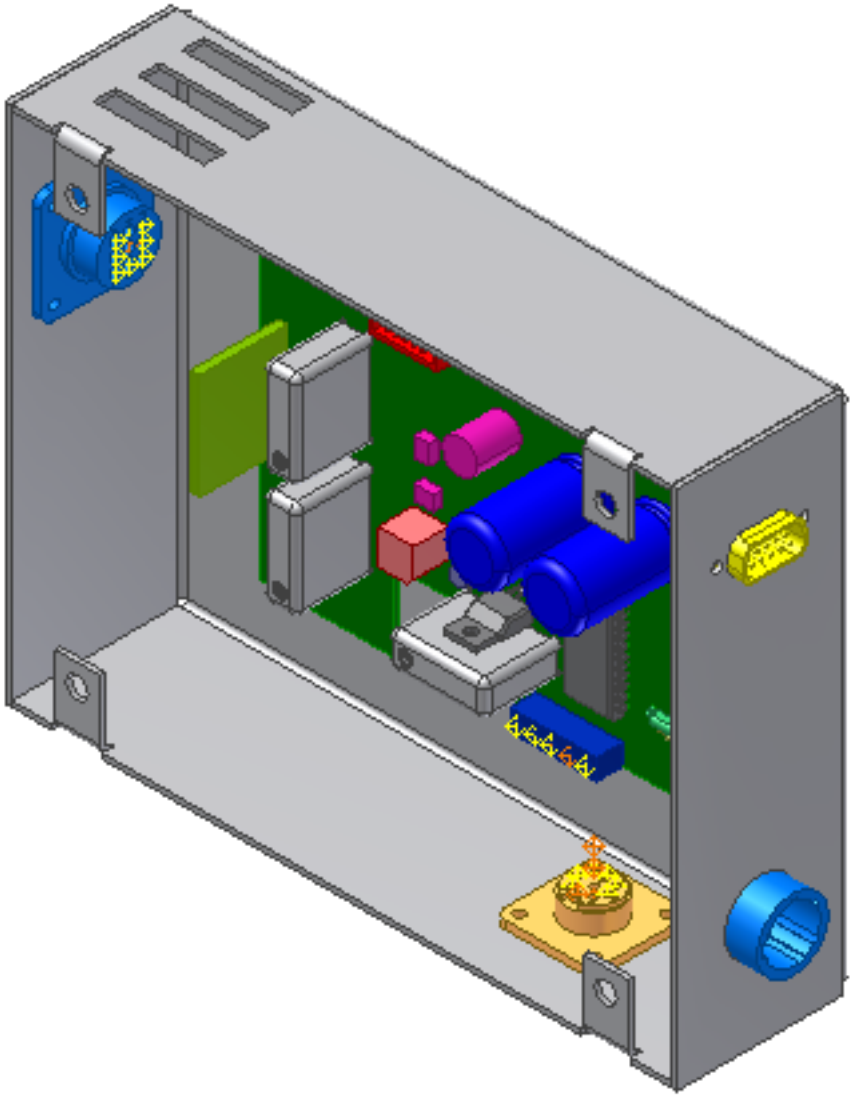
Locate the Cable and Harness Library File

Determine the location of the Cable and Harness library file. This tutorial uses the default location and library file.

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.



2 In the browser, double-click **Harness Assembly1** to edit it.

3 In the browser, right-click Harness Assembly1, and then select **Harness Settings** from the context menu.

4 On the Harness Settings dialog box, click the **File Locations** tab.

This harness assembly references the default cable and harness library file as set for the Design Data of the project. The default file and location is:

Microsoft®Windows®XP operating system:

- *Program Files\Autodesk\Inventor <version>\Design Data\Cable & Harness\Cable&HarnessDefaultLibrary.iwl*

Windows Vista®operating system:

- *Users\Public\Public Documents\Autodesk\Inventor <version>\Design Data\Cable & Harness\Cable&HarnessDefaultLibrary.iwl*

5 For this exercise, leave the library in the default location. Click **Cancel**.

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Create a Wire Library Definition

Create a definition for a wire type in the default library.

1  With the harness assembly still active, click **Cable and Harness tab ➤ Manage panel ➤ Library**.

2  On the Cable & Harness Library dialog box, ensure that **Wire** is the selected object type, and then click **New**.

3 On the Library Wire dialog box, General tab, in the Properties area, specify:

Name: `BLU_EE_22`

Category:`Alpha`

Part Number:`60820`

Appearance:`Blue`

4 In the Physical area, enter the following values related to the physical parameters of the wire:

Outer Diameter:`0.0540 in`

Gauge:`22`

5 Leave Core Size and Bend Radius blank.

6 Click **Save** to save the wire to the referenced library and do not close the dialog box.

The BLU_EE_22 wire is added to the Alpha list, and is saved for use in any harness assembly that references the library file.

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Create a Label Library Definition

Create a definition for a label in the default library.

1 With the Cable & Harness Library dialog box still open, click the arrow on the object type list, and change it from **Wire** to **Label**.

2  Click **New**.

3 On the **General** tab in the Properties area, specify:

Name:`Wire ID Label`

4 Under Default Contents, click the arrow to select **Wire ID** from the Property list, and then click **Insert Property**.

5 Click **Save** and do not close the dialog box.

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Copy Library Definitions

Copy the wire definition that you created.

1 With the Cable & Harness Library dialog box still open, change the object type list to **Wire**.

2 Select the **BLU_EE_22** wire that you created. It is located at the bottom of the Alpha category.

3  Click **Copy**.

A copy of the wire BLU_EE_22 is added to the wire list under the original wire. The default name of the copied wire is BLU_EE_22 Copy.

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Edit Properties of a Library Definition

Edit the properties of the wire definition you copied. You change the name and category for the definition.

1  On the Cable & Harness Library dialog box, select the wire **BLU_EE_22_Copy** in the list, and then click **Edit**.

As an alternative, you can right-click the wire in the list, and then select **Edit** from the context menu.

2 Change the settings as follows:

Name:`BLU_EE_22`

Category:`Generic`

NOTE Invalid names, such as those that duplicate an existing name, are indicated in red.

3 Click **Save**. Keep the Cable & Harness Library dialog box open.

The list of wires automatically updates to show the renamed wire in the Generic category.

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Add Properties to a Library Definition

Add custom properties to an existing library definition.

1. On the Cable & Harness Library dialog box, expand the Generic category, and then select the wire named **22AWG-GRN**.
2.  Click the **Edit** command.
3. Click the **Custom** tab.
4. Enter the following values, and then click **Add** to include the property in the list.

   Name:`Vendor`

   Type:`Text`

   Value:`5876`
5. Repeat with the following values to add another property.

   Name:`Volt`

   Type:`Voltage`

   Value:`300 V`
6. Select the **Color** property from the list.
7. Enter the following values then click **Modify** to change the property.

   Name:`Color`

   Type:`Text`

   Value:`Green/Yellow`
8. Click **Save**. Keep the Cable & Harness Library dialog box open.

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Delete Library Definitions

Delete one of the wire definitions you created.

1. On the Cable & Harness Library dialog box, select **BLU_EE_22A**. The wire is located under the Generic category.

2  Click **Delete**, and then click **Yes** to confirm the deletion.

Alternatively, you can right-click the wire, and then select **Delete** from the context menu.

3 Look at the list to verify that the wire was deleted. Remain in the Cable & Harness Library dialog box.

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Import into Cable and Harness Library

Import wires into the library. The configuration and data files provided for the tutorial.

The workflow to import definitions into the Cable and Harness Library is the same for all object types. Select the appropriate object type to import.

1  On the Cable & Harness Library dialog box toolbar, click **Import Library Objects** .

2 On the Import Library Wires dialog box, click **Browse** in the Data File area and select **LibraryGXLWires.csv** from the following directory.

- For Microsoft Windows XP, the default file location is *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness\Wire Library*
- For Microsoft Windows Vista, the default file location is *Public\Public Documents\Inventor <version>\Tutorial Files\Cable & Harness\Wire Library*

3 Repeat to set the Configuration File to **WireLibrary.cfg** located in the same directory.

4 Click **OK** to add the wire data to the open library.

A dialog box appears indicating the status of the import and confirms the name and location of the log file. The log file contains details of the import, including updates to wires, and errors associated with the import.

5 Click **Close** on the confirmation dialog box.

The Cable & Harness Library dialog box updates with the GXL data after the import.

6 Leave the Cable & Harness Library dialog box open.

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Export Library Data

Export the library wire definitions to a delimited file. Use the same configuration file that was used in the import process. Exported property values use the unit preference settings.

1  On the Cable & Harness Library dialog box toolbar, with the Wire object type selected, click **Export Library Object** .

2 On the Export Library Wires dialog box, click **Browse** in the Data File area and to the following directory.

- For Microsoft Windows XP, the default file location is *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness\Wire Library*
- For Microsoft Windows Vista, the default file location is *Public\Public Documents\Inventor <version>\Tutorial Files\Cable & Harness\Wire Library*

3 Enter **MyWires** in the File name field and click **Save**.

4 In the Configuration File area, browse to **WireLibrary.cfg** in the same directory.

The configuration file specifies which properties to export.

5 On the Export Library Wires dialog box, click **OK** .

6 On the confirmation dialog box, click **Close**.

7 On the Cable & Harness Library dialog box, click **Close**.

8 Return to the top-level assembly then save the assembly.

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Summary

In this tutorial, you learned to:

- Locate the cable and harness library.
- Create library definitions.
- Copy library definitions.
- Modify library definitions.
- Delete library definitions.
- Import and export library data.

Use your saved file from this tutorial to route wires and cables in the next tutorial.

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Create Wires and Cables 10

About this tutorial

Insert individual wires or cables.

Category	**Routed Systems**
Time Required	30 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial. WireListImport.csv WireListConfiguration.cfg

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

After connectors are placed and the harness assembly file is created, you can insert individual wires or cables.

Wires are inserted as occurrence wires. For cables, you select the pins for each wire or conductor in the cable. Wire and cable properties, such as color, diameter, and minimum bend radius are inherited from the library definition.

Objectives

- Insert wires and cables.
- Move wires and cables.
- Delete wires and cables.

- Replace wires.
- Assign virtual parts.
- Import harness data.
- Add wire points.
- Redefine and move wire point.
- Delete wire point.
- Set Occurrence properties.
- Change wire and cable displays.

Prerequisites

- Perform the previous tutorial, Cable and Harness Library, and save the tutorial file for use in this tutorial.
- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic "Fundamentals" and the New User tutorials for further information.

Navigation Tips

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

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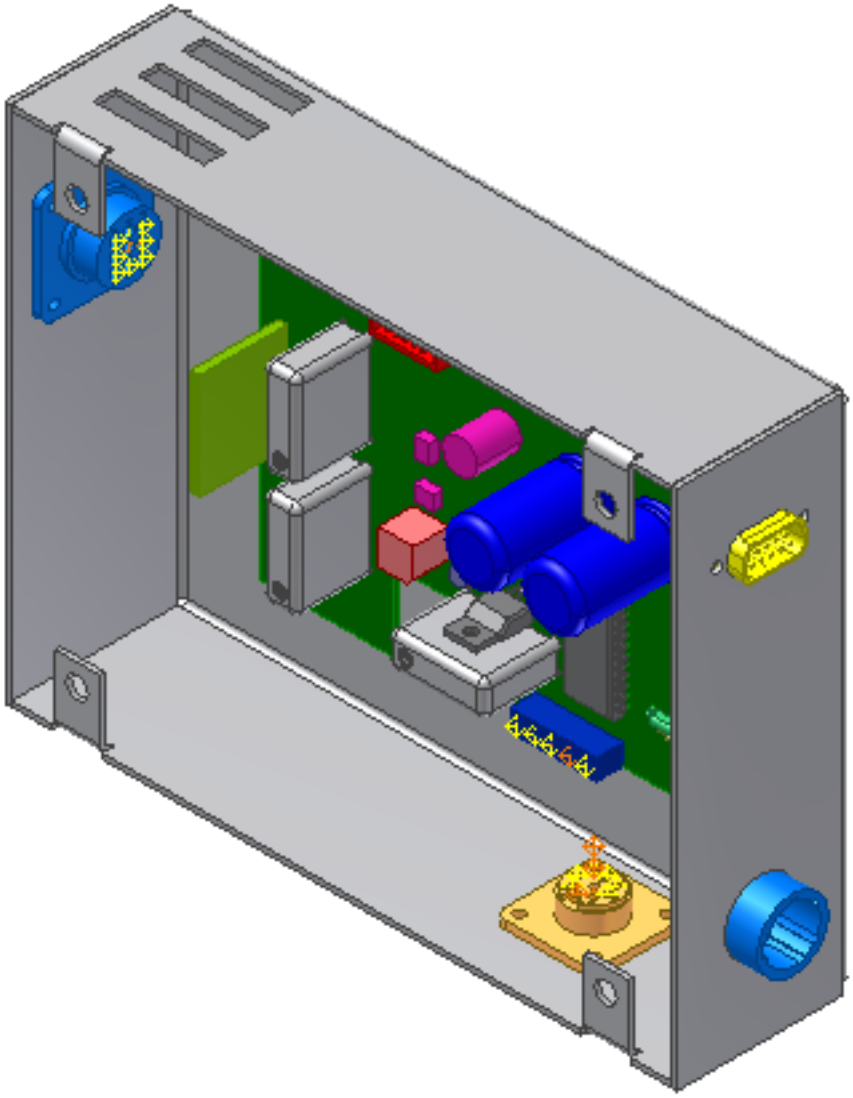
Set Modeling and Curvature Behavior

Before you begin creating harness objects, you set default configuration options on the Harness Settings dialog box.

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.



2 In the browser, double-click **Harness Assembly1** to edit it.

3 Right-click **Harness Assembly1**, and select **Harness Settings**.

4 On the **General** tab, ensure **Use Surfaces for All Segments and Wires** is selected. Use this setting for faster processing times while creating and editing harness assemblies.

5 Click the **Wires/Cables** tab.

6  Under Natural Curvature click **Without Natural Curvature**. This setting controls whether wires exit a connector tangent to the connector or straight from point to point.

7 If desired, click the other tabs and examine any settings of interest.

8 Click **OK**.

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Insert a Wire Manually

Provide a wire ID, select the wire to connect, and then select the pins for each end of the wire.

1  On the ribbon, click **Cable and Harness tab ➤ Create panel ➤ Create Wire** .

2 On the Create Wire dialog box, accept the default Wire ID (**Wire1**).

The default Wire ID is “Wire” plus an incremental number starting with 1.

3 From the list of wires, select the wire type to insert. Use the arrows to select the following category and wire name from the lists.

Category: Generic

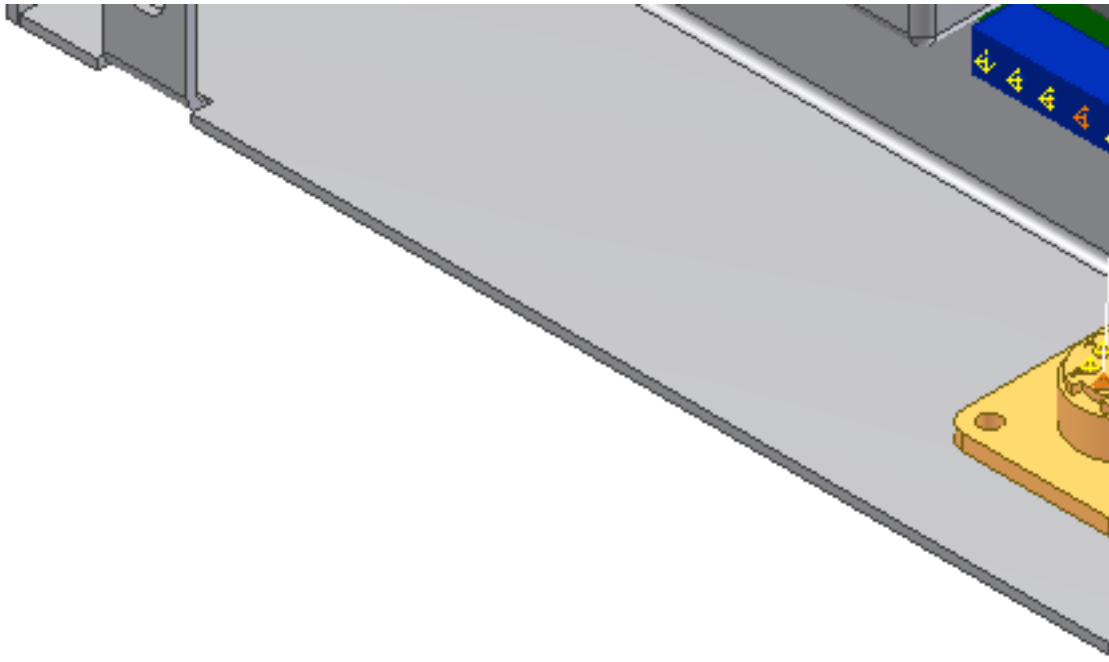
Name: 22AWG-GRN

4 Verify that the **Pin 1** button is depressed. If it is not, click the button.

5 In the graphics window, move the cursor over the pins in part 360124. As you select points for a wire, a tooltip displays the RefDes and pin number. When the tooltip displays as **U3 Pin1**, click the work point representing the start pin for the wire.

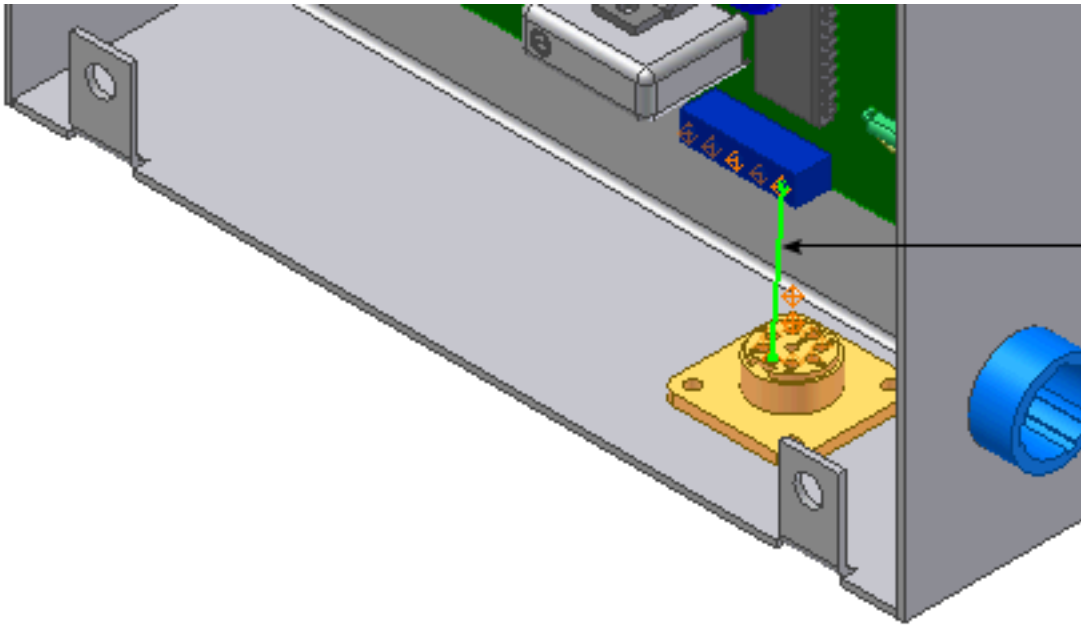
6 Pause the cursor over the first pin on part LTP.

As you select the second pin for a wire, a preview wire is displayed along with the tooltip.



7 When the tooltip is displayed as **J12 Pin 1**, click the point to select the second pin.

8 On the dialog box, click **OK** to create the wire.



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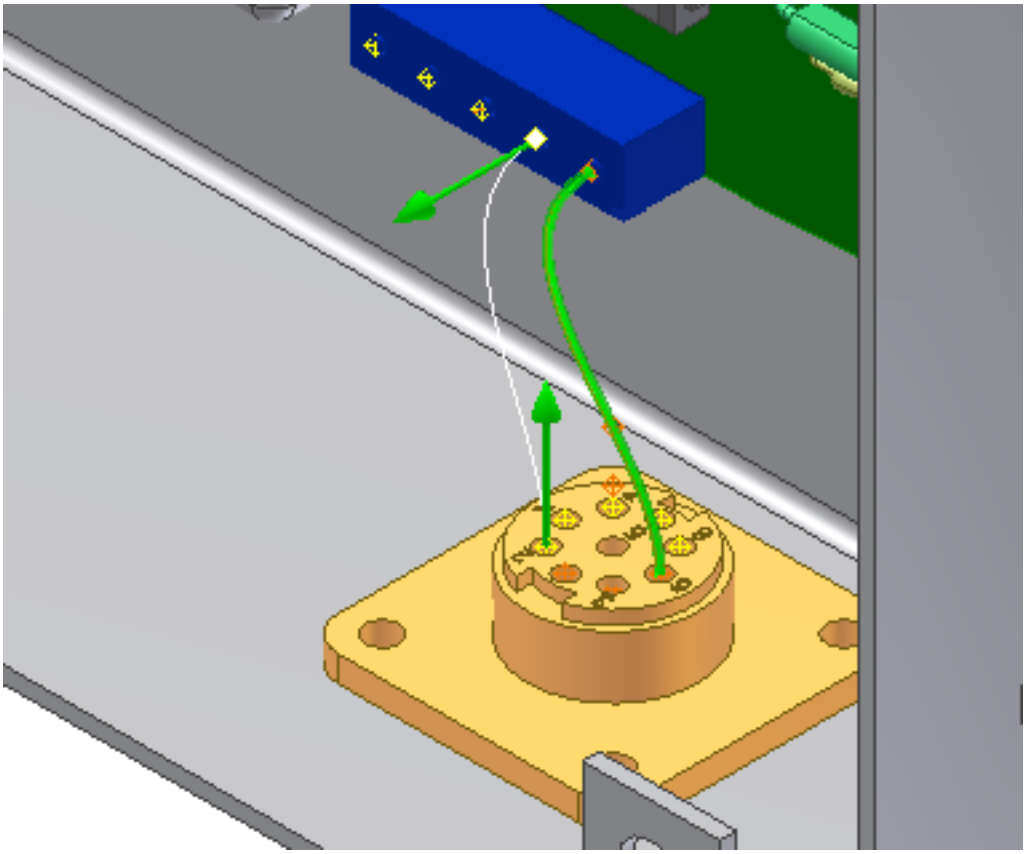
Insert Another Wire

Change the curvature setting back to With Natural Curvature then add another wire between the connectors.

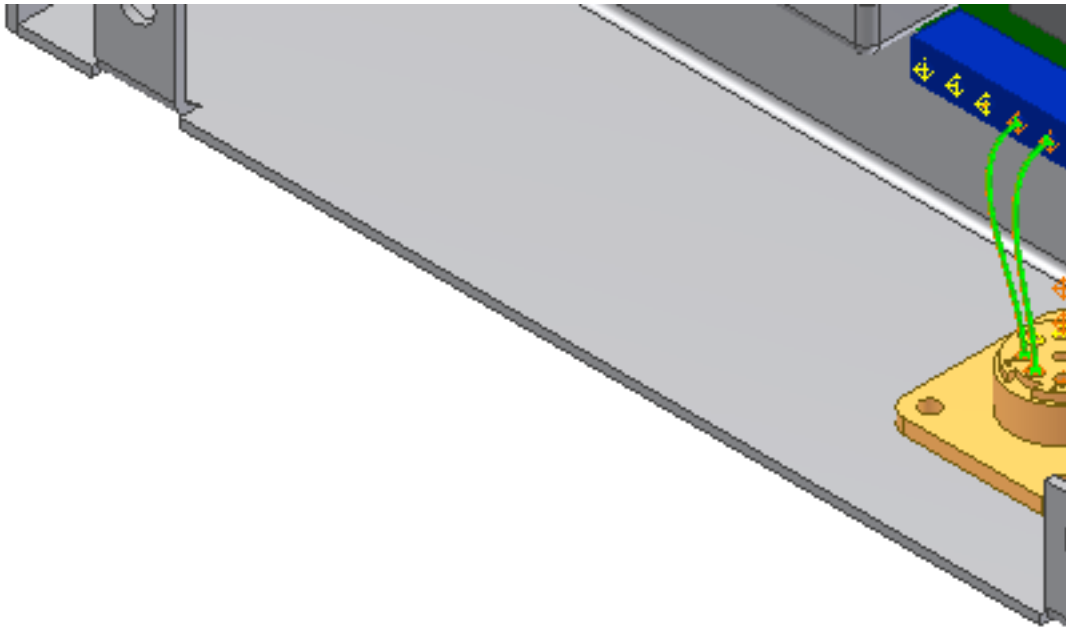
1  Before adding the second wire, use the Harness Settings command to change the natural curvature setting for wires and cables back to **With Natural Curvature**.

2 Add one more wire connecting **U3 Pin 2** to **J12 Pin 2**.

As you select each point for the wire, the outward direction arrow along with the preview wire and prompt display.



3 Right-click, and then select **Apply** from the context menu.



4 Click **Cancel**.

5 In the browser, expand the **Harness Assembly1** and the **Wires** folder to view the added wires in the hierarchy.

6 Return to the top-level assembly.

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Insert Cables Manually

Add a second harness assembly, and then add cable wires. You can document the cable harness in a separate nailboard drawing.

1 On the ribbon, click **Environments tab ➤ Begin panel ➤ Cable and Harness**.

2 Enter **Harness Assembly2.iam** for the name and accept the default file location.

3 Add the following reference designators for the connectors. In the browser, right-click the connector and select Harness Properties to set the RefDes.

For part 360575:1 add RefDes **U7**.

For part 360575:2 add RefDes **U8**.

4 On the ribbon, click **Cable and Harness tab ➤ Create panel ➤ Create Cable**.

5 On the Create Cable dialog box, accept the default Cable ID (Cable1).

By default, the Cable ID is “Cable” plus an incremental number starting with 1.

6 From the list of cables, select the cable to insert. Select the category and cable name shown.

Category: Belden

Name: 8441

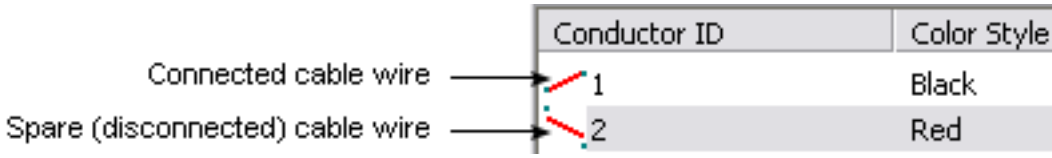
This cable contains two wires to connect in the assembly.

7 Verify that the first cable wire and the **Pin 1** button are selected.

8 Rotate the view so that the pins on 36575:2 are visible. Select **U8 Pin1**.

Cable wires are automatically assigned Wire IDs as they are connected. By default, the Wire ID is the Cable ID plus the Conductor ID.

9 Connect the other end of the cable wire to part 36575:1. When the tooltip displays as **U7 Pin 1**, click the work point. The status of the cable wire in the list changes to connected. The system automatically advances to the next cable wire.



10 Connect the second cable wire from **U8 Pin 2** on part 36575:2 to **U7 Pin 2** on part 36575:1.

11 Click **OK**.

12 In the browser, expand the harness assembly and the Cables folder to view the added cable and cable wires in the hierarchy.

   Cable wires are named with the Cable ID followed by the conductor ID, and the conductor ID in parentheses.

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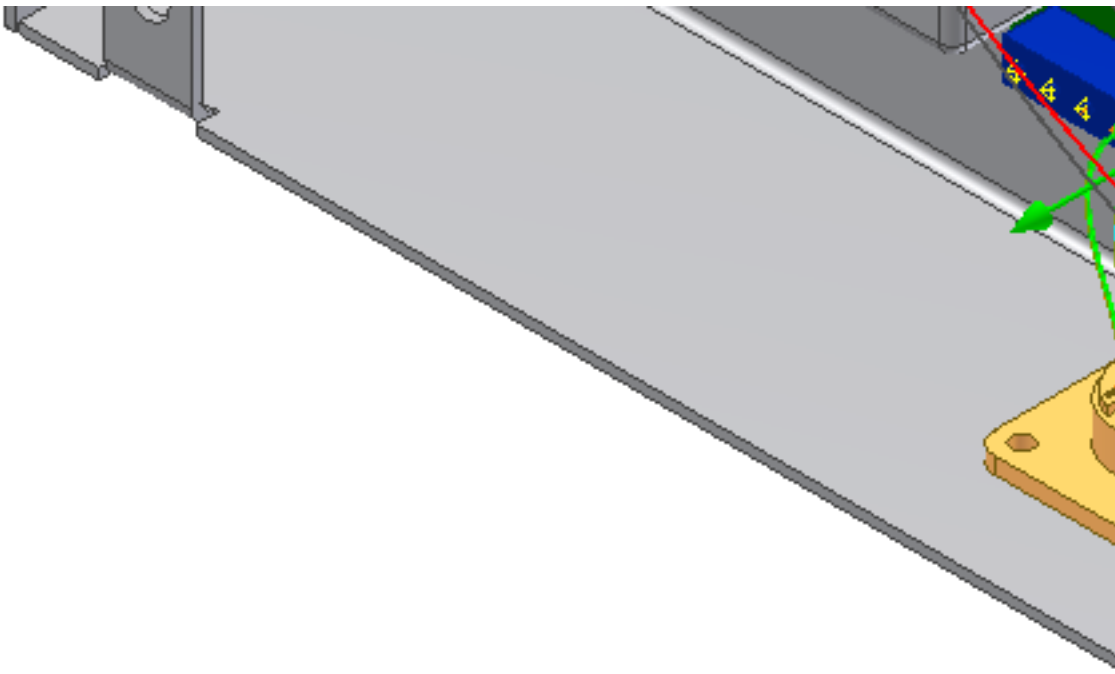
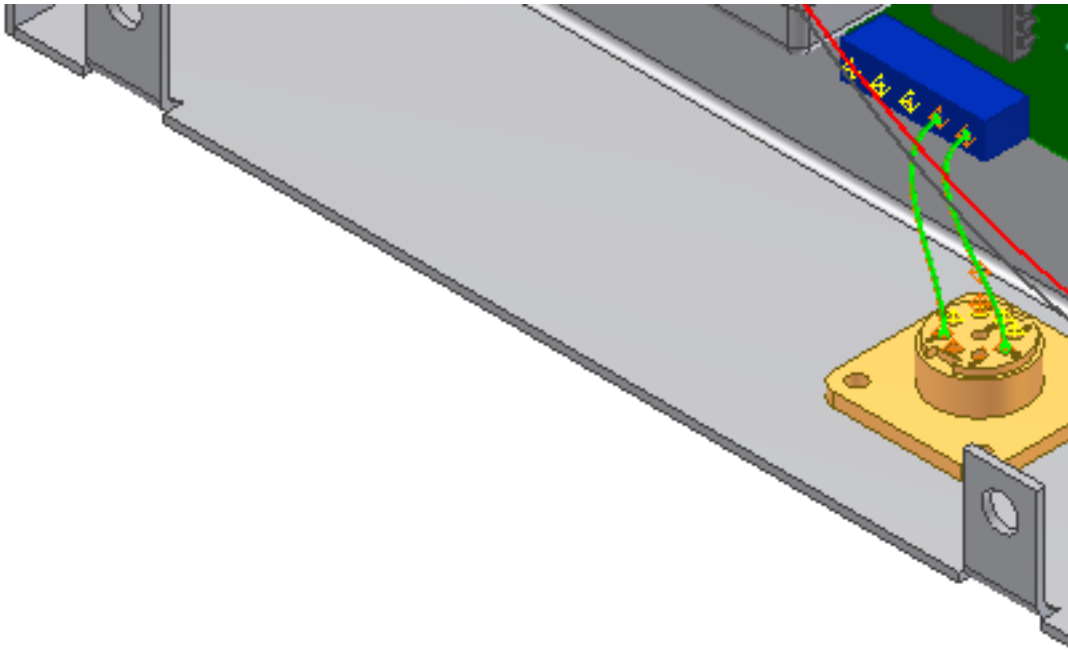
Move Wires

Move Pin 2 of the wire you added previously to another pin on the same connector.

1 Activate **Harness Assembly1**.
2 In the browser or graphics window, right-click **Wire1**, and then select **Edit Wire** from the context menu.
3 On the Edit Wire dialog box, verify the **Pin 1** button is selected.
4 In the graphics window, move the cursor over the pins to preview the wire connection possibilities before making a selection.
5 When the tooltip displays as **U3 Pin 6**, click the pin, as shown in the following illustration.



6 Right-click and select **Apply** to create the wire in the new location.
The wire is redrawn using the new pin, and the wire length updates.



7 Click **Cancel**.

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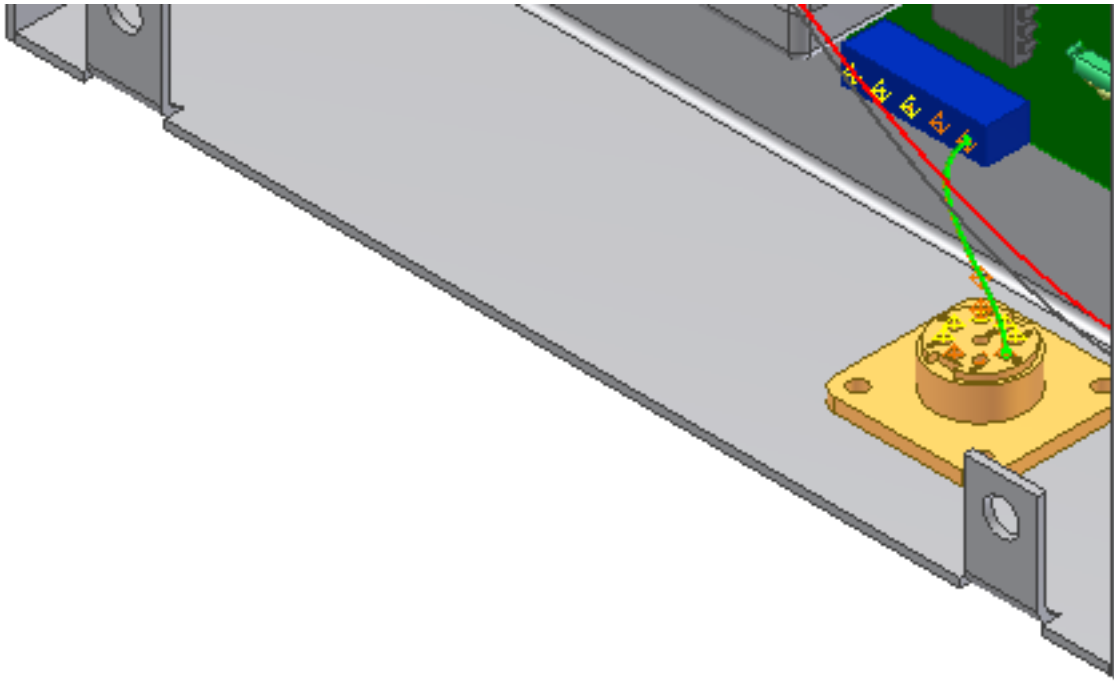
Delete Wires

Remove a wire from the assembly using the Delete command.

1 In the browser or graphics window, select **Wire2**.
2 Right-click and select **Delete** from the context menu.
The wire is deleted.



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Remove Cables and Cable Wires

You can disconnect a cable wire to remove it from your design:

1. In the browser, activate **Harness Assembly2**, and then expand the harness part to see the Cables folder.
2. Expand the Cables folder and the Cable, and then select **Cable1:1(1)**.
3. Right-click and select **Edit Cable**.
4. On the Edit Cable dialog box, select wire 1 to disconnect.
5. Click **Disconnect**, and then click **Apply**.

   The cable wire termination status updates, and the cable wire is removed from the graphics window.
6. Reconnect the wire to Pin 1 on both connectors as you did earlier, and then click **OK**.

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Replace Wires

You can edit a wire to change the wire definition used. This changes the wire type, appearance, and so on.

1. Activate **Harness Assembly1**.
2. In the browser or graphics window, select **Wire1**, right-click, and then select **Edit Wire**.
3. On the Edit Wire dialog box, select:

   Category: Belden

   Name: 9916-VIO
4. To redraw the wire using the new definition, click **Apply**, and then **Cancel**.

   The wire in the graphics window is redrawn in violet.
5. Return the wire to the original, green wire definition. On the Edit Wire dialog box, select:

   Category: Generic

   Name: 22AWG-GRN
6. Click **OK**.

   NOTE Cables cannot be replaced. Delete them and add them again with the appropriate cable chosen from the library list.

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Assign Virtual Parts

Use the Property dialog box to assign the Wire ID label you created previously to a wire pin.

1. Expand the harness part, then the Wires folder, then Wire 1 in the browser.
2. Right-click **Pin 6 (6 RefDes U3)** and select **Wire Pin Properties** from the context menu.

3 On the Wire Pin Properties dialog box, click the **Virtual Parts** tab and set the following:

Type: Label

Category: (none)

Name: Wire ID Label

4 Click **Add**, and then **OK**.

NOTE You can also assign virtual parts to various objects in a harness assembly using the **Assign Virtual Parts** command on the Cable and Harness tab.

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Import Harness Data

Import seven discrete wires into the harness assembly using import files that are already created. To view the file contents, open the file in a text editor.

1  With Harness Assembly1 active, click **Cable and Harness tab ➤ Manage panel ➤ Import Harness Data**.

2 Click **Browse** to locate and open the import files listed in the following list. They are located in the *Tutorial Files\Cable & Harness\Wire List Import* folder. You may need to set the Files of type to **Wire List Data File**.

Data File (.xml or .csv): WireListImport.csv

Configuration File (.cfg): WireListConfiguration.cfg

3 Click **OK** to add the wires to the active harness assembly.

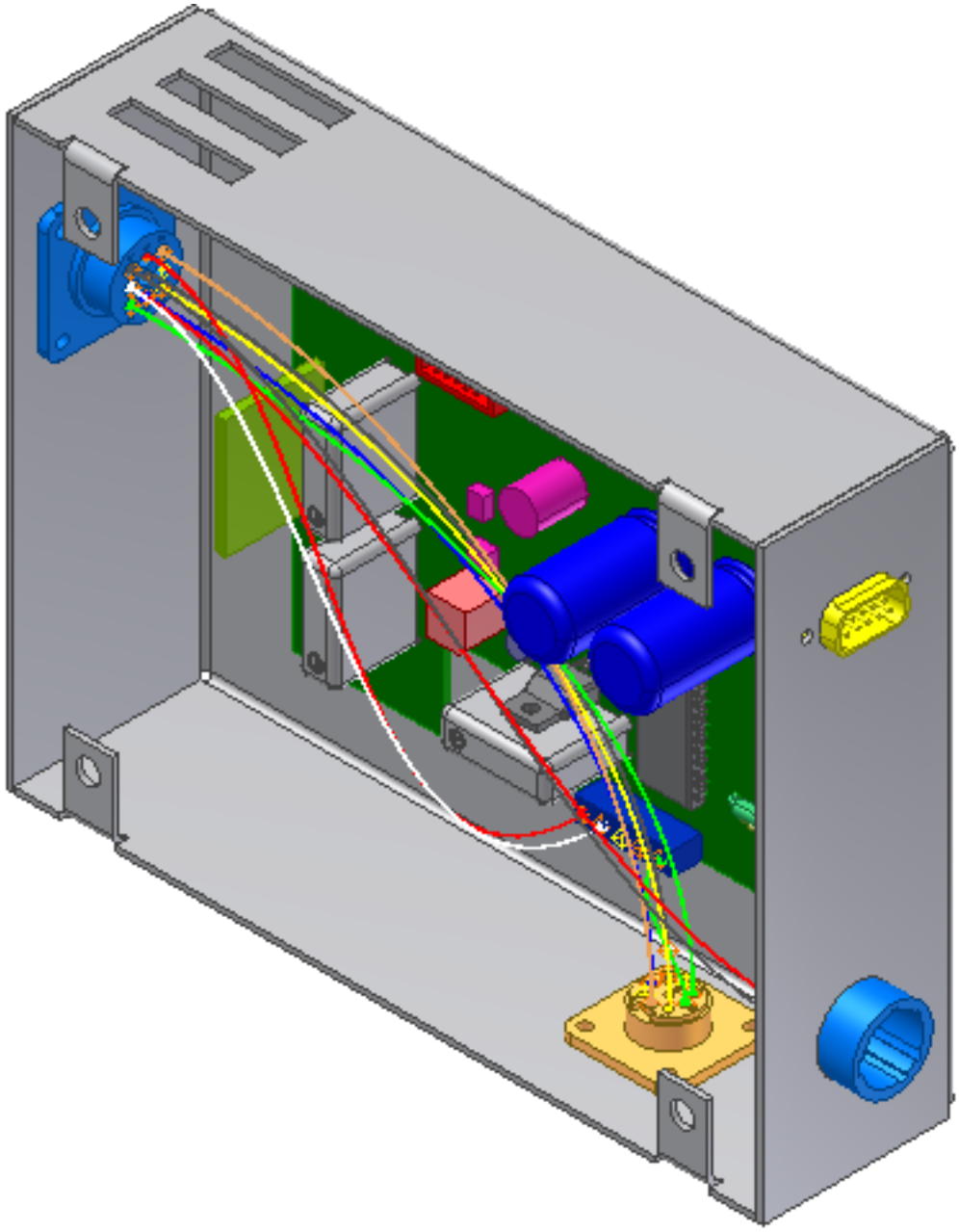
The Imported Harness Data dialog box is displayed and lists the items that will be imported. For this example, there are no errors. If errors exist, you can right click the items in question for more information on the issue and how to fix it.

Optionally, expand the folders on the Import Harness Data dialog box to view details on the import.

4 Click **OK** in the Imported Harness Data dialog box.

A message box displays the status of the import. The message indicates that seven wires imported successfully. The log file contains details on the import, including any associated errors or warnings.

5 On the message dialog box, click **Close**. The wires are added to the harness assembly.



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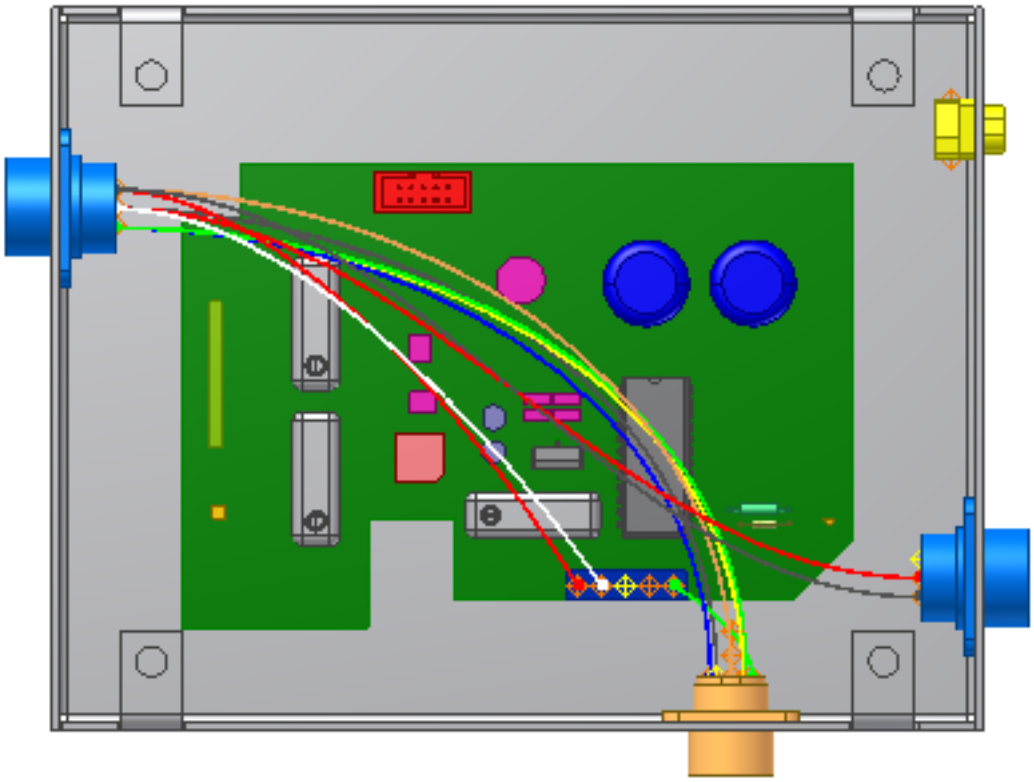
Add Wire Points

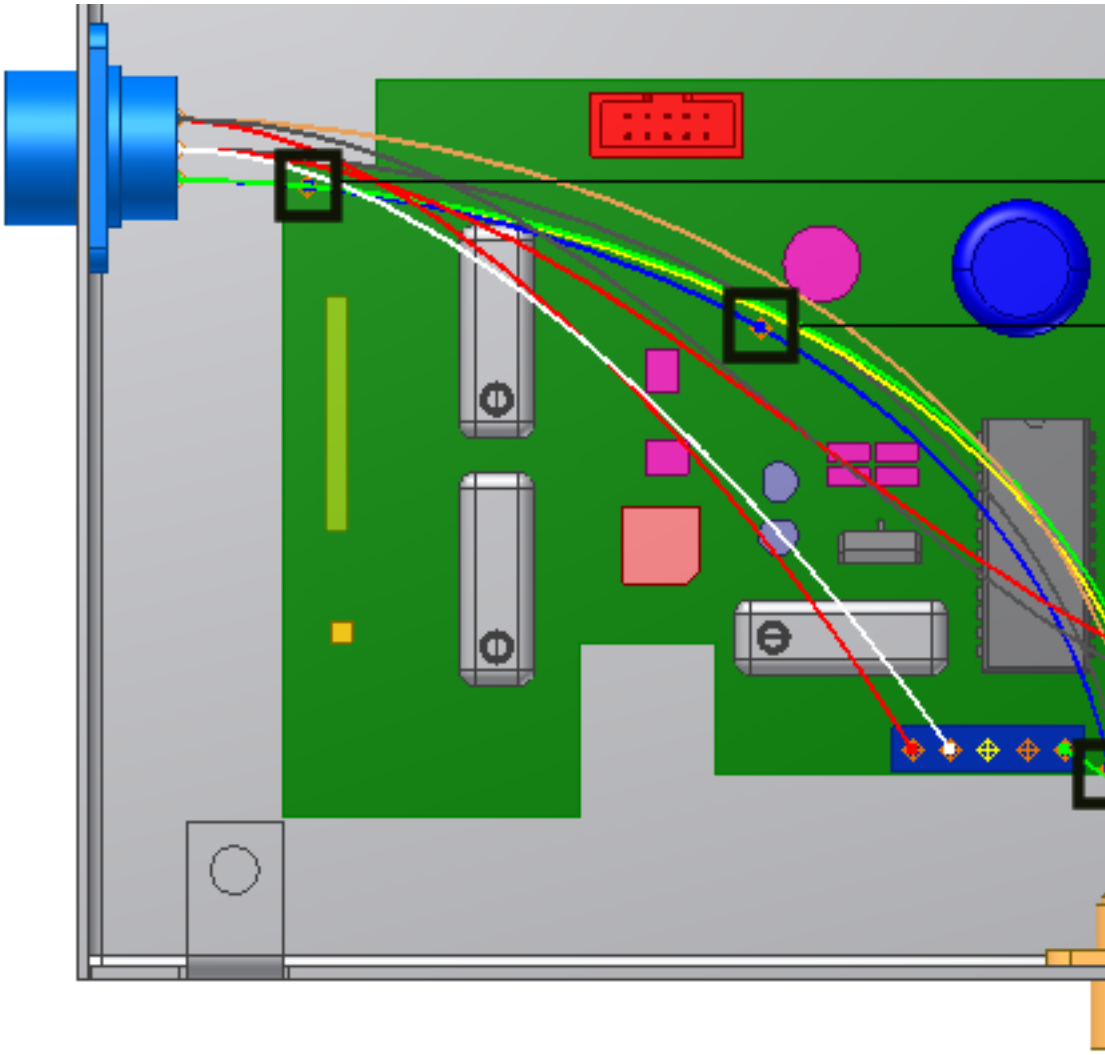
You can add points to wires. The points can be moved to route the wires around existing components. If you plan to add segments to the harness assembly, the segments will control the position of the wires and additional wire points are not necessary.

1 Rotate the view until it looks like the following image.



2 In the browser, right-click wire **2207**, and then select **Add Points** on the context menu.

3 Move the cursor over the wire in the graphics window, and then click the wire to create the three points in the order shown.



A grounded work point is created at each pick location, and the wire recomputes, changing shape slightly after each point is added.

4 Right-click and select **Finish**.

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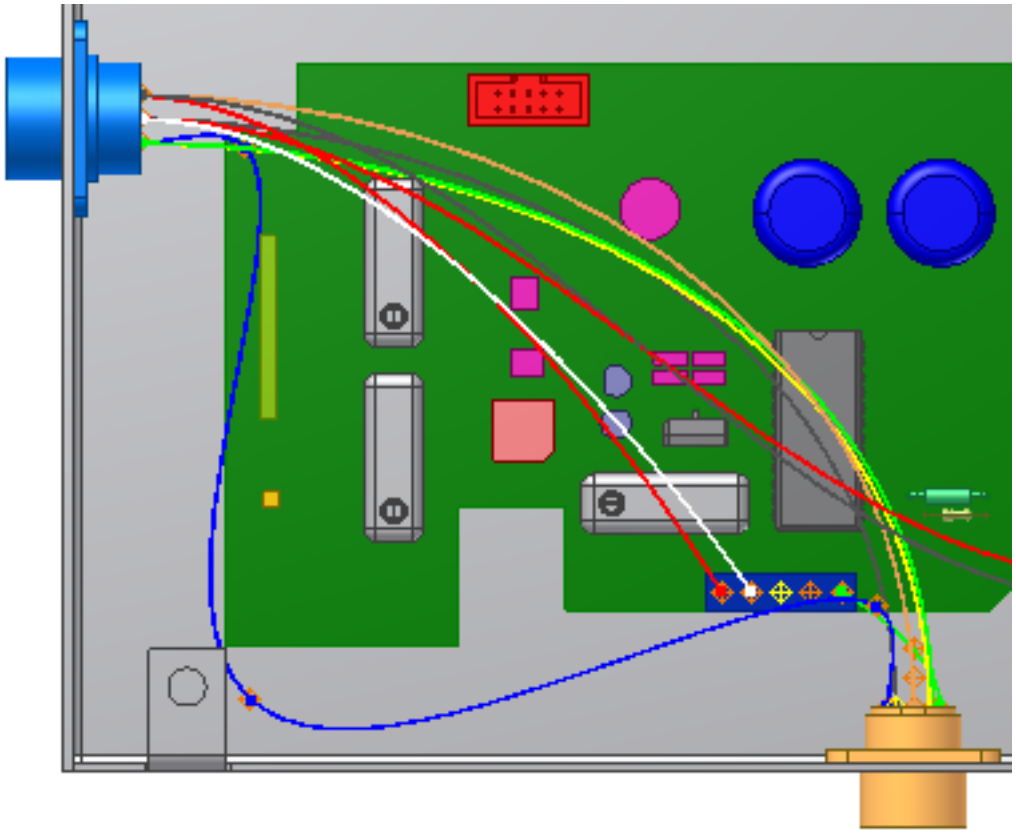
Redefine and Move Wire Points

Redefine the points to an arbitrary location or feature to achieve the shape you need.

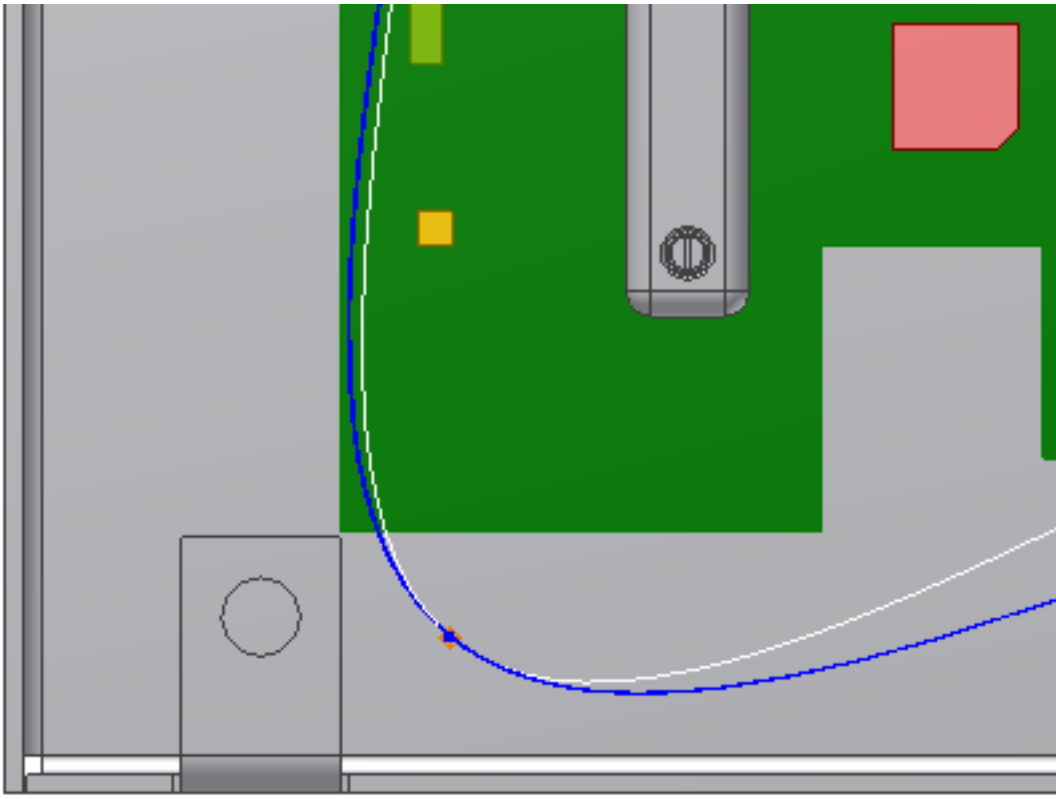
1. In the browser, expand wire 2207, right-click the second work point you created and then select **Redefine Point** from the context menu.
2. In the graphics window, pause the cursor near the tab on the enclosure as shown. A tooltip shows the offset value for the point.
3. In the graphics window, right-click (not on the point), select **Edit Offset** from the context menu. Then enter **.200** and click **OK** to change the offset .
4. Click the point in the location shown to create a point that is offset from the face of the enclosure. The wire recomputes.



5 In the browser or graphics window, right-click the third work point created, and then select **3D Move/Rotate**.

6 Select or drag a triad segment to move the point to the location shown.

The position of the wire is previewed and any bend radius violations noted before applying the move.



7 Click **OK.** The wire recomputes to the new location.

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Delete Wire Work Points

You can delete individual wire work points or all points at once.

1 In the browser, right-click the first work point created, and then select **Delete Point** on the context menu. The wire recomputes.

2 To remove all points from the wire, right-click **Wire 2207**, and then select **Delete All Points** on the context menu.

   The wire returns to its original state and the length of the wire is recomputed.

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Override Library-level Properties

Change the value of the custom Volt property for a wire.

1 In the browser or graphics window, expand the Harness Assembly1 part and the Wires folder.
2 Right-click **Wire1**, and select **Harness Properties** on the context menu.
3 Click the **Custom** tab. Notice that the custom properties added at the library level are displayed with a dark background.
4 Click the **Volt** property in the table.
5 Select the property Value of 300 volts and change it to **200 volts**.
6 Click **Modify** to override the property in the table, and then click **Apply**.

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Restore Library-Level Properties

Restore the library-level property value for the custom Volt property with the value override.

1 With the Wire Properties dialog box open, in the custom property table, click the **Volt** row to select it.

   The library-level property name and value appear in the bottom left of the dialog box.
2 Right-click anywhere in the row, and then select **Restore** on the context menu.

   The wire library replaces the occurrence value with the library value.
3 Click **OK**.

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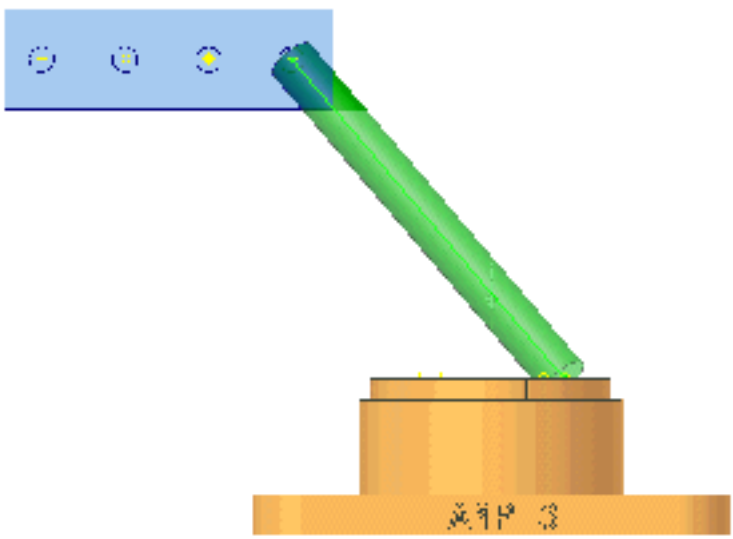
Change Wire and Cable Displays

Wires and cables display as rendered or centerline (default). Use centerline for optimal performance while creating and routing wires and cables.



rendered display

1. In the browser or graphics window, select any wire from the list of wires.
2. Right-click and select **Display as Rendered** on the context menu.
3. To change the display for all wires, select the Wires folder in the browser, right-click, and select **Display All as Rendered** on the context menu.

   All wires in the selected harness assembly are displayed as rendered.
4.  On the Ribbon click **Cable and Harness tab ➤ Visibility panel ➤ Centerline Display** to return the wires to the default centerline display mode.

5 Return to the main assembly and save all files. This assembly is used in the next exercise.

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Summary

In this tutorial, you learned how to:

- Insert wires and cables.
- Move wires and cables.
- Delete wires and cables.
- Replace wires.
- Assign virtual parts.
- Import harness data.
- Add wire points.
- Redefine and move wire point.
- Delete wire point.
- Set Occurrence properties.
- Change wire and cable displays.

Use your saved file from this tutorial to work with segments in the next tutorial.

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Create Segments

11

About this tutorial

Create segments and add segment branches.

Category	**Routed Systems**
Time Required	30 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial

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

You can define segment paths as close to the appropriate results as possible, and then adjust them later. You can also develop them using work features you set when planning the segment path.

Objectives

- Create segments.
- Add segment branches.
- Redefine or move segment work points.
- Insert segment work points.

Prerequisites

- Perform the previous tutorial, Create Cables and Wires, and save the tutorial file for use in this tutorial.

- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic “Fundamentals” and the New User tutorials for further information.

Navigation Tips

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

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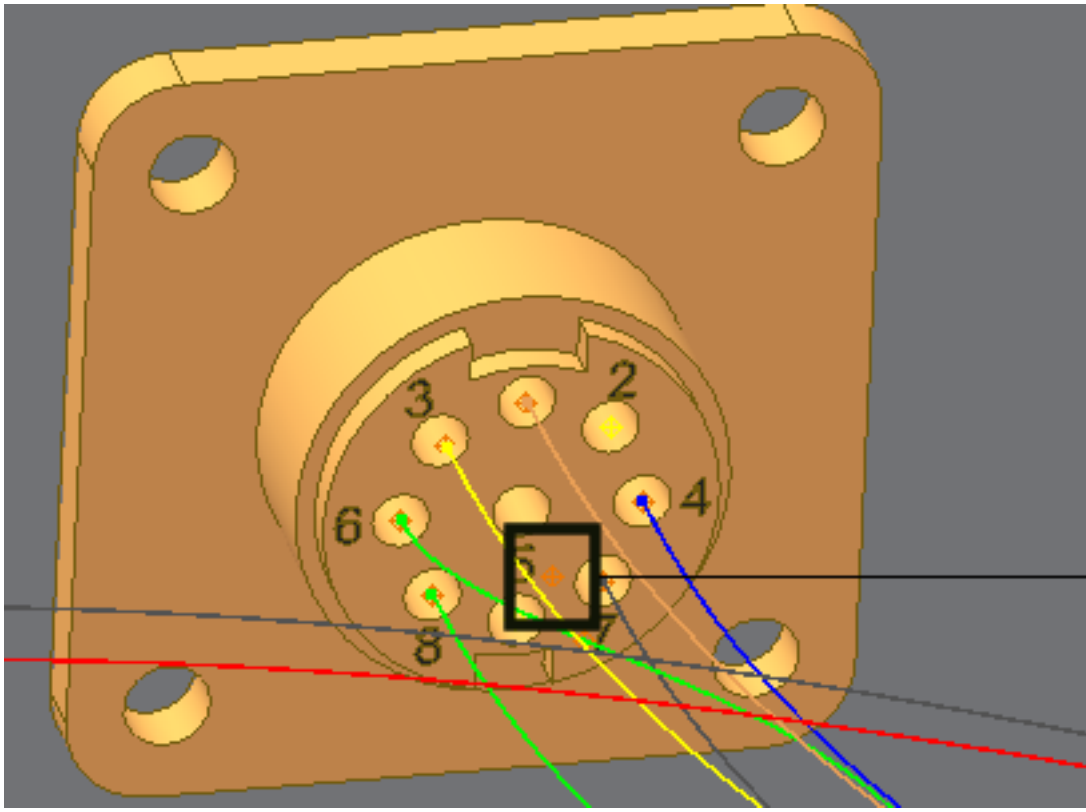
Create Segments

Define a harness segment between two circular connectors (360124:1 and 360575:1) and around the PCB board in the enclosure. Adjust the offset while you work.

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.
2. In the browser, double-click **Harness Assembly1** to activate it for editing.
3.  On the ribbon, click **Cable and Harness tab ➤ Create panel ➤ Create Segment**.
4. Rotate and zoom in to component 360124 then select the work point shown in the following image.



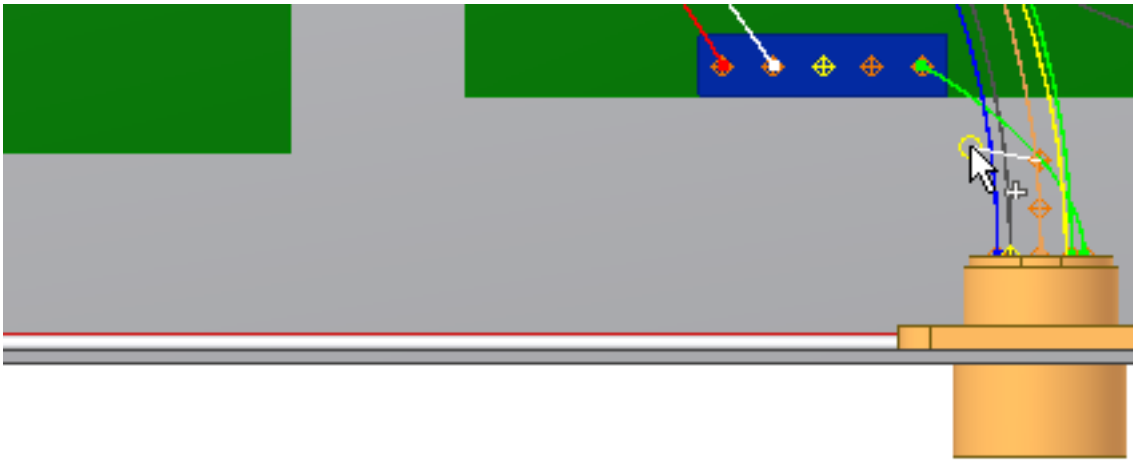
A segment work point is created at the selected point.

5 Move the cursor over the bottom of the enclosure until the tooltip and offset symbol appear. The tooltip displays the offset value as 0.100 in, and the symbol shows the actual offset distance as a guide for your selections.

6 Before you select the second point, increase the offset to prepare for a branch in the harness. To increase the offset, right-click, select **Edit Offset** from the context menu, and then enter **0.270**.

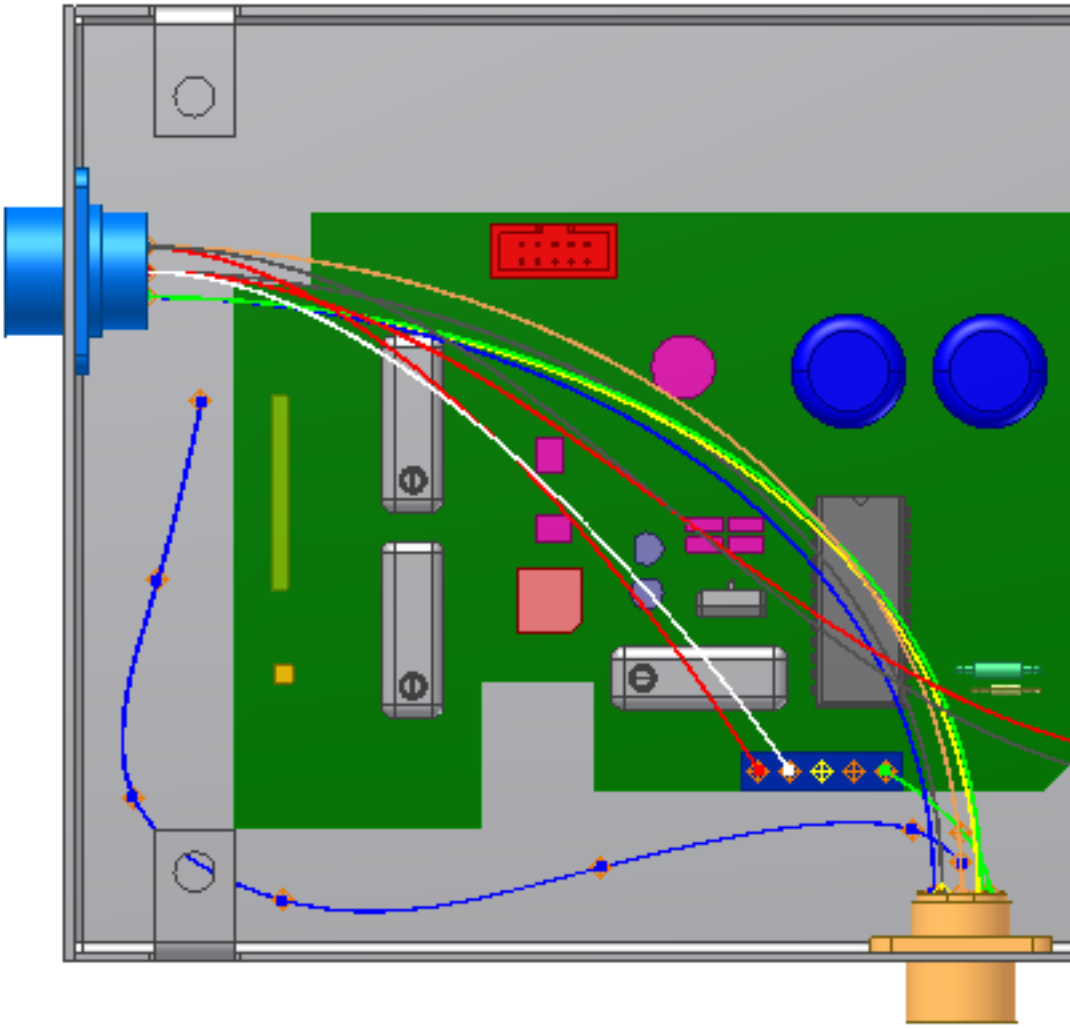
7 For the second point orient and zoom the view as shown, and then select a point based on the bottom of the enclosure.

As you make your selections, the system displays a straight line from the selected point to the current cursor position.



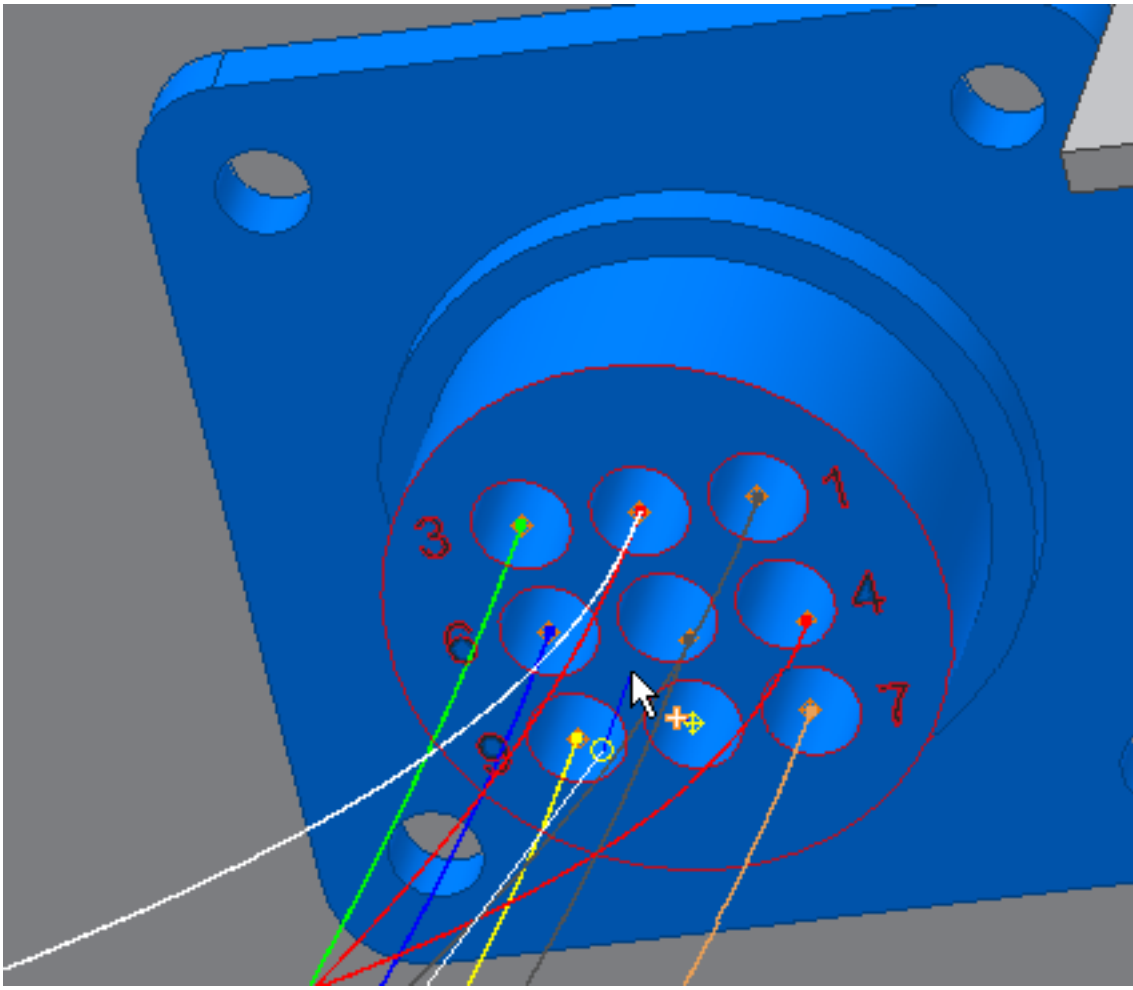
8 Continue selecting points around the printed circuit board (PCB) and toward the other connector to create the segment shape.

The following image shows several additional point selections. Exact placement of the work points is not important. You can adjust the work points after the segment is complete.



9 Right-click, select **Edit Offset**, and then enter **0.200** to change the offset value for the last point.

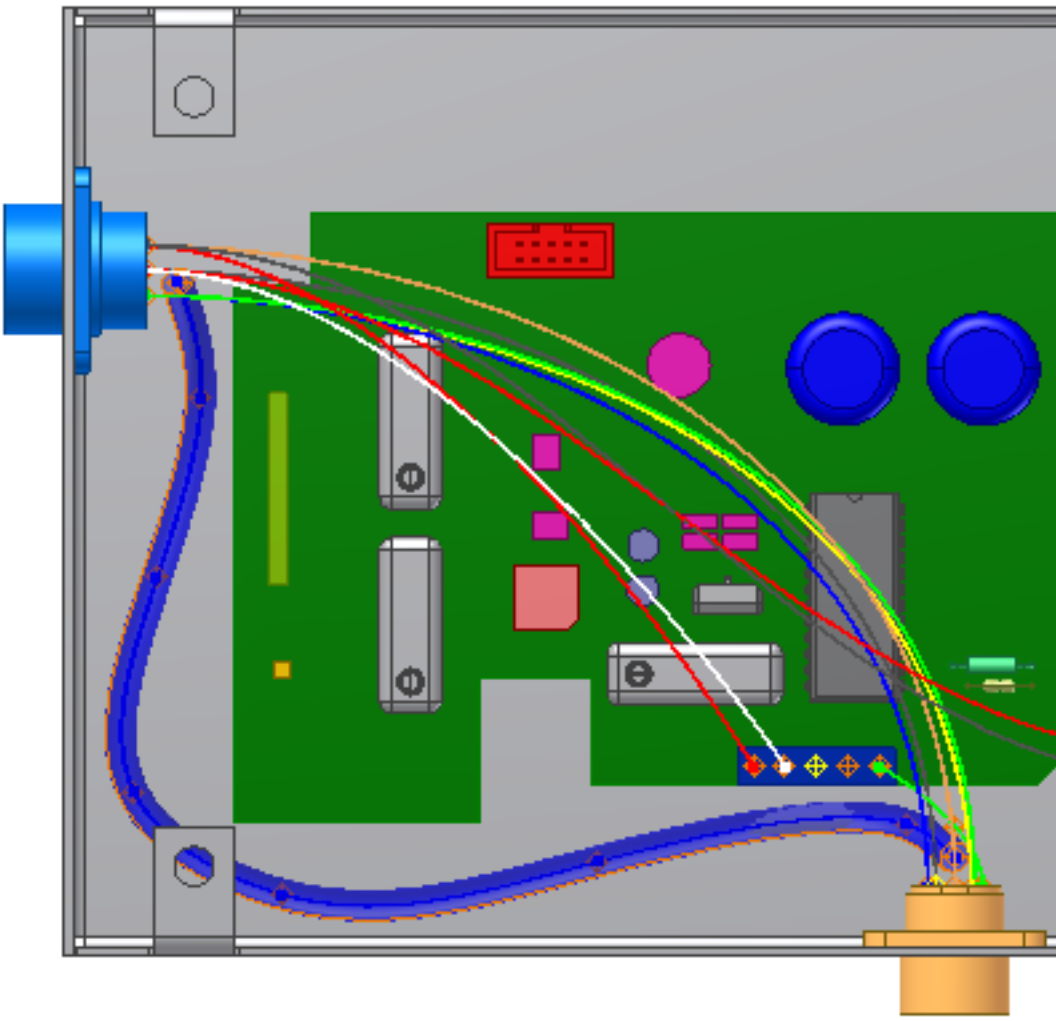
10 Click to select the point as shown in the illustration.



11 Right-click and select **Finish** to complete the segment.

NOTE If an error message is displayed and the sweep cannot be generated, adjust the work points in areas of highest curvature to reduce the curve.

The system generates the segment, as shown in the following illustration.



12 Save the harness assembly.

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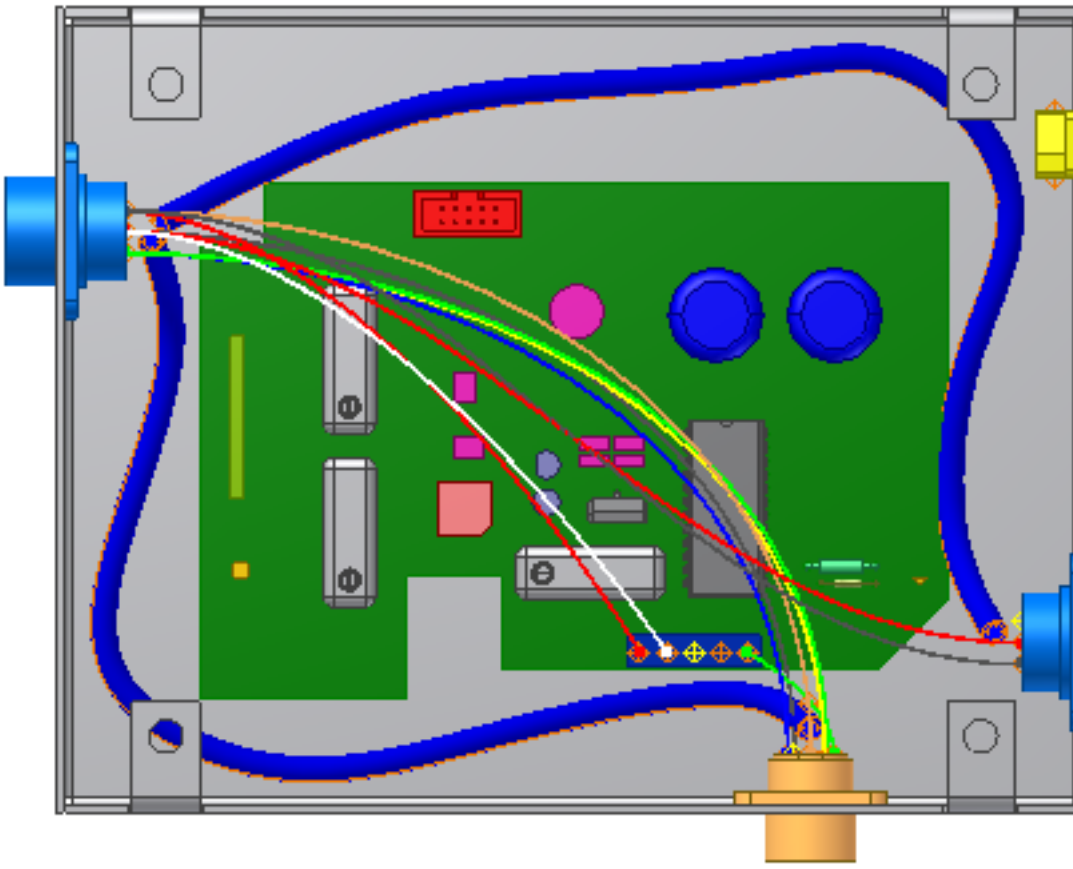
Create an additional segment

Define a second harness segment between two circular connectors (360575:2 and 360575:1) and around the PCB board on the opposite side of the first segment.

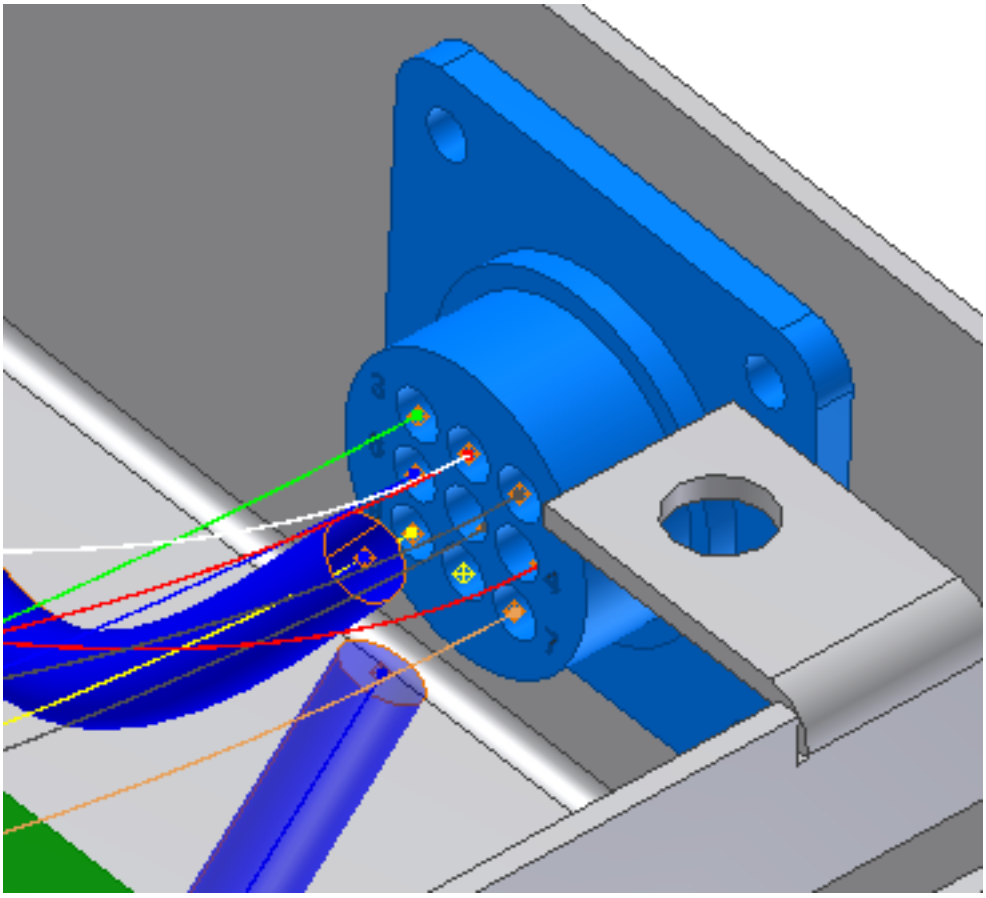
1. Activate **Harness Assembly2**.
2.  Click **Cable and Harness tab ➤ Create panel ➤ Create Segment**.
3. Right-click, select **Edit Offset** then change the offset to **0.200**.
4. Click the face of connector 360575:2 near pin 5.
5. Continue selecting points as shown to create a segment like the one shown in the following illustration.



6 Before you select the last point on the connector face, change the offset to **0.270**.

7 Click the bottom of the connector face near pins 7 and 8.

The new segment is positioned like the following illustration.



8 Right-click and select **Finish** to complete the segment.

9 Save the harness assembly.

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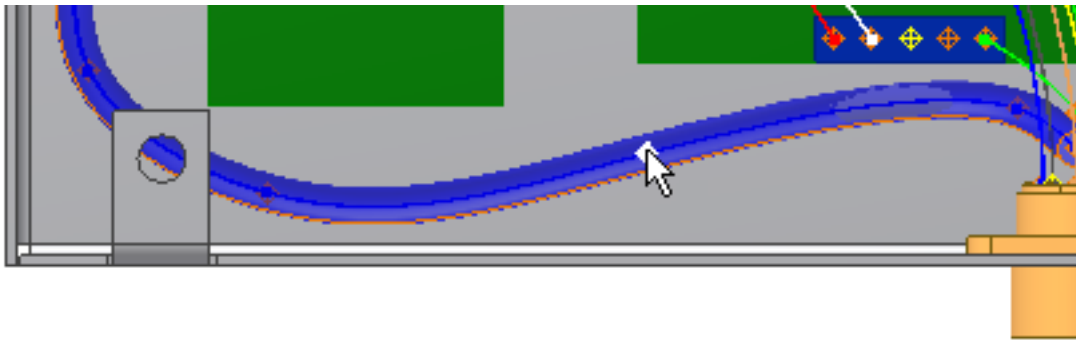
Add Segment Branches

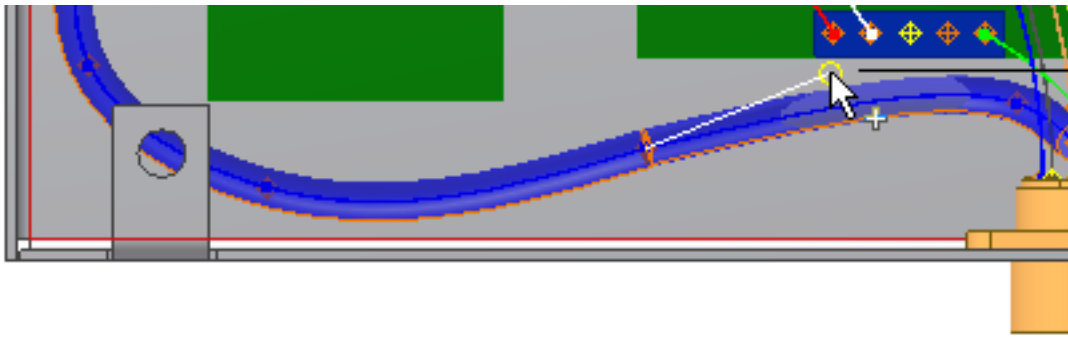
1 In the browser, double-click **Harness Assembly1**.

2  On the ribbon, click **Cable and Harness tab ➤ Create panel ➤ Create Segment**.

3 Pause the cursor over the harness segment to branch, and then click the segment work point shown as the branch start point.

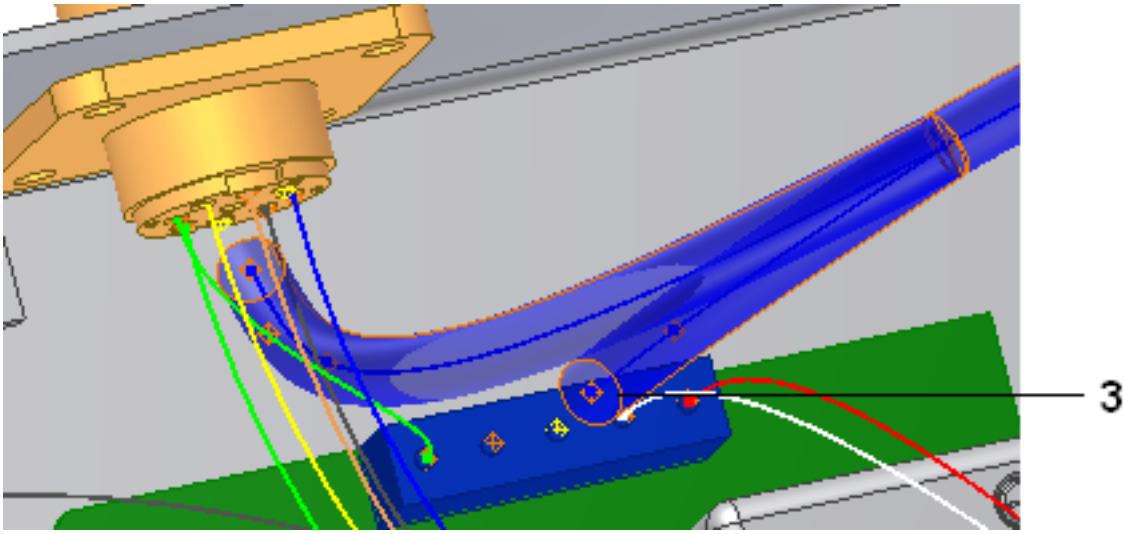


4 Change the offset to **0.470**, and then select one point (2) between the segment and the LTP:1 connector.



5 For the final point (3), change the offset to **0.270**, and then select the face of the connector LTP:1 near pin 3.

6 Right-click and select **Finish** to complete the branch with a shape like the one in the image.



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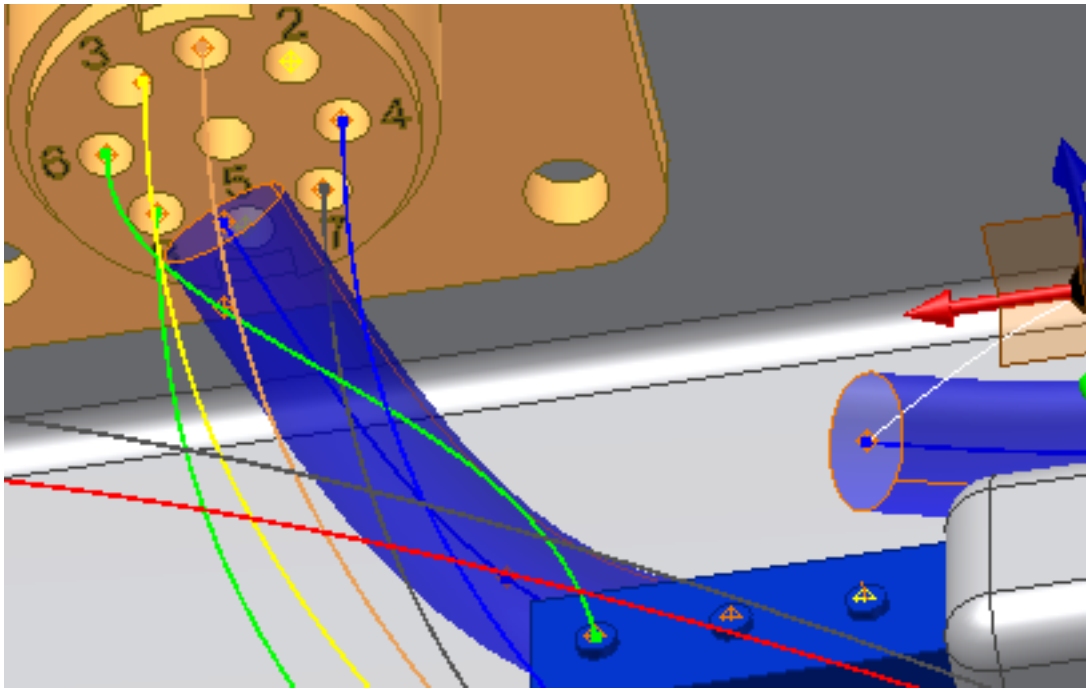
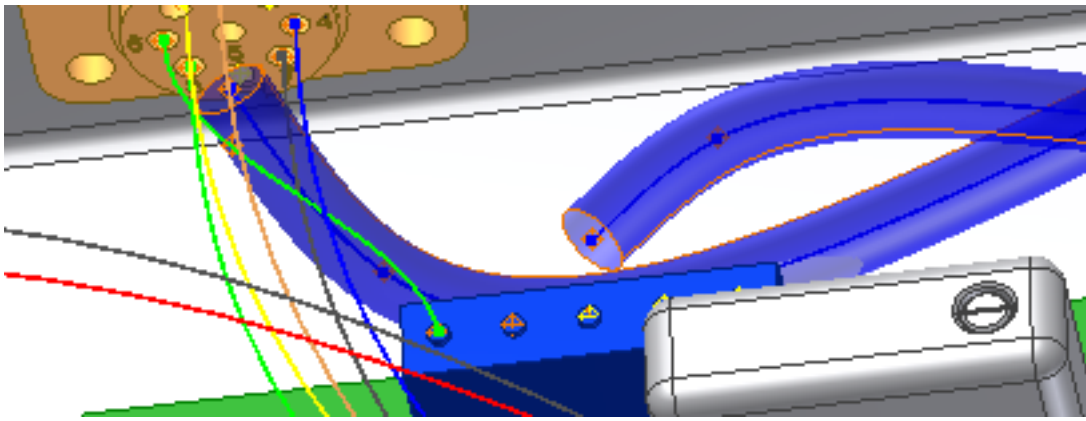
Redefine or Move Segment Work Points

Use **3D Move/Rotate** to adjust a work point for the segment branch.

1. To get a better view of your data, turn off the visibility for any geometry obstructing your view.
2. In the graphics window, right-click the second branch work point (2) as shown in the following illustration.
3. Right-click and select **3D/Move Rotate** from the context menu.
4. Drag the point vertically until the preview looks like the following shape, right-click, and select **Apply**.



5 Continue adjusting until you are satisfied with the shape, right-click, and select **OK**.



NOTE Once a segment is created, you can verify whether any points along the segment violate the bend radius value. In the browser, right-click the segment then select **Bend Radius ➤ Check** from the context menu. Verify that the bend radius multiplier is set before checking the points. Select "bend radius, check" in the Help index for more details on bend radius checking.

6 Save the harness assembly.

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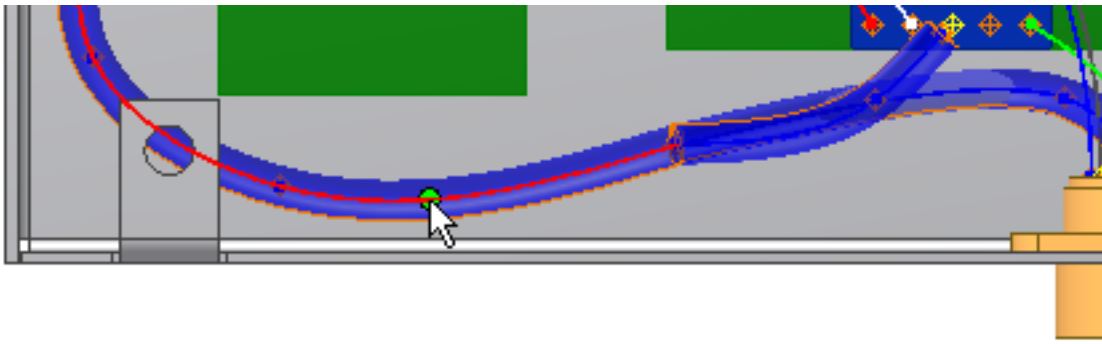
Insert Segment Work Points

Insert additional points for more control over the segment shape in certain areas.

1 In the browser or graphics window, right-click Segment2, and then select **Add Points** from the context menu.

2 Pause the cursor over the area of the segment shown in the illustration.

The path and work points highlight as you move the cursor over the segment.



3 Click the path to add the point.

The point is added and the segment recomputes.

4 Right-click and select **Finish**.

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Delete Harness Segment Work Points

Delete the point you created. It was only added to gain practice with the commands.

Delete a work point from a single segment

1. With the harness assembly still active, right-click the point you just added.
2. Select **Delete Point**.

   The point is removed and the segment recomputes.

   If the point is a common endpoint of two segments, they merge into a single segment. If the point is the common endpoint of three or more segments (branches), no merging occurs. Instead, each segment has a unique work point at its end, which you can manipulate independently.
3. Return to the top-level assembly and save all files. This assembly is used in the next tutorial.

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Summary

Replace with your summary. Follow your summary with a list of bullets that highlight each important point taught:

- Create segments.
- Add segment branches.
- Redefine or move segment work points.
- Insert segment work points.

Use your saved file from this tutorial to route wires and cables in the next tutorial.

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Route Wires and Cables 12

About this tutorial

Route and unroute wires and cables

Category	**Routed Systems**
Time Required	20 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial.

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

Various methods are available to place unrouted wires (including cable wires) into segments in the active harness. Unrouting removes the wires from all or selected segments.

Objectives

- Define manual routes.
- Route wires.
- Define automatic routes.
- View wire and cable paths.
- Unroute wires.
- Unroute cables.
- Unroute all wires or cables from all segments.

Prerequisites

- Perform the previous tutorial, Create Segments, and save that tutorial file for use in this tutorial.
- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic "Fundamentals" and the New User tutorials for further information.

Navigation Tips

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

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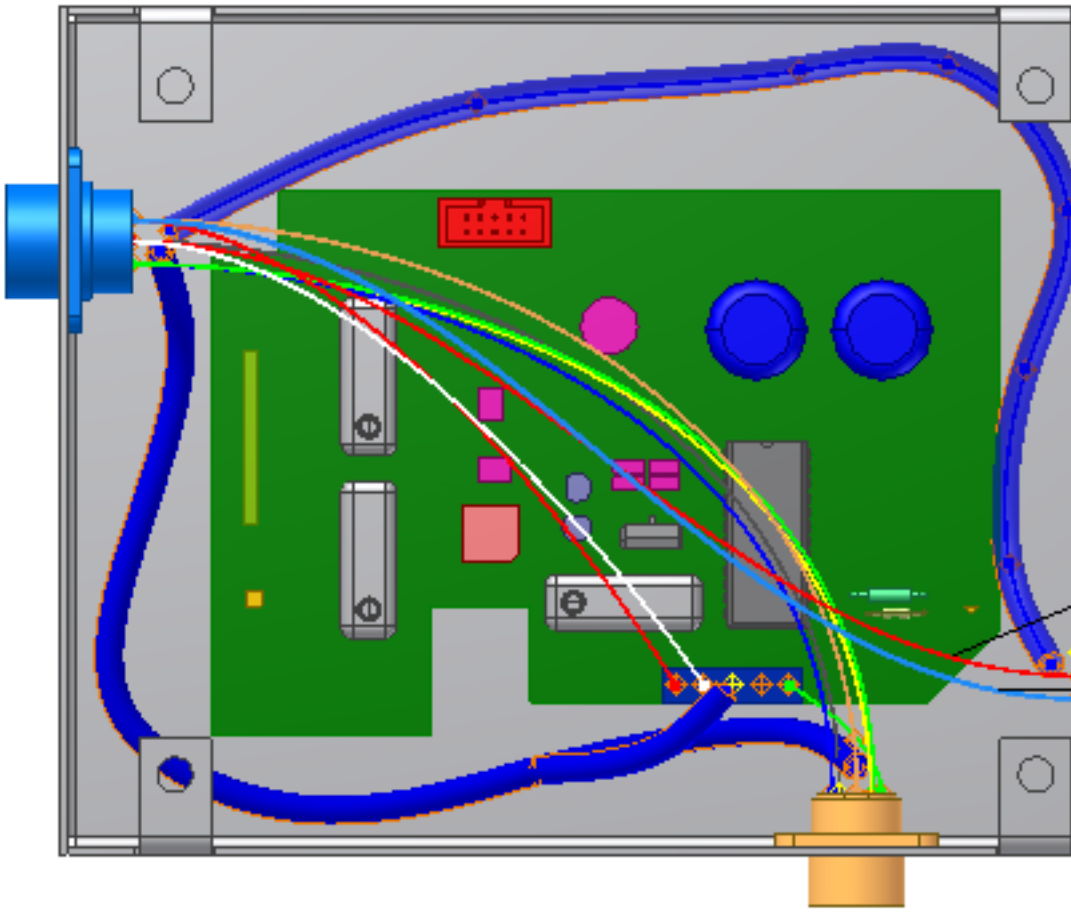
Define Manual Routes

Route a Belden cable into a single segment.

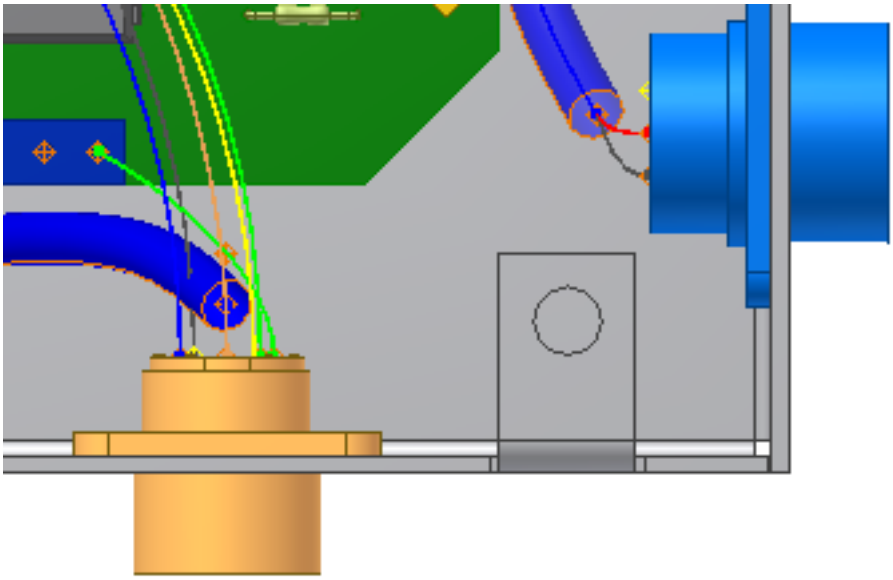
1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.
2. Activate **Harness Assembly2**.
3.  On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Route**.
4. In the graphics window, select a cable wire (**1** or **2**) to route.



5 Select the **Single Segment (Manual)** check box.

6 Select the **First Segment** button, select the **segment (3)** in the graphics window as shown previously, and then click **OK**.

Both cable wires are routed at once because the two wires represent a single cable. The cable lengths and bundle diameters are updated automatically.



NOTE All cable wires are routed or unrouted together and must take the same path (routed through the same segments) in the harness assembly.

7 Save the harness assembly.

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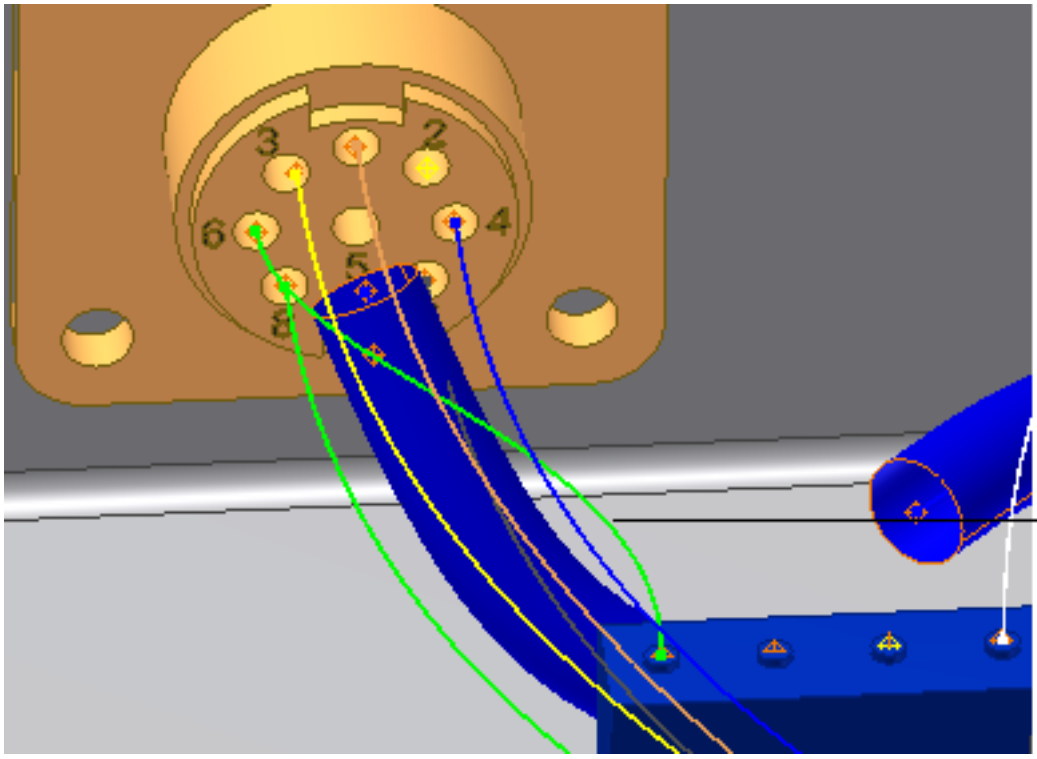
Route Selected Wires

Select a wire and route it manually into multiple segments.

1 Double-click **Harness Assembly1** to activate it.

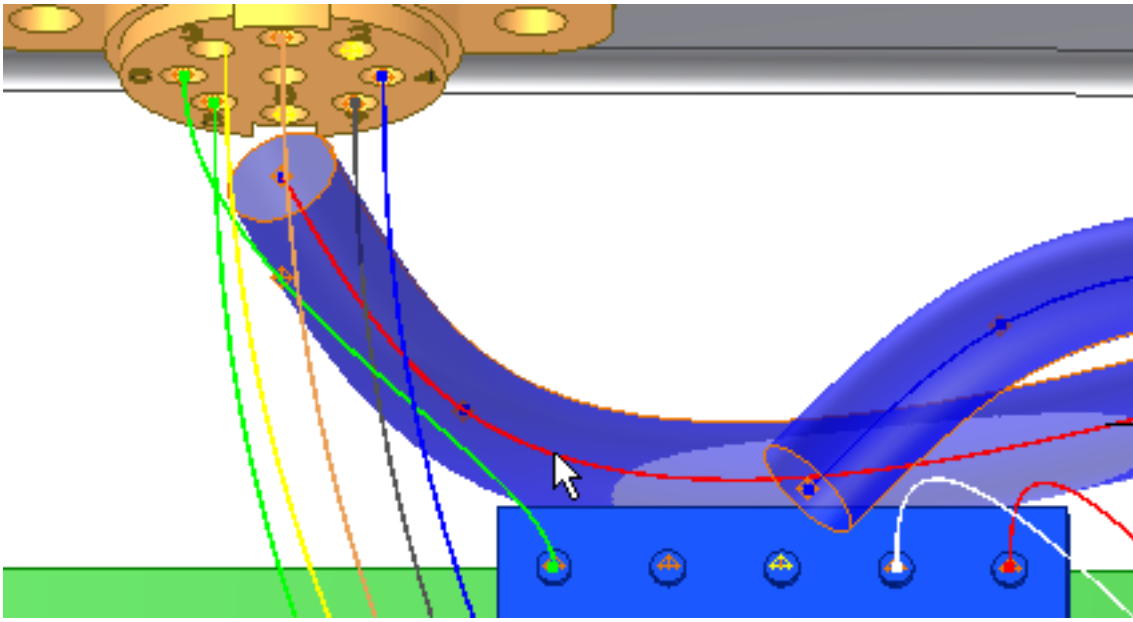
2 On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Route**.

3 In the graphics window, click the wire (**1**) as shown in the illustration. In the browser, this wire is named Wire1.



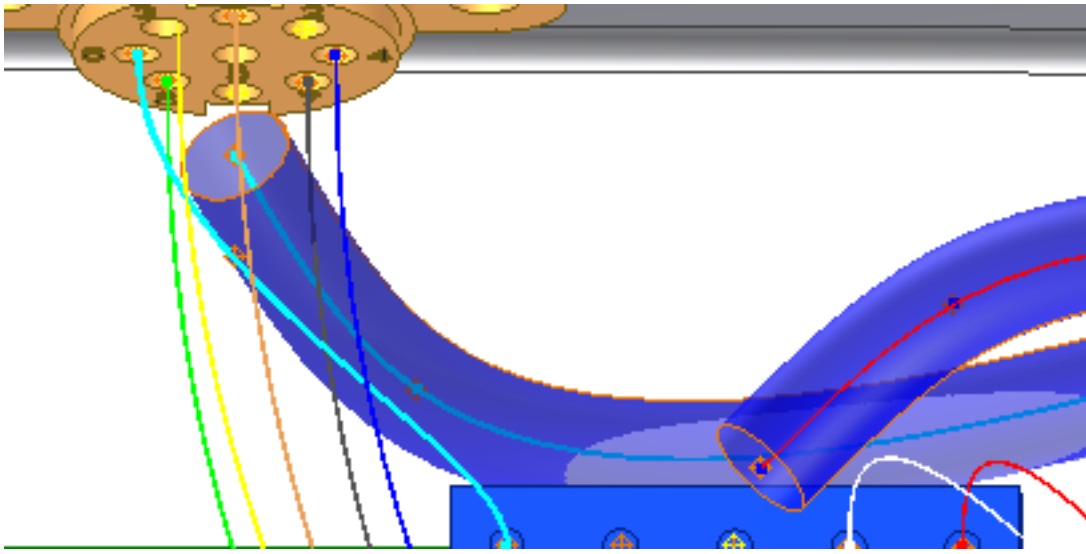
On the Route Wires dialog box, the Wires button changes from red to white to indicate that the wire selection is complete.

4 Click the **First Segment** button, and then click the first segment (1) as shown in the following illustration.

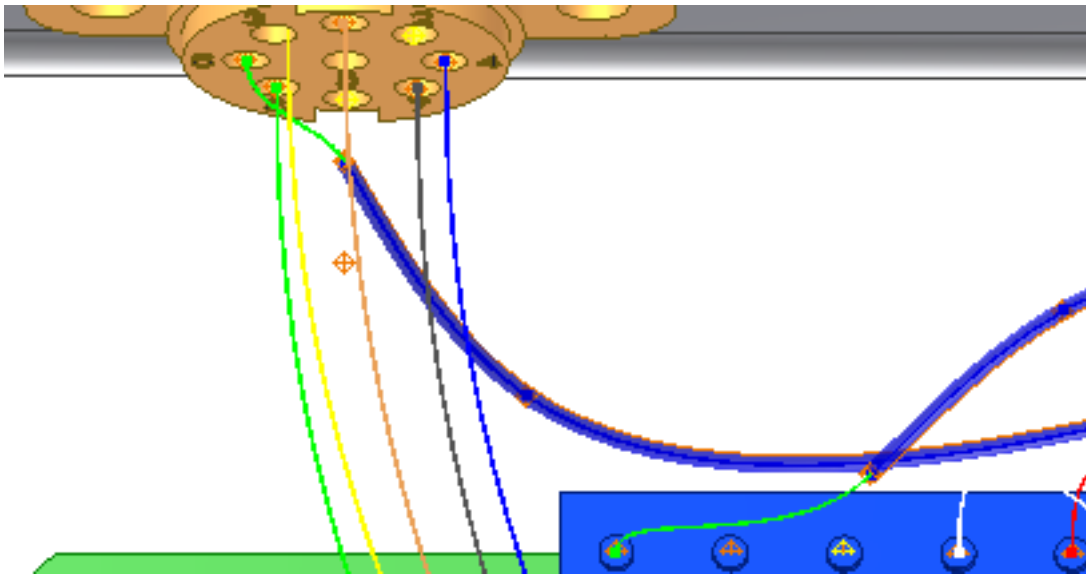


The First Segment button changes color, and the Last Segment button is depressed.

5 In the graphics window, click the branch segment (**2**), and then click **OK**.



The wire is routed into the main segment and out the branch. The segment diameter and wire length are calculated automatically.



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Route All Wires and Cables Automatically

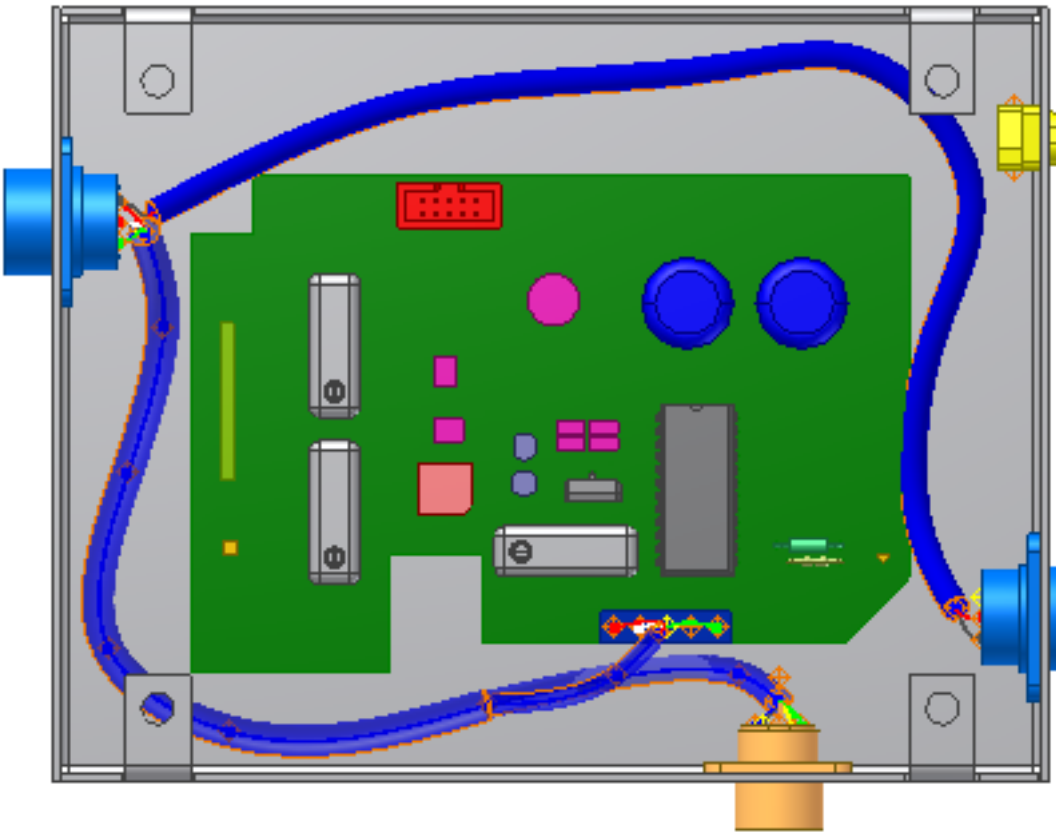
When automatically routing, the system uses the shortest path over 0.005 meters between segment ends. Routes of less than 0.005 meters fail.

Automatically route the remaining unrouted wires and cable wires into the segments.

1.  On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Automatic Route**.
2. On the Automatic Route dialog box, select **All Unrouted Wires**.

   The dialog box indicates that there are seven unrouted wires selected for routing.
3. Click **OK**.

   The remaining wires are routed into the segments, and the dialog box is closed.



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View Wire and Cable Paths

Highlight all the segments a routed object passes through.

1. With Harness Assembly1 still active, in the graphics window or the browser, select **Wire1**.
2. With the wire selected, right-click and select **View Path** from the context menu.

The wire stub (the exposed portions of the wires from the end segment to the pin), and the path of the segment the wire passes through are highlighted in the graphics window.

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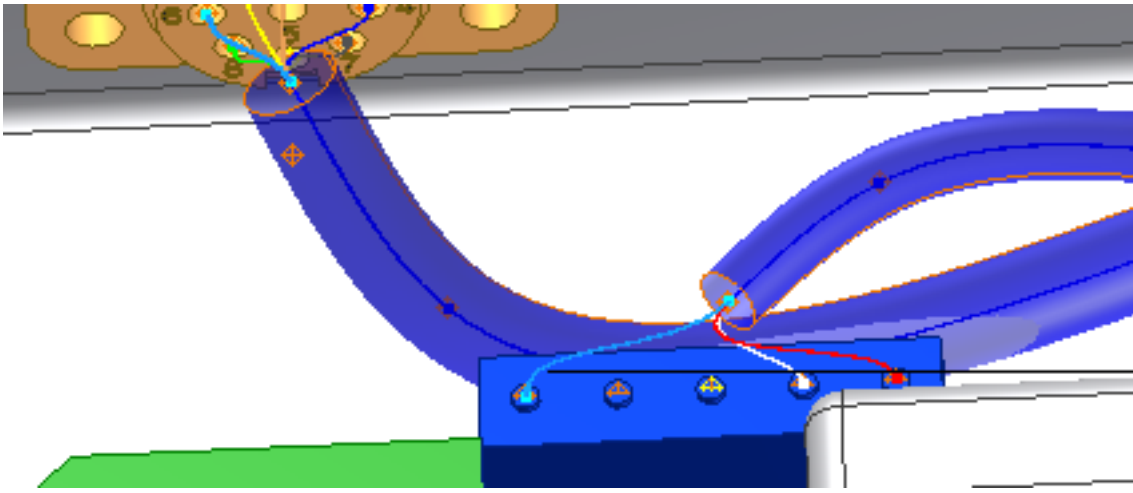
Unroute Wires

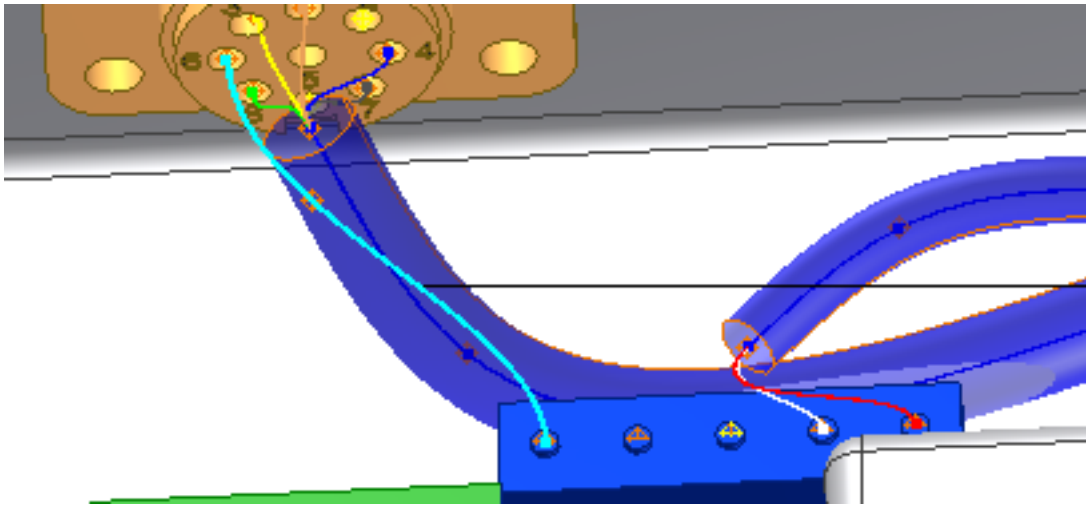
Unroute selected wires from selected segments. Individual, or discrete wires are unrouted independently of one another.

1. On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Unroute**.
2. Verify the **All Wires** check box is not selected. In the graphics window, select Wire1 (1) as the wire to unroute.



3. Click **OK** in the Unroute dialog box.

   Wire1 (1) returns to its unrouted state.



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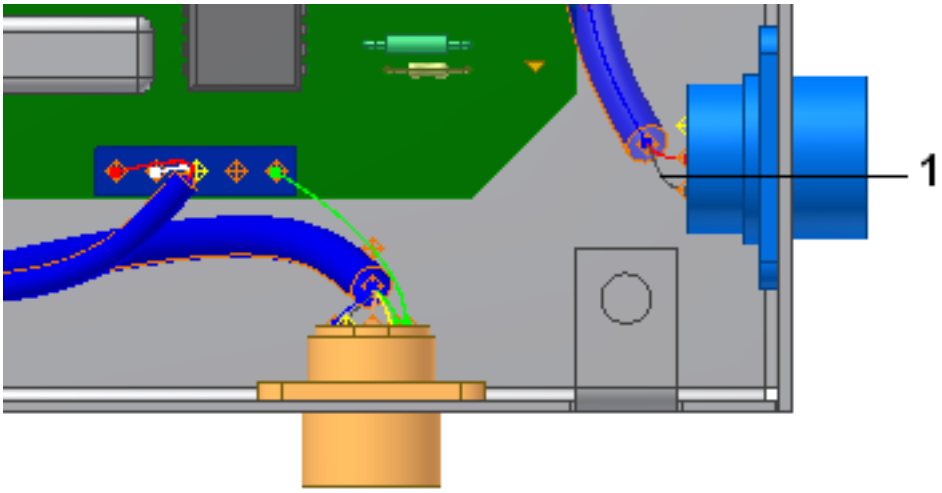
Unroute Cables

Unroute a cable. All cable wires associated to the same cable are unrouted together.

1. Activate **Harness Assembly2**.
2.  On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Unroute**.
3. Verify the **All Wires** and **All segments** check boxes are cleared.
4. Make sure that the **Wires** button is selected, and then in the graphics window, select the black cable wire (1), as the cable wire to unroute.

   The cable wire is labeled as Cable1:1 in the browser.



5 Click the **Segments** button, and then, in the graphics window, click the segment between the two blue connectors (parts 360575:1 and 360575:2).

6 Click **OK**.

Both wires in the cable return to their unrouted state.

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Unroute All Wires from All Segments

Unroute all remaining wires from all segments.

1 Activate **Harness Assembly1**.

2 On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Unroute**.

3 On the Unroute dialog box, verify that both the **All Wires** and **All Segments** check boxes are selected.

4 Click **OK**.

The wires are unrouted and wire lengths update to reflect the change.

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Route All Wires and Cables

Return all wires and cables to their routed state.

1. Use the Automatic Route command to route all wires and cables in **Harness Assembly1** and **Harness Assembly2**.
2. Return to the top-level assembly and save all files. The assembly is used in the next tutorial.

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Summary

In this tutorial, you learned to:

- Define manual routes.
- Route wires.
- Define automatic routes.
- View wire and cable paths.
- Unroute wires.
- Unroute cables.
- Unroute all wires from all segments.

Use your saved file from this tutorial to create splices in the next tutorial

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Create Splices

13

About this tutorial

Work with splices in wires.

Category	**Routed Systems**
Time Required	10 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial.

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

Splice the wires for a visual representation of the splice. Redefine the splice on the segment for optimal positioning, and then route the wires.

Objectives

- Create splices.
- Splice wires or add wires to the splices.
- Redefine the splice.
- Access properties for splices and splice pins.

Prerequisites

- Perform the previous tutorial, Route Wires and Cables, and save the tutorial file for use in this tutorial.

- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic “Fundamentals” and the New User tutorials for further information.

Navigation Tips

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

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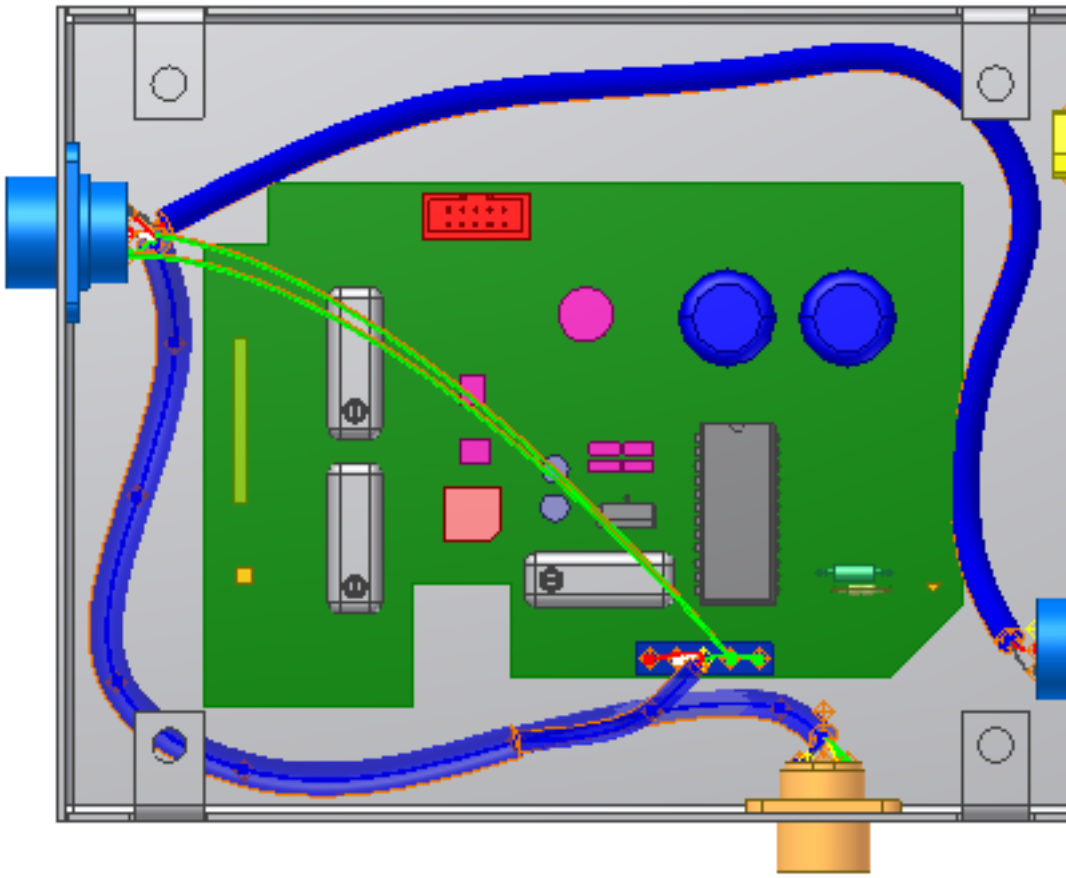
Create Splices

Create the splice at an arbitrary location, and then add two wires to the splice.

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.
2. In the browser, double-click **Harness Assembly1**.
3. Create a Generic 22AWG-GRN wire from **U7 Pin 3** on Connector 360575:1 to **J12 Pin 2** on LTP:1. Use the default Wire ID.
4. Create another Generic 22AWG-GRN wire from **U7 Pin 5** on connector 360575:1 to **J12 Pin 2** on LTP:1. Use the default Wire ID and then click **OK**. The new wires are shown in the following image.



5  On the ribbon, click **Cable and Harness tab ➤ Create panel ➤ Create Splice**.

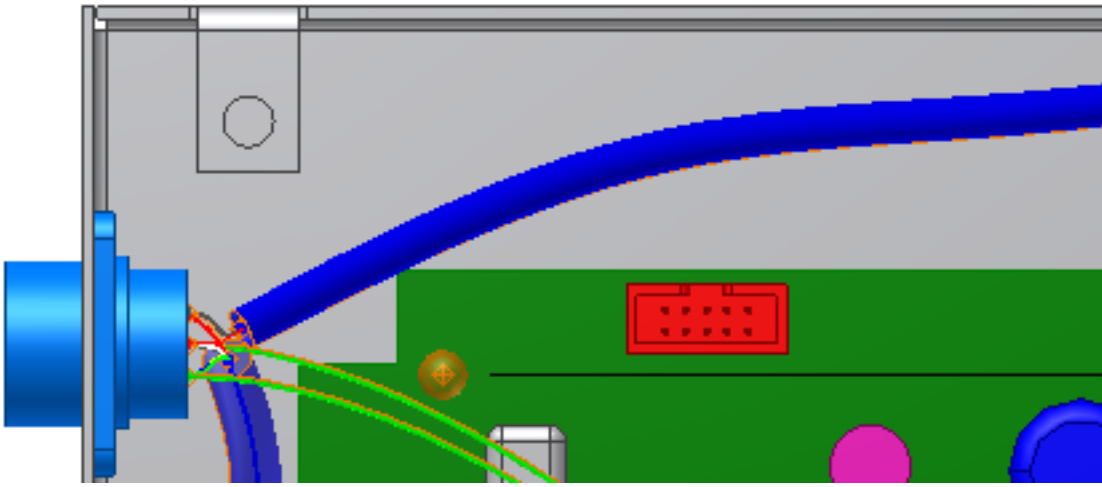
6 On the Create Splice dialog box, leave the default values.

A green circle that represents the splice is attached to the cursor. A line representing the offset distance extends from the splice object.

7 Click in the graphics window near the location shown in the following image to place the splice. You move the splice later in the tutorial, so the exact location is not important.

The splice (1) is shown in the following image.

NOTE To change the offset right-click again, select **Edit Offset**, and then enter a value. The default offset is set on the Splice tab of the Harness Settings dialog box.



8 Click **OK** to create the splice.

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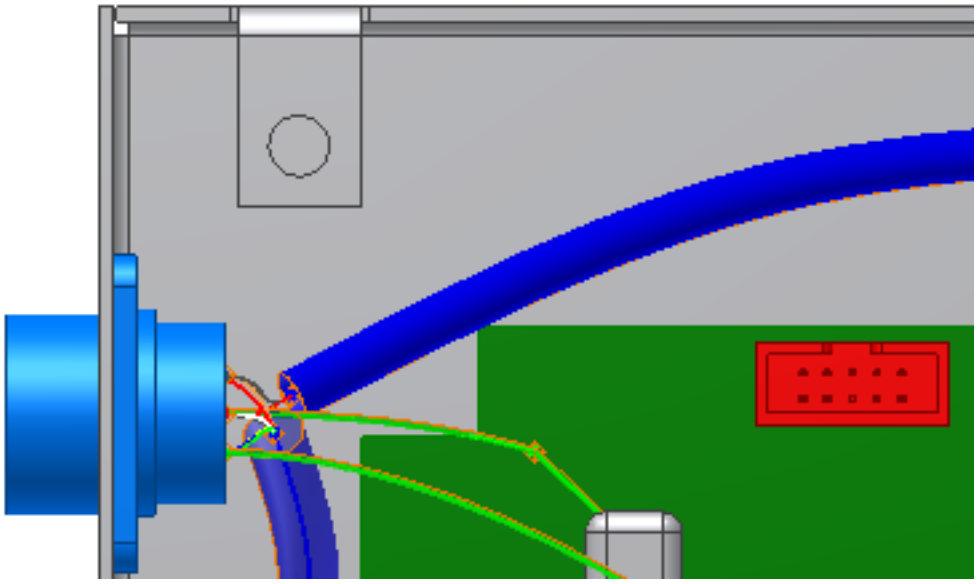
Splice Wires or Add Wires to Splices

To visualize the best placement of the splice in the harness, place the splice on the wires before you redefine the splice onto the segment.

1 In the browser or graphics window click the splice, right-click, and then select **Splice Wire** on the context menu.

2 Click **Wire 2**, the wire to splice.

The wire is split and added to the browser with the unique name, Wire 2_1.



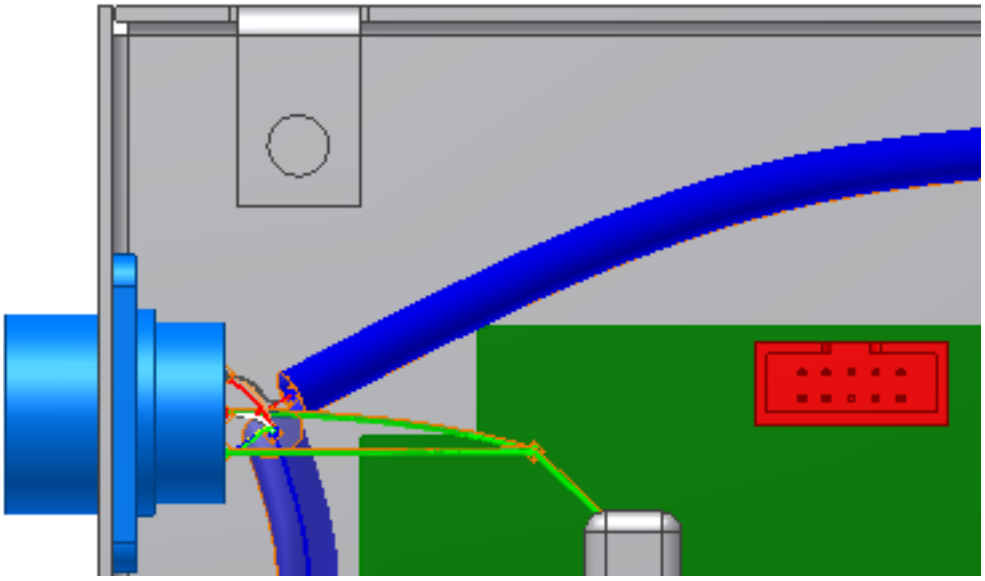
3 Click **Wire3**, right-click, and then select **Splice** on the context menu.

4 Click the splice.

5 On the confirmation dialog box, click **OK** to remove redundant wires.

Redundant wires (wires that share a common point when spliced) are deleted.

NOTE Redundant wires are removed only when you manually splice wires. If you import wires or use **Create Wire** or **Edit Wire** to place them on splices, you do not remove redundant wires.



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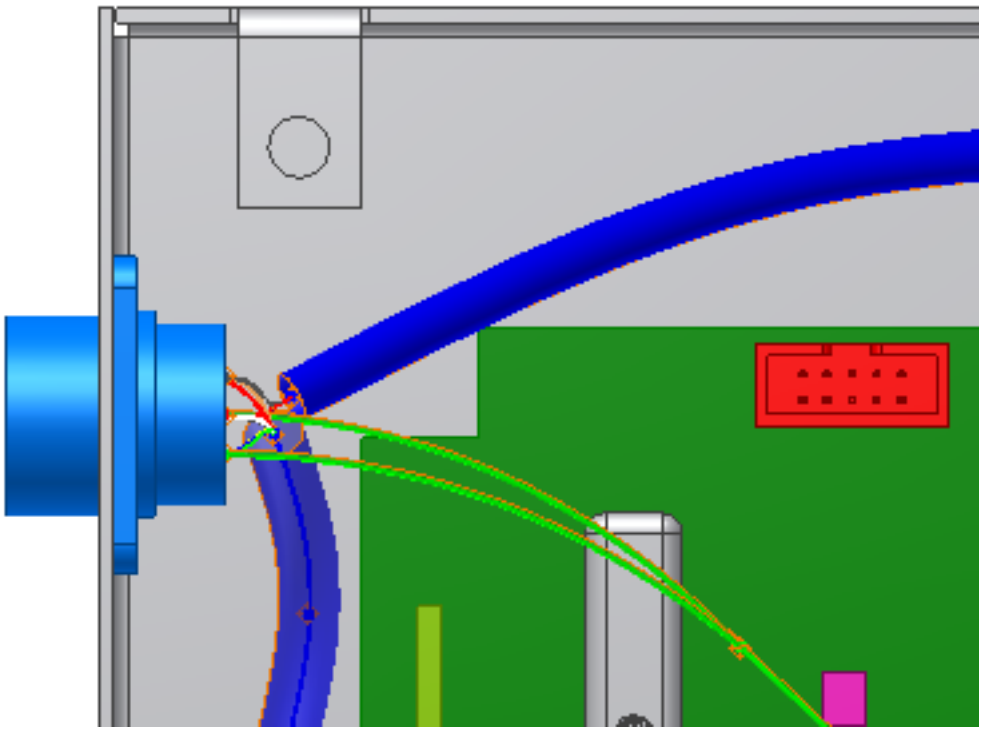
Redefine the Splice

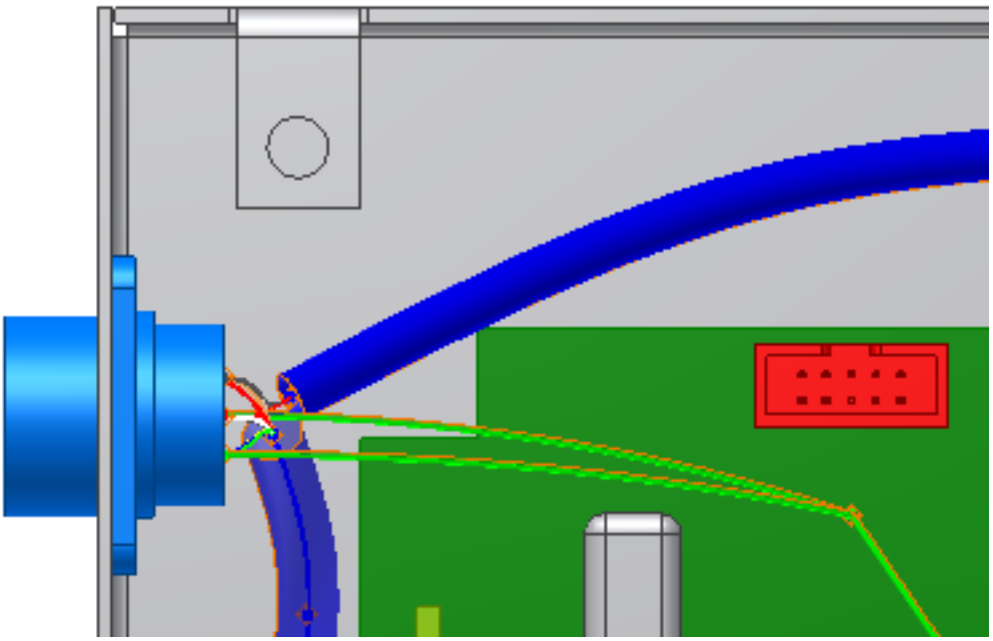
Redefine a splice along the wire or to an arbitrary location

1. Right-click the splice, and then select **Redefine Splice** on the context menu.
2. Click a new location along the same wire.

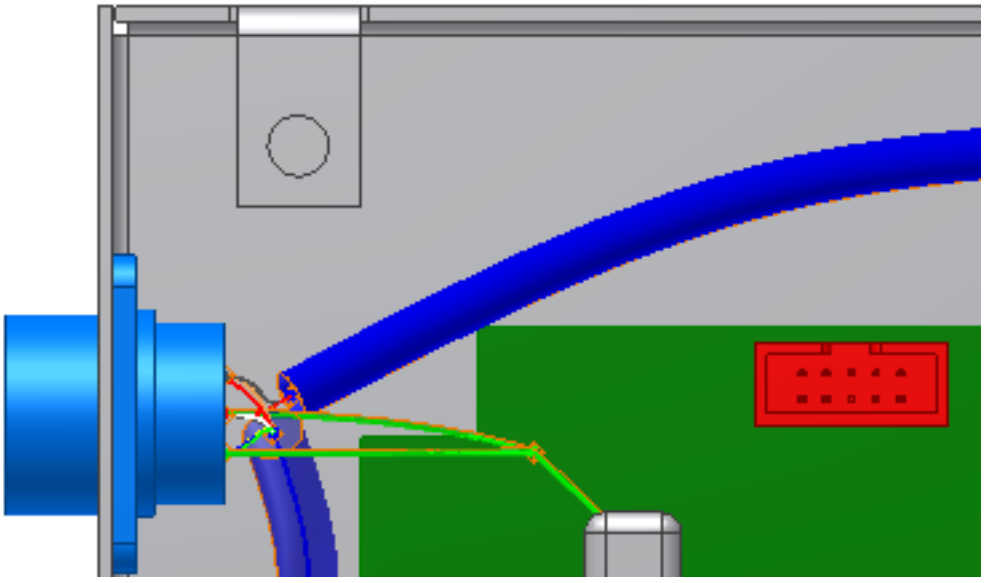


3 Right-click the Splice again, select **Redefine Splice**, and then select a new location in the enclosure.

The wires move with the splice.



4 Use **Undo** twice to return the splice to the original location.



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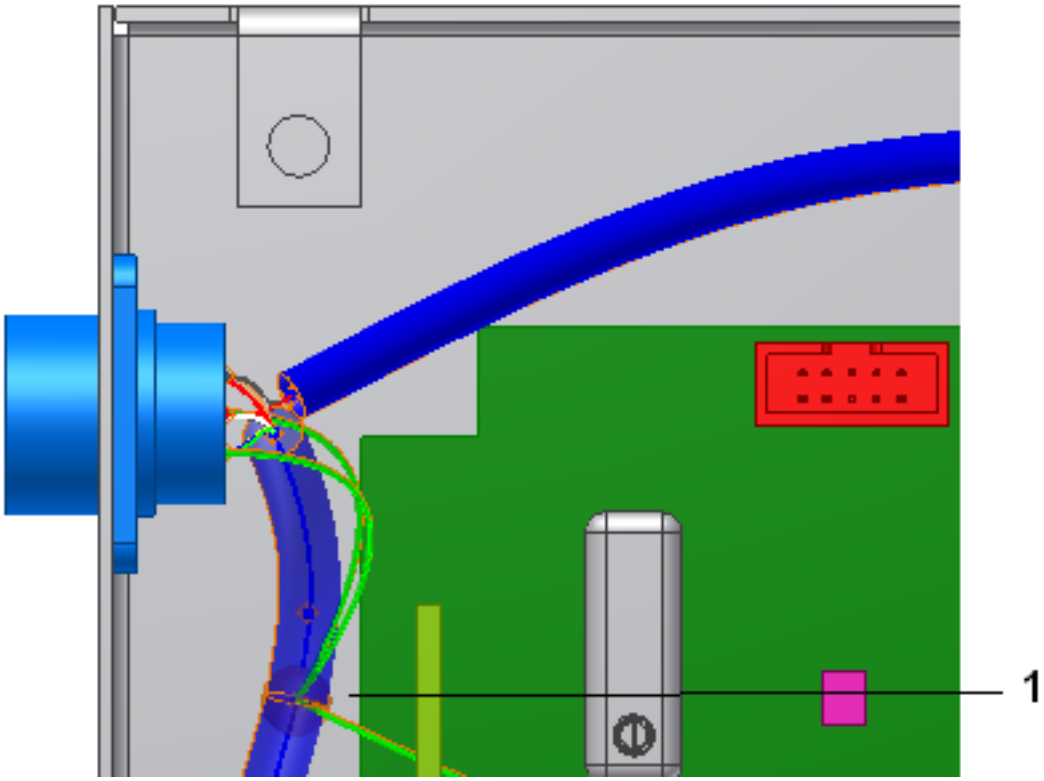
Redefine the Splice onto a Segment

It is recommended that you place splices on segments. Redefine the splice onto a segment, and then route the wires.

1. Right-click the splice in the graphics window, and then select **Redefine Splice** on the context menu.
2. Click the segment to locate the splice (1). Click directly on the spline at the center of the segment, or click a segment work point.

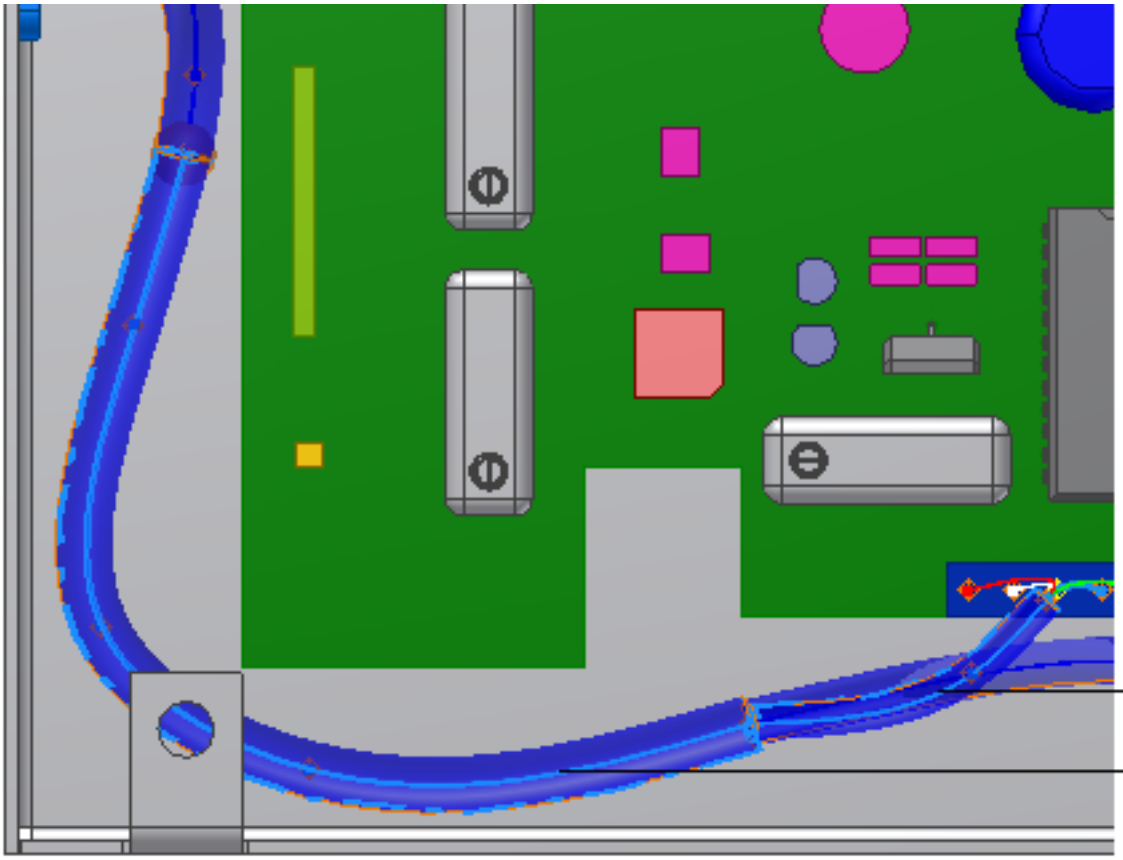


3. On the ribbon, click **Cable and Harness tab ➤ Route panel ➤ Automatic Route**.
4. Make sure that **All Unrouted Wires** is selected, and then click **OK**.

The wires are routed into the spliced segment and out the branched segment.

5 In the browser, right click **Wire2_1**, and then select **View Path** to make sure that the wires were routed as intended.

The wire path (1) is highlighted to and from the splice along with the segments that the wire is routed through.



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Access Properties for Splices and Splice Pins

1. In the browser or graphics window, right-click the splice, and select **Harness Properties**. The Splice Properties dialog box displays.
2. Examine the properties. There is an embedded length value of 0. You can assign Virtual Parts to the splice through this dialog box. Click **OK**.
3. In the browser, under the Splices folder, expand the splice to view the splice pins. Right-click a splice point, and then select **Harness Properties** to see the Splice Pin Properties dialog box.
4. Examine the properties, and then click **OK**.
5. Return to the top-level assembly and save all files. The assembly is used in the next tutorial.

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Summary

In this tutorial, you:

- Create splices.
- Splice wires or add wires to the splices.
- Redefine the splice.
- Access properties for splices and splice pins.

Use your saved file from this tutorial to work with ribbon cables in the next tutorial

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Ribbon Cables 14

About this tutorial

Create ribbon cables.

Category	**Routed Systems**
Time Required	35 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial.

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

Before you create a ribbon cable in the harness assembly, the definition for the raw ribbon cable must exist in the Cable and Harness Library. Add raw ribbon cable definitions to the library the same way you add other harness objects.

Objectives

- Define raw ribbon cables.
- Place connectors from content center.
- Create ribbon cables.
- Adjust ribbon cable orientation and shape.

Prerequisites

- Perform the previous tutorial, Create Splices, and save that tutorial file for use in this tutorial.

- This tutorial uses a component from the Content Center. The Content Center must be installed and configured to complete the tutorial.
- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic "Fundamentals" and the New User tutorials for further information.

Navigation Tips

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

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Define Raw Ribbon Cables

You can use generic connectors provided in the Content Center, or create custom connectors that you author and publish to Content Center. Create a custom raw ribbon cable definition.

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.
2. Activate the **Ribbon Cable** harness assembly.
3. On the Ribbon, click **Cable and Harness tab ➤ Manage panel ➤ Library**.
4. Select **Raw Ribbon Cable** as the harness object type.
5. Click the **New** command and examine the properties that are unique to ribbon cables. Examples include the ability to set different colors for conductor one and the ribbon cable body.
6. On the General tab, enter the manufacturing information for the ribbon cable:

   Name: 28-AWG-9-Conductor

   Category: Tutorial

   Part Number: 28-AWG-9-Conductor

   Conductor One: Red

   Ribbon Body: Gray (Light)

   Number of Conductors: 9

   Pitch: 0.05 in

   Height: 0.035 in

Width:. 0.450 in

Gauge: 28

(Core size is optional. Leave it blank.)

7 Click **Save**, and the new raw ribbon cable is added to the library in the Tutorial category.

8 Click **Close**.

The ribbon cable definition is added to the Cable and Harness library.

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Place Connectors from Content Center

The Content Center contains a sample ribbon cable connector. Place one instance of a 10-pin connector in the Ribbon Cable harness assembly.

1  On the ribbon, click **Assemble tab ➤ Component panel ➤ Place from Content Center**.

2 On the Place from Content Center, Category View pane, navigate to **Cable & Harness ➤ Connectors ➤ Ribbon Cable** and select the **Ribbon Cable Connector**.

3  Click **Table View** to view the family members for the ribbon cable connector.

4 In the preview table, note the Member name for the 10-pin connector. In this case, it is the first member in the list.

5 Double-click the image of the ribbon cable connector to select it.

6 On the Ribbon Cable Connector dialog box, ensure that the following are selected:

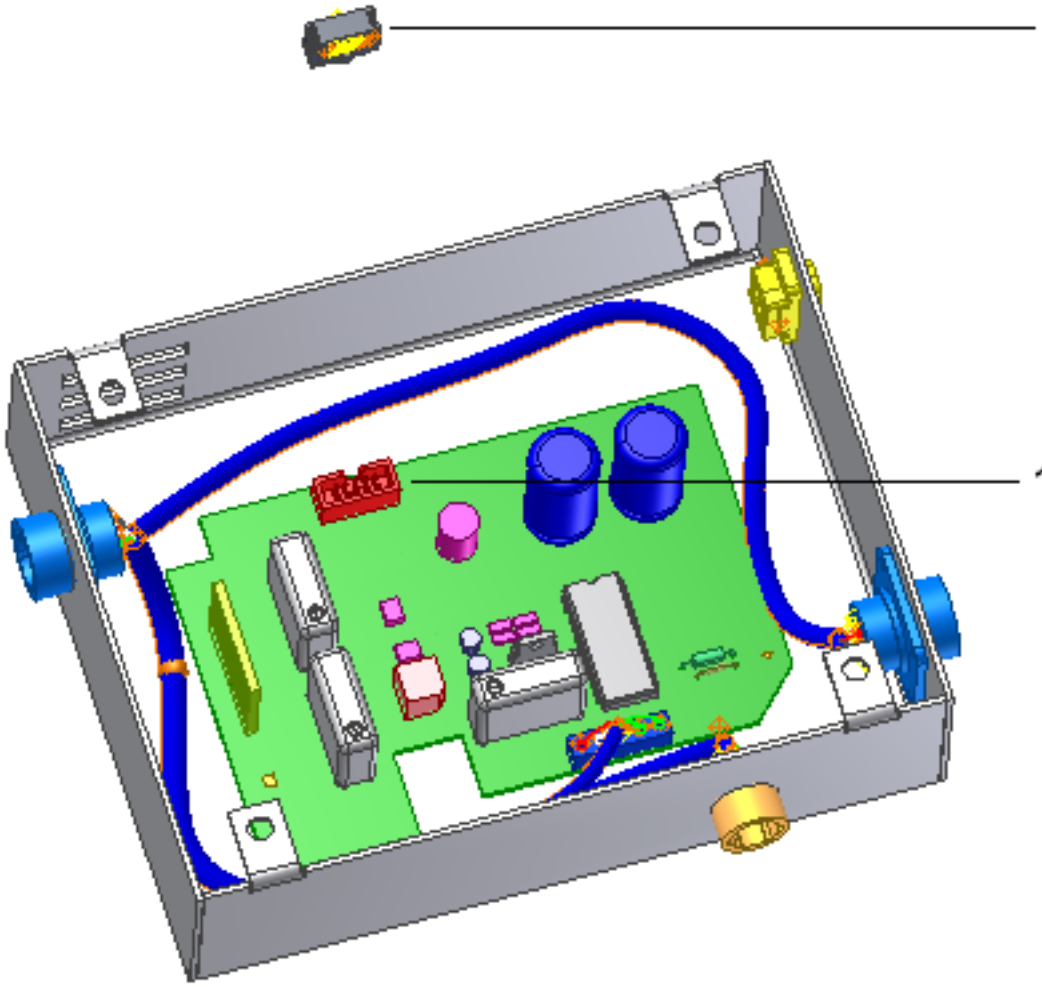
- `Ribbon Cable Connector-01`
- `As Standard`

7 Click **OK**.

In the graphics window, the part is attached to the cursor.

8 Rotate the enclosure to get a better view of the RC Male Connector (1).

9 Click in the background of the graphics window to place the part (2). Exact placement is not important.



10 Right-click and select **Done**.

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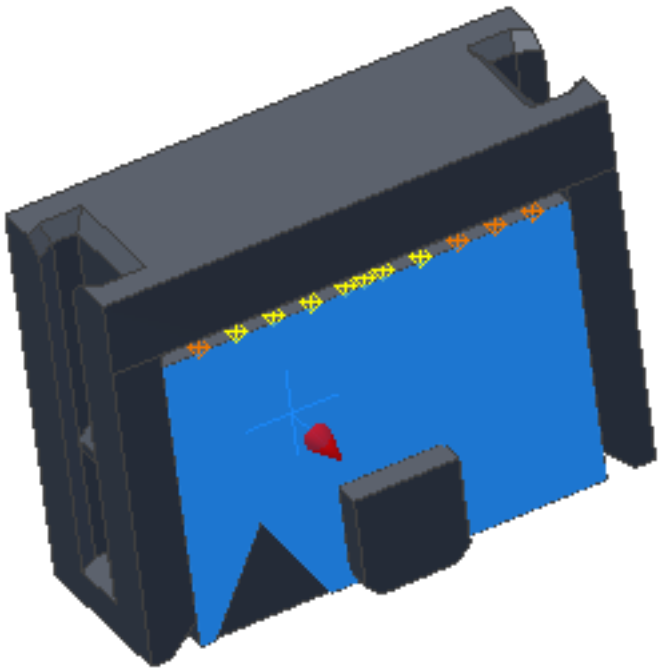
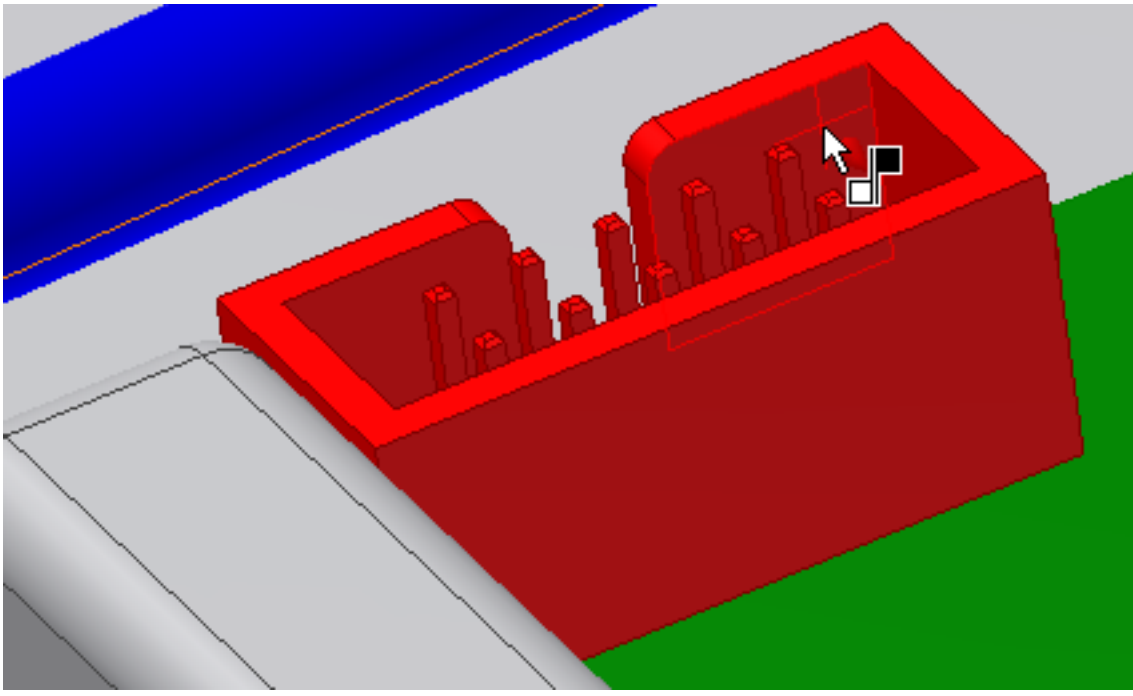
Constrain the Connector

Place three mate constraints to constrain the ribbon cable connector in place. The following steps use the Constrain command, but you can also use the Assemble command.

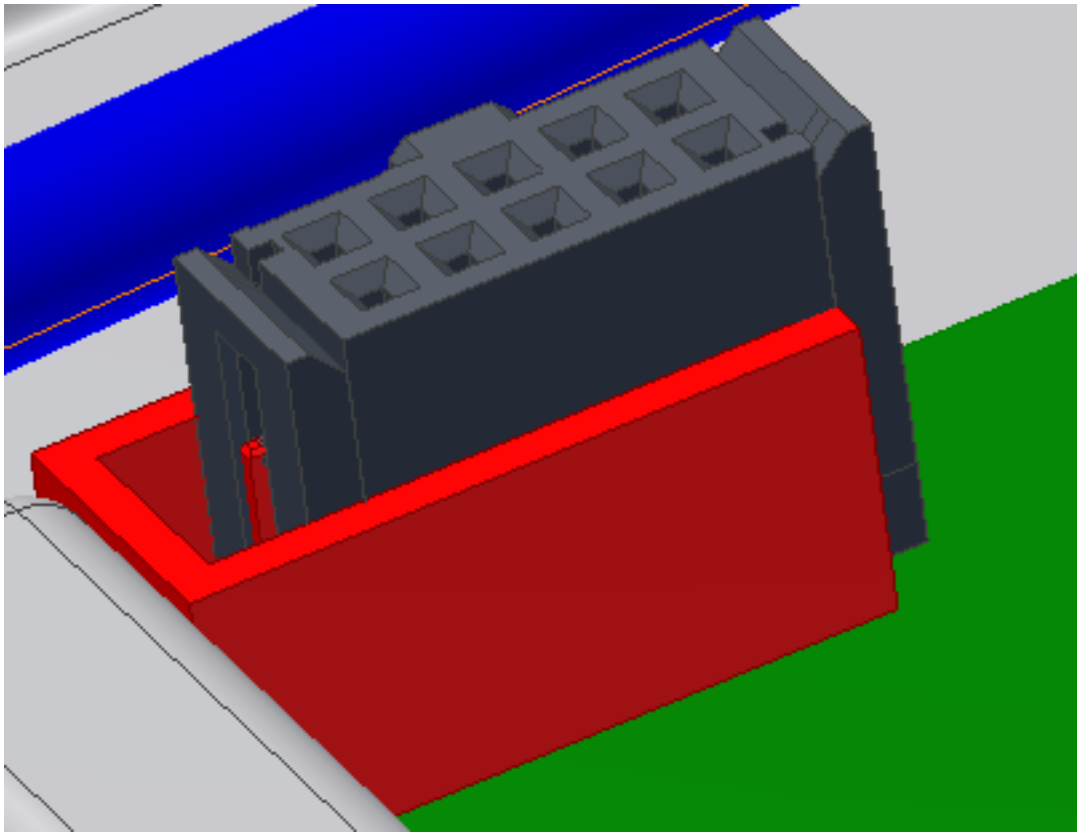
1. Activate the top-level assembly, **Enclosure_Assembly.iam**.
2.  On the ribbon, click **Assemble tab ➤ Position panel ➤ Constrain**.
3. Ensure **Show Preview** is selected.
4. Ensure **Mate constraint** and the first selection button is selected, and then click the face of the ribbon cable connector.



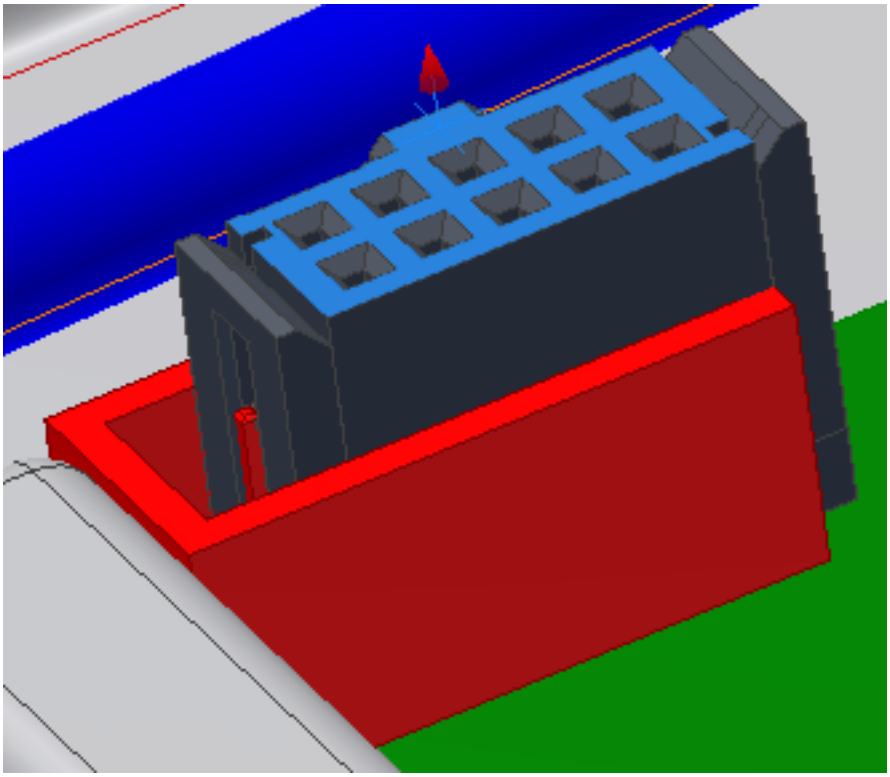
5. The second selection tool is now active. Click the wall of the **RC Male Connector** as the geometry to mate to.



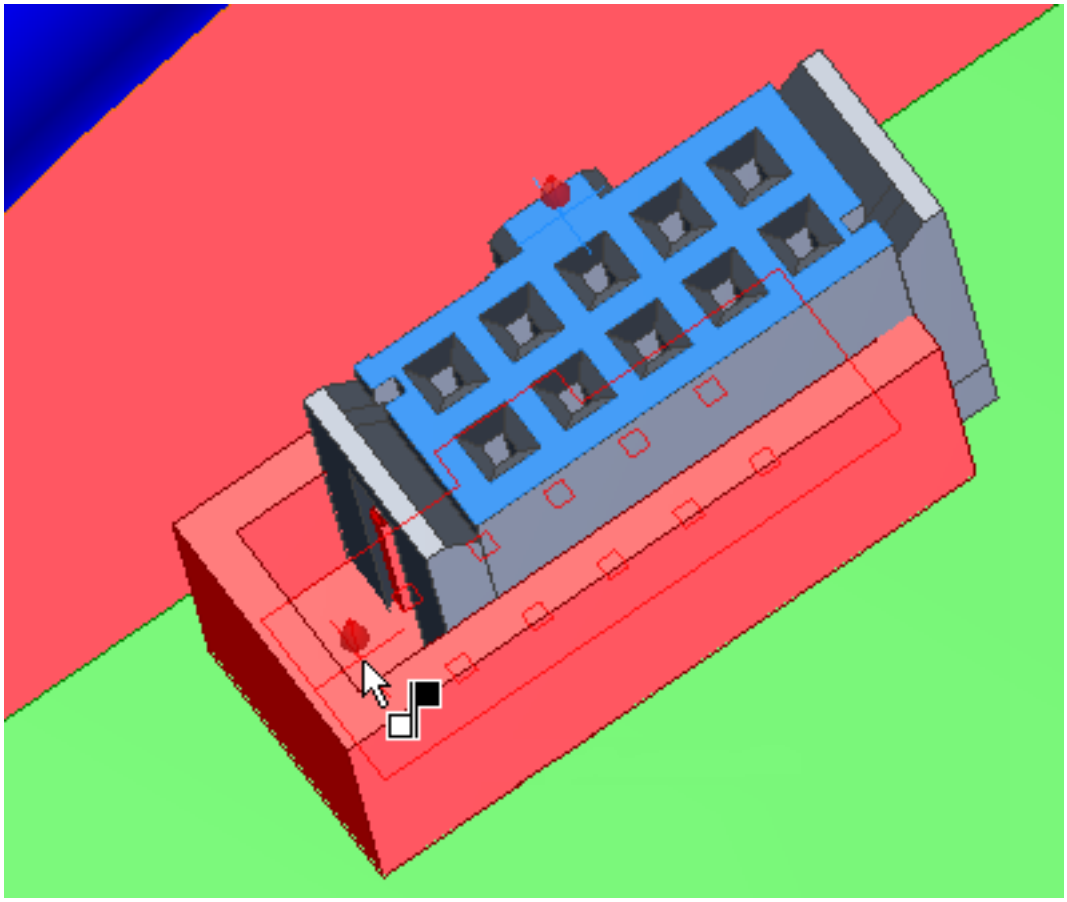
6 When the preview shows the connector has moved into place, click **Apply** to create the first mate constraint.



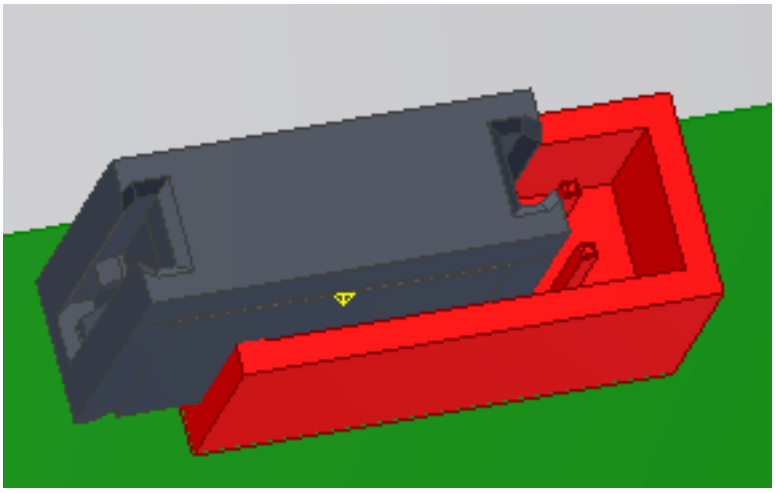
7 For the second mate constraint, click the top face of the ribbon cable connector.



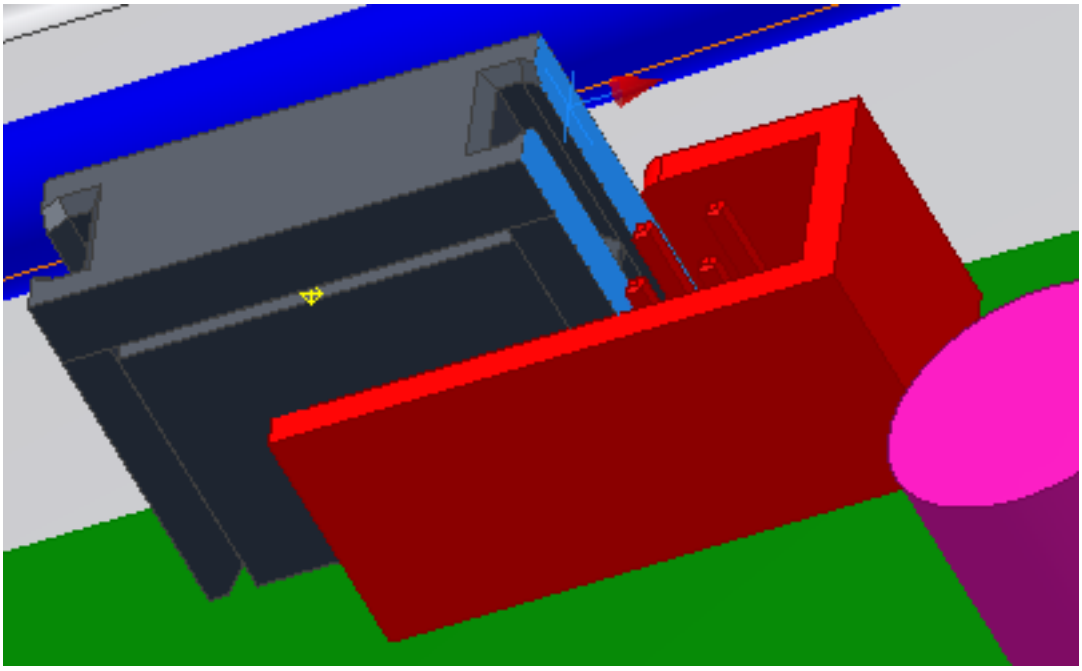
8 Click the bottom face of the **RC Male Connector**.



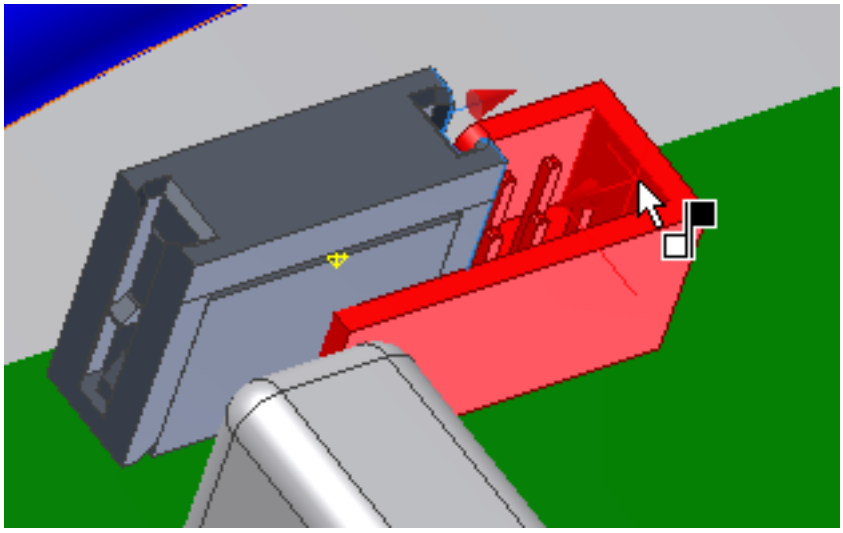
9 When the preview shows the connector has moved into place, click **Apply** to create the second mate constraint.



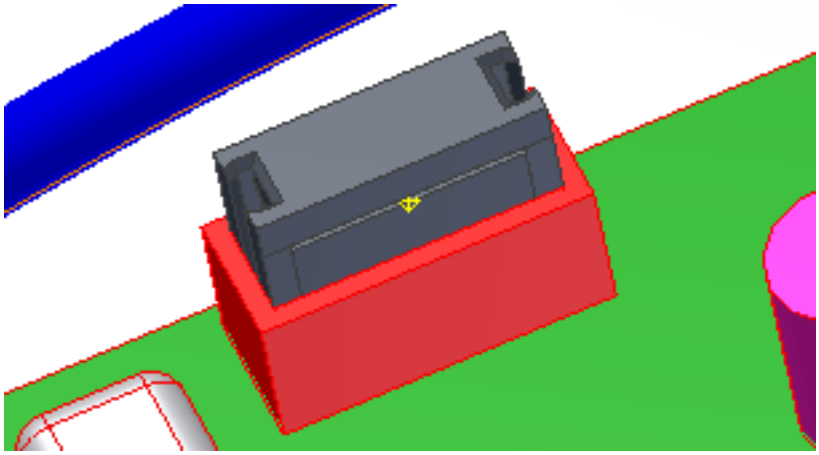
10 For the final mate constraint, click the right side of the ribbon cable connector.



11 Click the inside face of the **RC Male Connector**.



12 When the connector moves into place, click **OK**.



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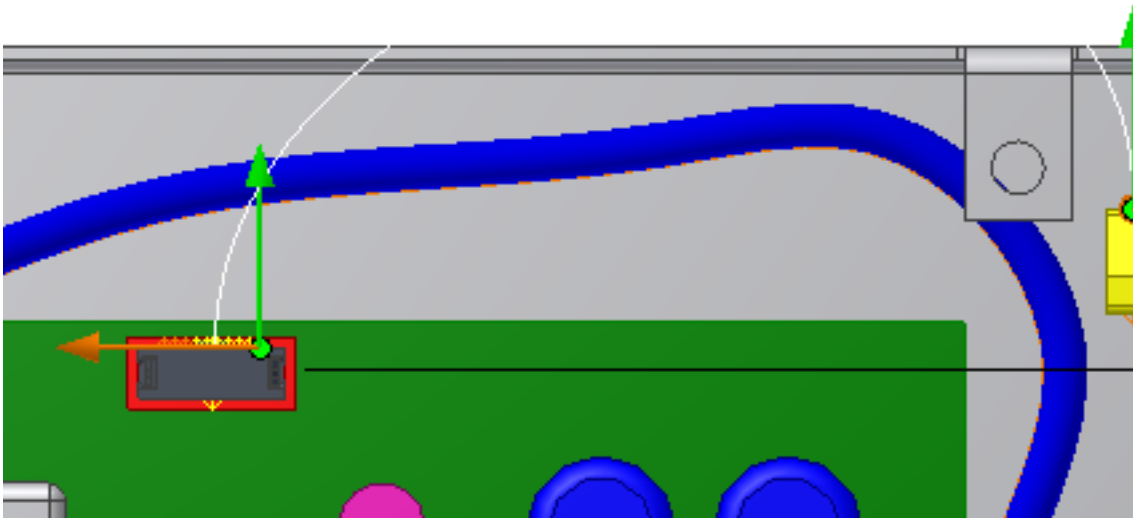
Create a Ribbon Cable

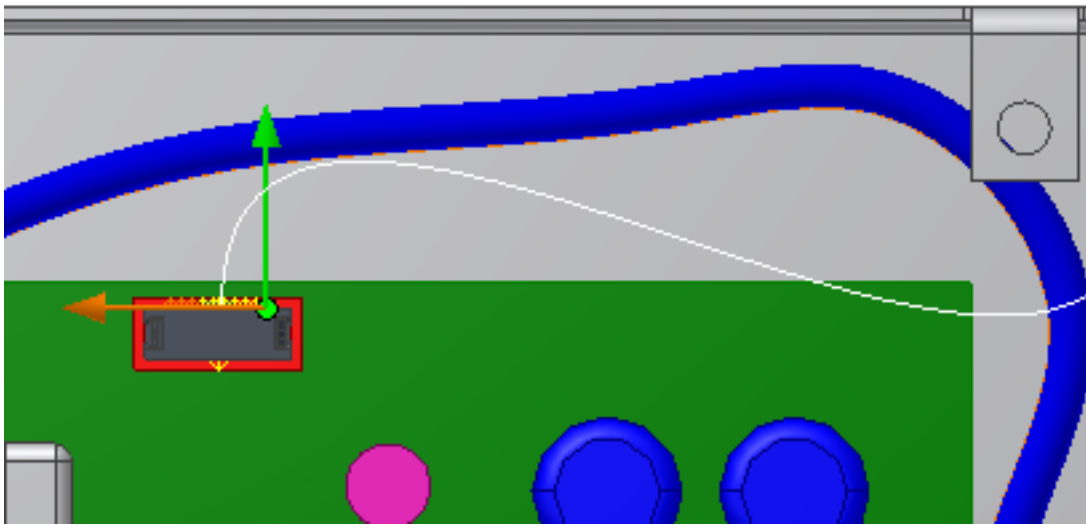
Create a ribbon cable between a start and end connector.

1. Activate the **Ribbon Cable** assembly.
2.  On the ribbon, click **Cable and Harness tab ➤ Create panel ➤ Create Ribbon Cable**.
3. Ensure that the Category is set to **Tutorial** and the Name is set to **28-AWG-9-Conductor**.
4. In the graphics window, select the start connector (1) and the end connector (2).

   The directional arrows and the spline appear. The green arrow indicates the outward direction of the connector. The orange arrow shows the orientation for the width of the ribbon cable body. The spline indicates the initial shape of the ribbon cable.

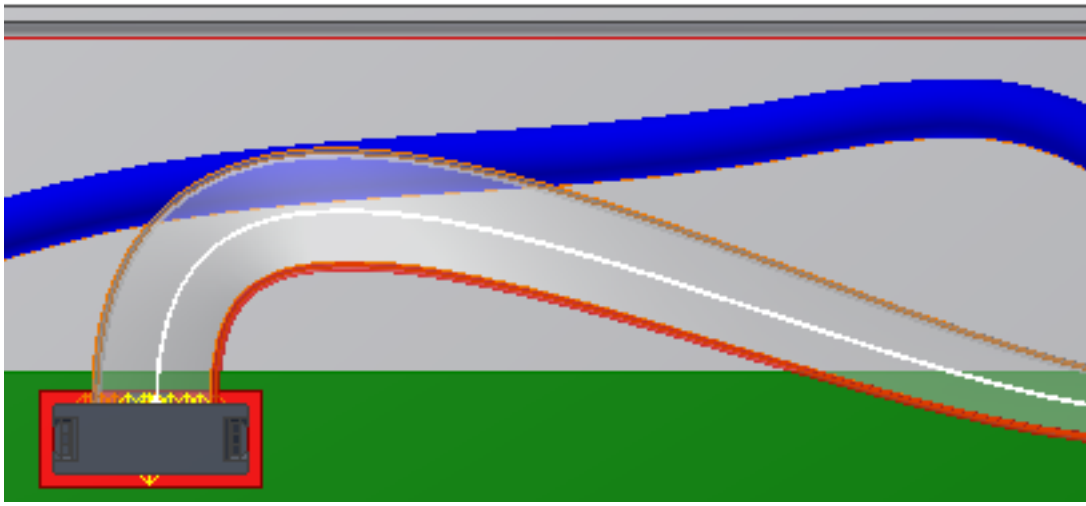


5.  The outward direction on the end connector is not correct. Click the outward direction tool in the End Connector area to change the direction.



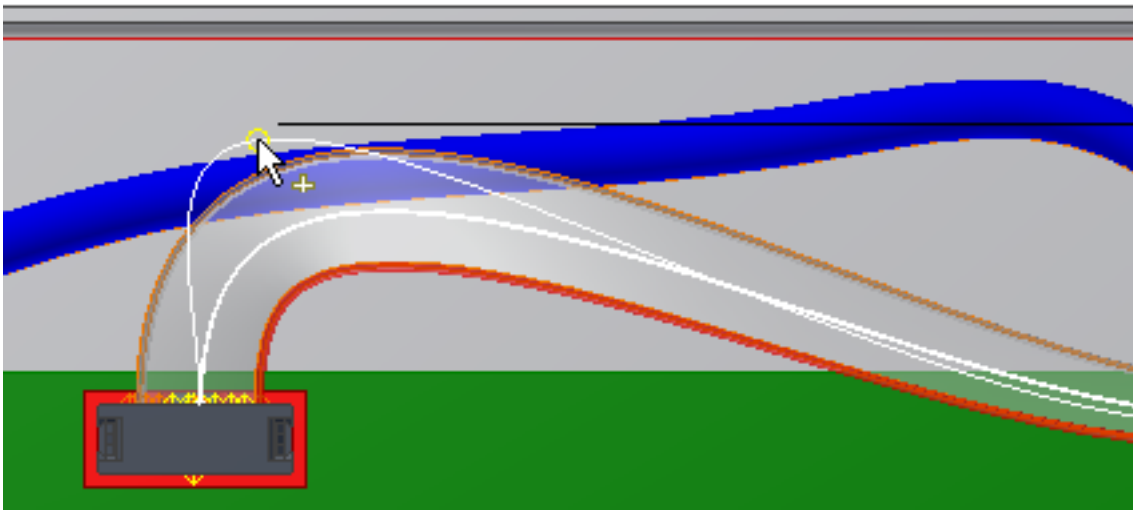
6 Click **OK**.

You are still in ribbon creation mode and can add intermediate points to change the current shape.

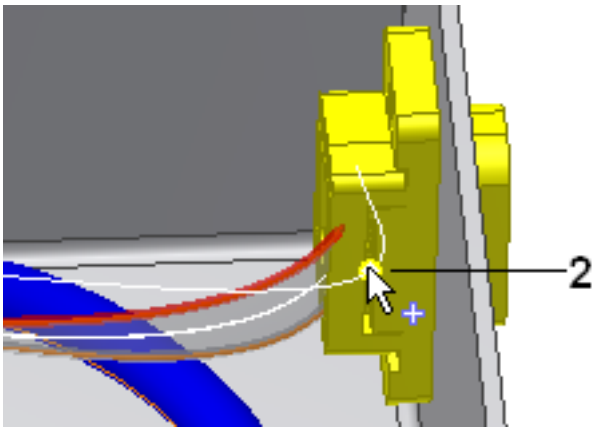


7 Right-click, select **Edit Offset**, and enter **0.500 in**.

8 Click in the graphics window to add an intermediate point (1) to control how the ribbon cable approaches the start connector.



9 Click in the graphics window to control the approach for the end connector. This time, click an existing work point (2).



10 Right-click and select **Finish**.

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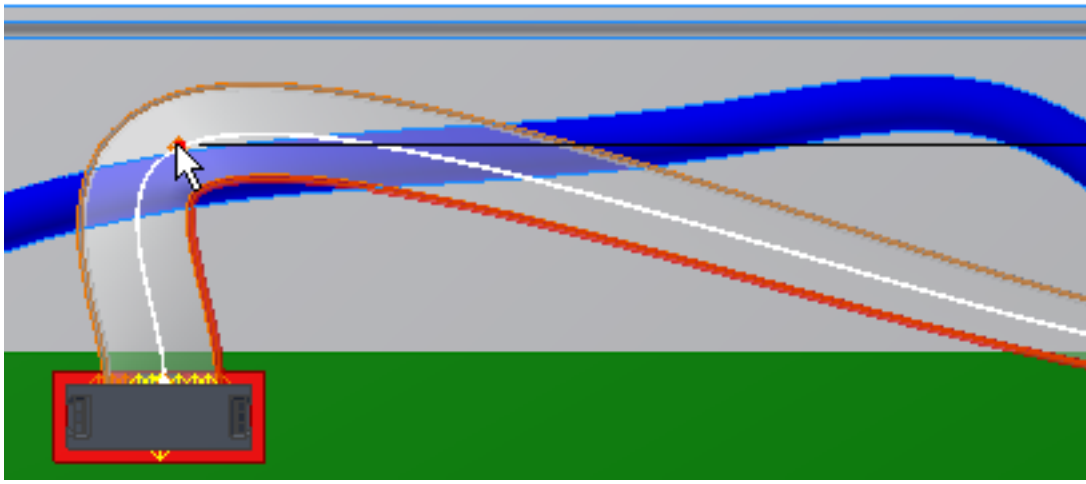
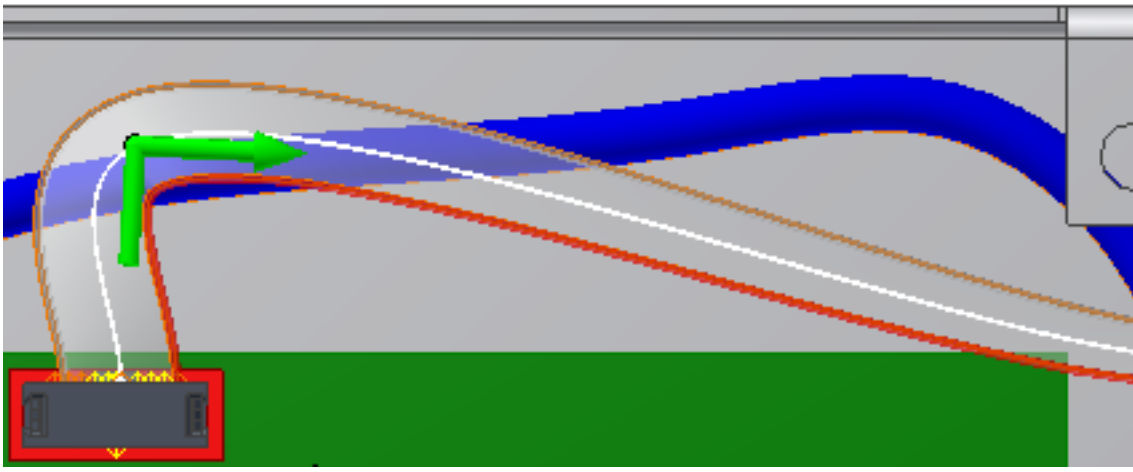
Adjust Ribbon Cable Orientation and Shape

Add a single fold to the ribbon cable to align it with the slot at connector DSub Ribbon.

1  Click the **Cable and Harness tab ➤ Create panel ➤ Create Fold**, and then click the first point (1) you created.

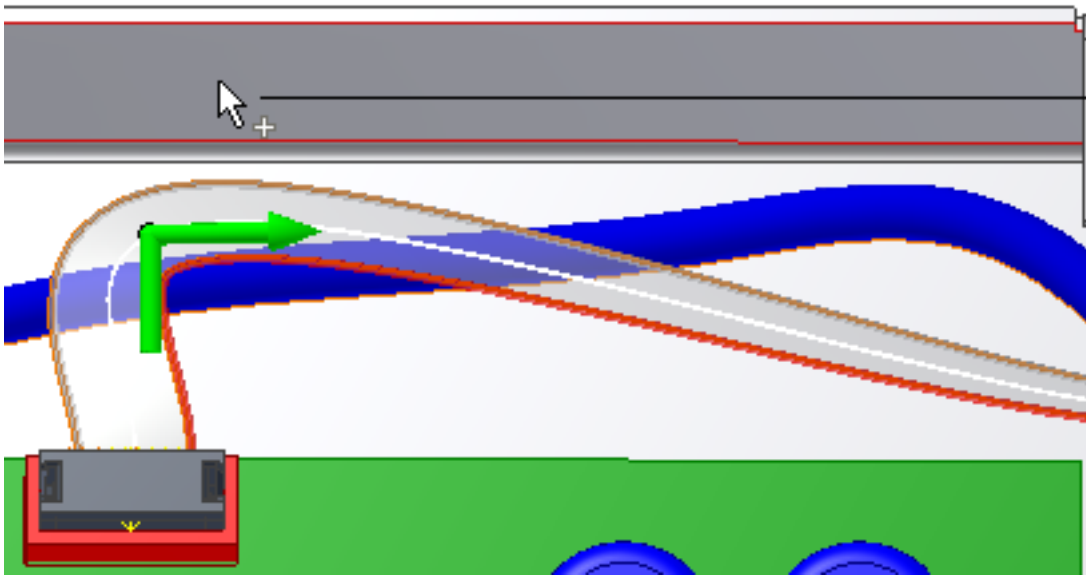


An arrow appears at the fold point indicating the direction of the next work point n the path of the ribbon cable. It is the correct direction for this fold.



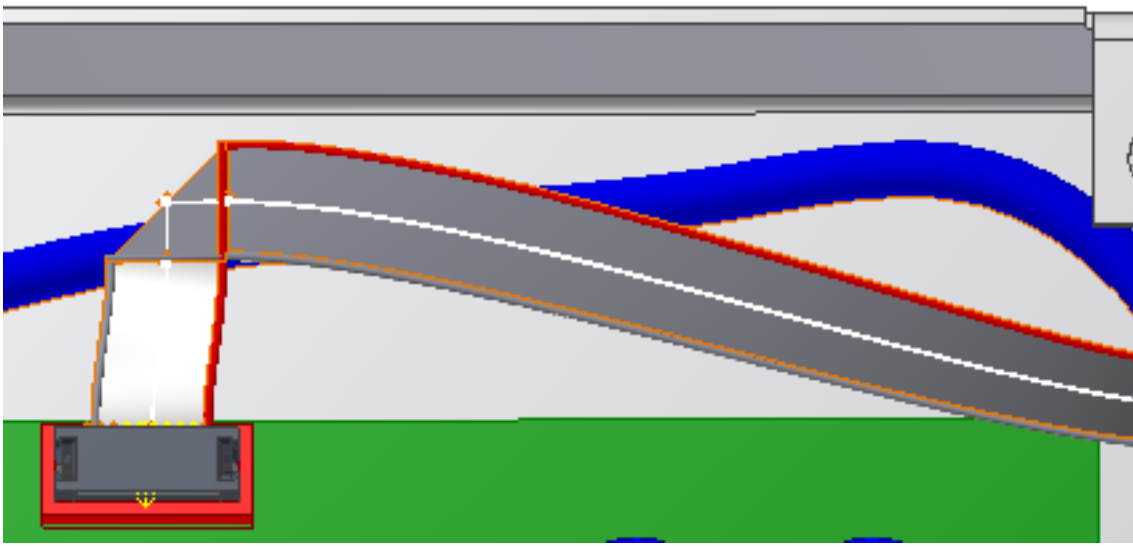
2  In the dialog box, under Alignment, click **Shaft**, and then click the face of the enclosure (1).

This action aligns the shaft perpendicular to the selected face. To align the shaft parallel to existing geometry, select an edge.



3 Click **OK,** and the fold is created with the specified direction and alignment.

NOTE Like editing other harness objects, you can right-click ribbon cable work points and use **Redefine Point** and **3D Move/Rotate** to move route points. You cannot rotate ribbon cable work points. In folds, you can manipulate only the work point used to create the fold.



4 In the browser, expand the Ribbon Cables folder and the ribbon cable. You should see Ribbon1, Fold1, and Ribbon2.

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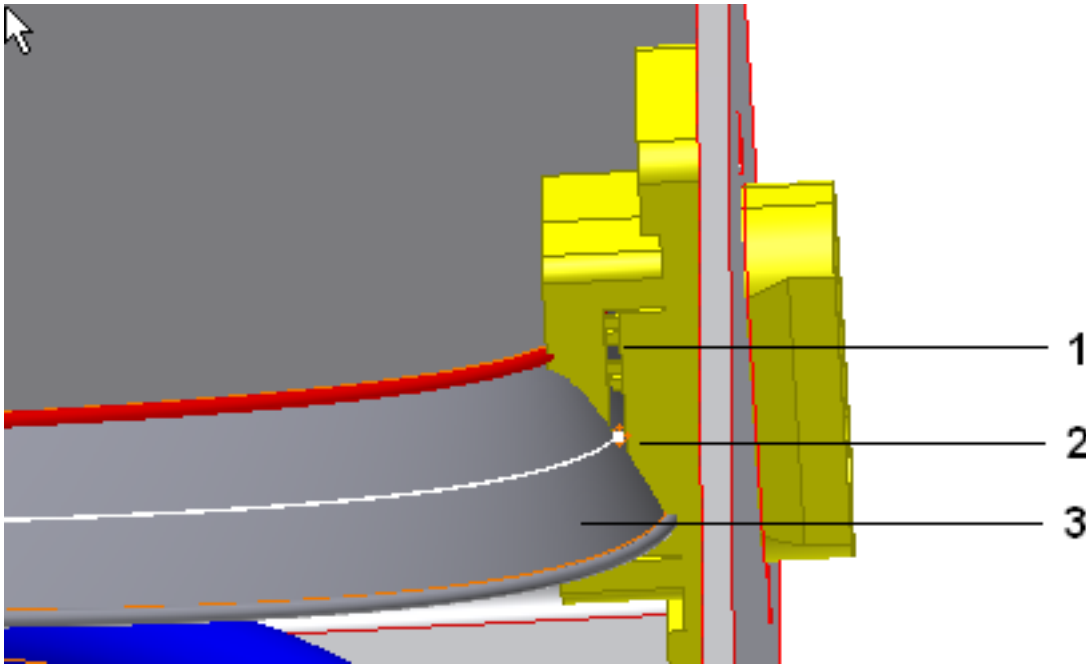
Adjust Twist

The ribbon cable body aligns parallel to the slot in the D-Sub Ribbon Cable connector. To adjust the ribbon cable at a selected point, use Twist Control.

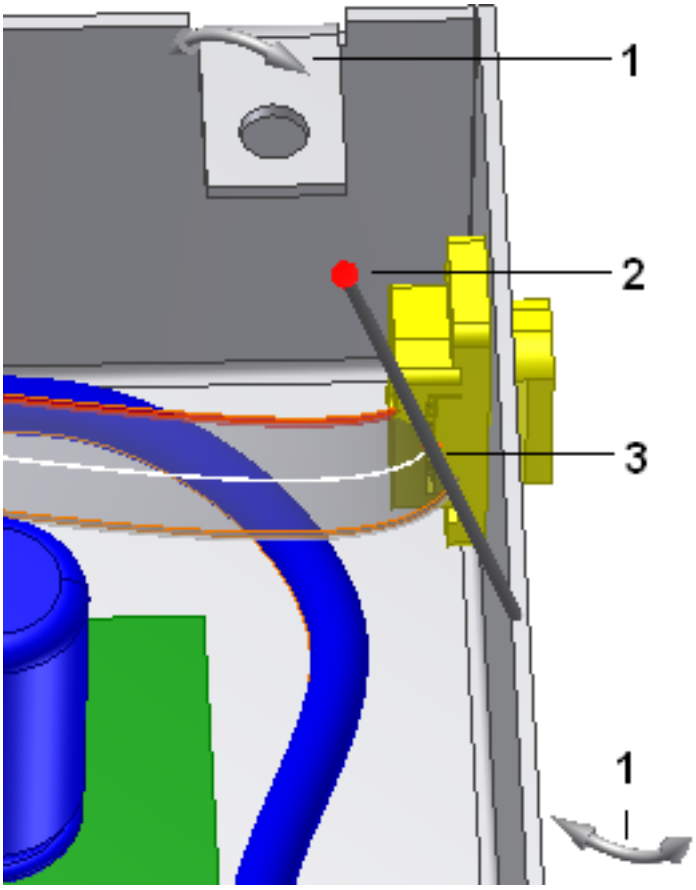
1 With the ribbon cable harness assembly still active, zoom into the D-Sub Ribbon cable. The cable body (3) is twisted in relation to the slot (1) on the connector.

2 Right-click the point you selected earlier (2) and select **Edit Twist**.



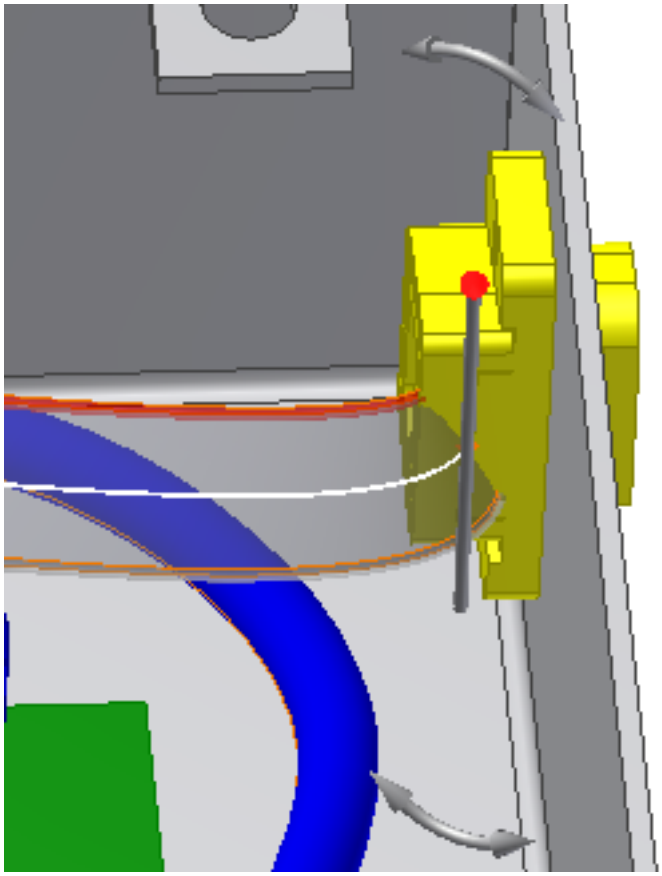
3 The Twist Control appears at the selected point.

The rotation handles (1) enable you to change the current orientation. The red ball (2) is the conductor one indicator. The orientation bar (2) represents the orientation of the surface on the ribbon cable at the selected location.

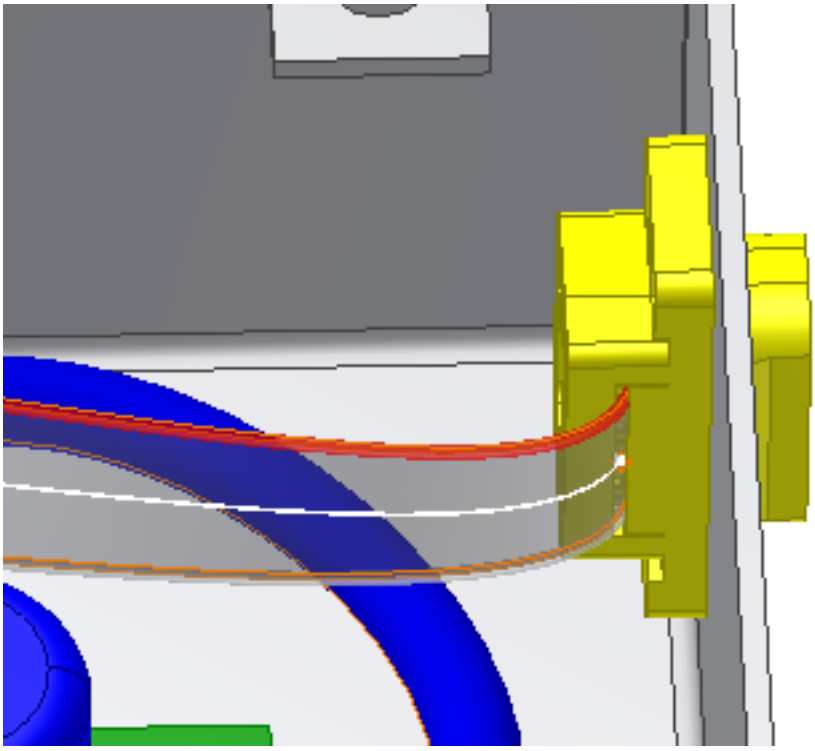


4 Click and drag a rotation handle until the ribbon body aligns with the slot, and release the mouse button.

NOTE If appropriate, click the minus (-) and plus (+) on the keypad to decrease and increase the size of the Twist Control.



5 Right-click and select **Apply**.
The ribbon cable body recomputes to the new location.



6 Return to the top-level assembly and save all files. This assembly is used in the next tutorial.

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Summary

In this tutorial, you:

- Define raw ribbon cables.
- Place connectors from content center.
- Create ribbon cables.
- Adjust ribbon cable orientation and shape.

Use your saved file from this tutorial to work with ribbon cables in the next tutorial

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Document Cable and Harness Assemblies

15

About this tutorial

Use the sample file provided to generate a simple bill of materials report for the parts and wires in the harness design.

Category	**Routed Systems**
Time Required	45 minutes
Tutorial Files Used	Enclosure_Assembly.iam saved from previous tutorial. Part_Splice_Wire_Cable_BOM.cfg Wire Run List.cfg

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

Objectives

- Generate reports.
- Create nailboards.
- Manipulate the harness.
- Change nailboard displays.
- Annotate nailboard drawings.
- Print nailboards and drawings.

Prerequisites

- Perform the previous tutorial, Ribbon Cables, and save that tutorial file for use in this tutorial.
- Know how to navigate the model space with the various view commands.
- See the Autodesk Inventor Help topic "Fundamentals" and the New User tutorials for further information.

Navigation Tips

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

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Create Part and Wire Bills of Material

Create a BOM report for HarnessAssembly1. Before you begin, verify that the harness assembly is saved and updated (**Update** command is not active).

NOTE You can also create a complete bill of materials and parts lists using standard Autodesk Inventor features.

1. Open **Enclosure_Assembly.iam** from the Cable & Harness subfolder. It is the same assembly you used in the previous tutorial.
2. Double-click **Harness Assembly1** to activate it for editing.
3.  Click **Cable and Harness tab ➤ Manage panel ➤ Report**.
4.  On the Report Generator dialog box, click **Create Report**.
5. On the Create Reports dialog box, click **Add File to List**. Then navigate to *Tutorial Files\Cable & Harness\Report Generator\Part_Splice_Wire_Cable_BOM.cfg*, and click **Open**.

   An output file **Harness Assembly1_Part_Splice_Wire_Cable_BOM.csv** is added to the list at the same time as the selected configuration file. It is located in the same directory as the associated harness assembly by default.
6. Double-click the output file name in the list, enter **Part_Splice_WireCable_BOM1** and click **Open**.

7 Click **OK** in the Create Reports dialog box.

The report and the output file generate with the specified name and location. The report displays in the Report Generator document window.

The path of the last selected configuration file is the default path for subsequent file add operations.

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Create Wire Run List Reports

Modify the sample wire run list report to include more information, including link types.

1  If Report Generator is not already open, on the ribbon, click **Cable and Harness tab ➤ Manage panel ➤ Report**.

2  On the Report Generator toolbar, click **Edit a configuration (.cfg) file**.

3 Select *Tutorial Files\Cable & Harness\Report Generator\Wire Run List.cfg* , and click **Open**.

4 On the **Info** tab, click in the input area before the existing text. Enter **Modified**, followed by a space, at the beginning of the text.

NOTE This information is included in the report only if **Print header** is selected on the **Format** tab.

5 On the **Filter** tab, verify that **Wire** and **Cable Wire** are selected as the objects.

6 On the **Table** tab, scroll to the right. Columns 5 - 10 use link types to get the RefDes and pin name for each wire end.

7 Leave the settings on the **Format** tab as they are. Then click **OK** to display the Save As dialog box. Name the file **Modified_Wire Run List.cfg**, and then click **Save**.

8  On the Report Generator toolbar, click **Create Report**.

9 On the Create Reports dialog box, click **Add File to List**. Then navigate to the *Modified_Wire Run List.cfg* file in the Report Generator directory, and click **Open**.

10 Click **OK**. The report is displayed in the Report Generator.

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Create Custom Reports

Create a custom report by first creating a configuration file from a blank configuration file.

1  If Report Generator is not already open, on the ribbon, click **Cable and Harness tab ➤ Manage panel ➤ Report**.

2  On the Report Generator toolbar, click **Edit a configuration (.cfg) file**.

3 Enter the name of the new configuration file as **custom_partbom.cfg**, and click **Open**.

4 On the **Info** tab, enter **Customized Part Bill of Material, no quantity**.

5 On the **Filter** tab, in **Select objects to report**, select **Part**.

6 In **Object filters**, select **All** for both the **Property Names** and **Property Values**.

7 Click the **Table** tab then click **Insert Column** twice to add two columns. Insert each column before **Column 1**.

8 Enter the values in the columns, as indicated in the following table. To enter data, click in the input box, and then enter the value. If an arrow displays, click the arrow and select from the list.

The first column lists the reference designator for the part, followed by the part number.

You can configure each column, and display any property.

	Column1	Column2
Column header	REFDES	PartNumber

	Column1	Column2
Link type	NONE	NONE
Name	REFDES	PARTNUMBER
Column width	15	15
Subtotal function	COUNT	NONE
Property data?	YES	YES
Units	NONE	DEFAULT
Round up	NONE	NONE
Print units suffix?	NO	YES
Print column?	YES	YES

9 On the **Format** tab, enter **2** as the first column to sort.

10 Select the following print options if they are not already selected:

`Print filter information`

`Print header`

`Print information`

`Print column headers`

11 Click **OK** then click **Save** to save the new configuration file.

12 Generate a report using this configuration file to view the resulting output. Each of the parts in the design display in an individual row.

13 Close the Report Generator.

14 Return to the top-level assembly and save all files.

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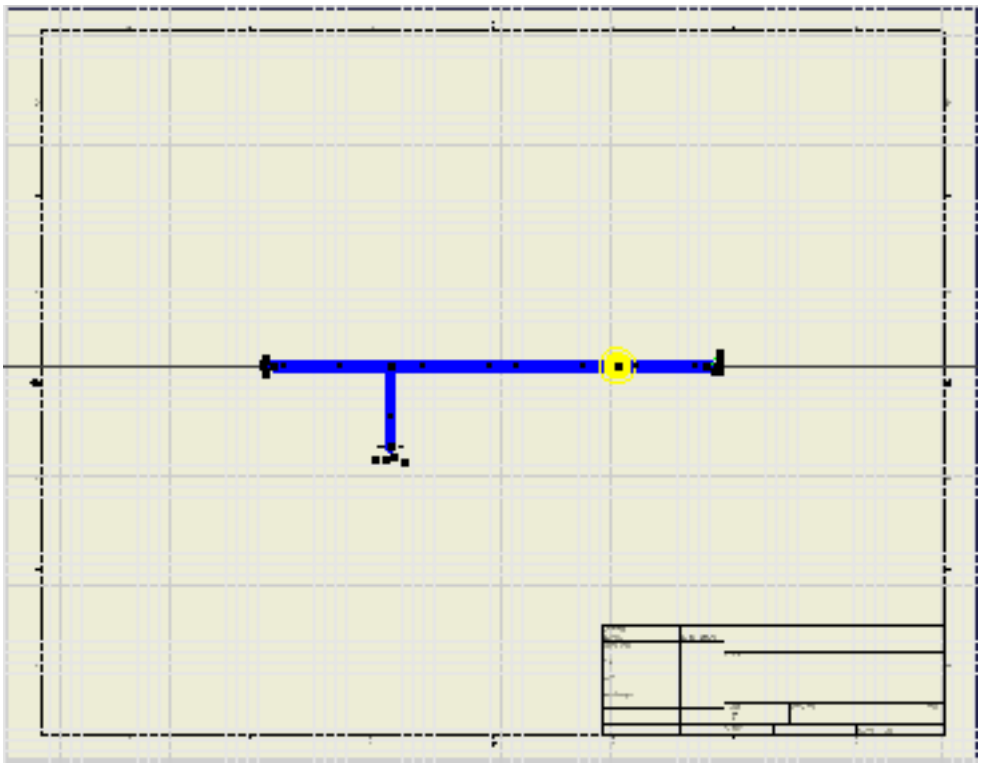
Create a Nailboard Drawing

View the default display settings for the active assembly and create a nailboard drawing.

1. Create a drawing using the Standard.idw template.
2.  On the Ribbon, click **Place Views tab ➤ Create panel ➤ Nailboard**.
3. On the Nailboard view dialog box, verify that the File is set to **Enclosure_Assembly.iam**.
4. Click the Nailboard View drop-down and select Harness Assembly1. Click **OK** to create the view.

   The nailboard drawing view is created and the nailboard sketch is active. The harness is placed in the drawing as shown.



5 Right-click in the graphics window, and select **Finish Sketch**.

6 Save your work using the file name *Harness Assembly1.idw* in the Cable & Harness subfolder.

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Move the Harness

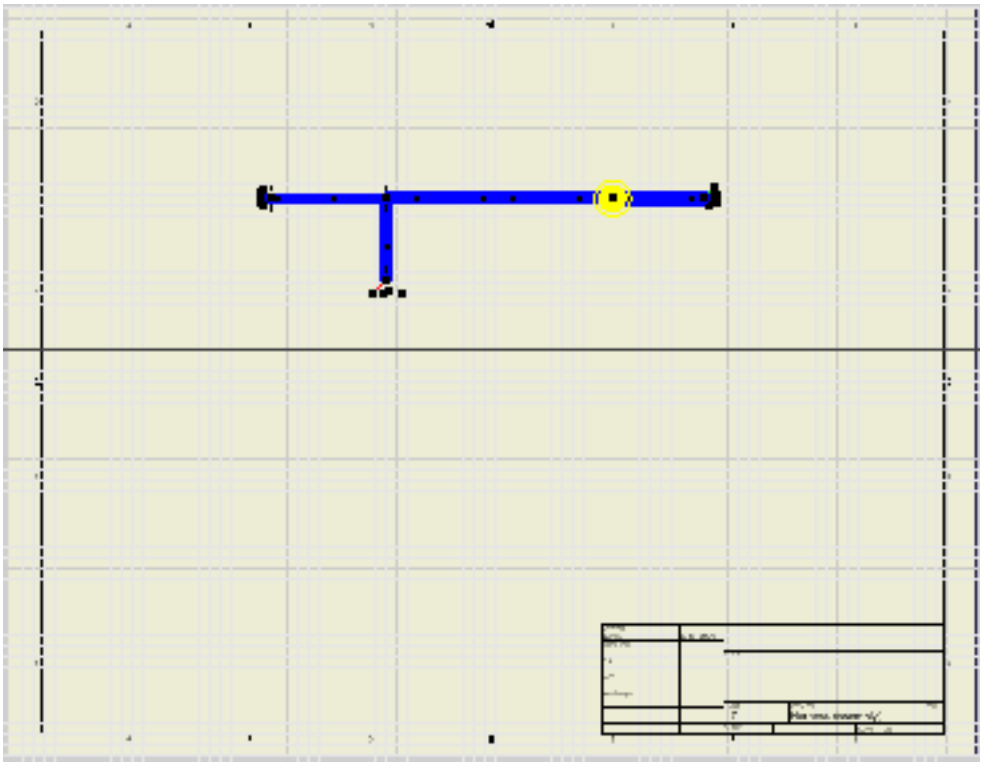
Move the entire harness to a new location on the drawing. For a move to occur, no pivot points can exist.

1  On the ribbon, click **Nailboard tab ➤ Edit panel ➤ Edit**.

2 Click the harness segment, and then drag the cursor to the location shown in the illustration.

3 Release the mouse to complete the move.

The harness is moved and redrawn in the selected location.



4 Undo to return the harness to its original position in the center of the drawing.

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Arrange the Harness Segments

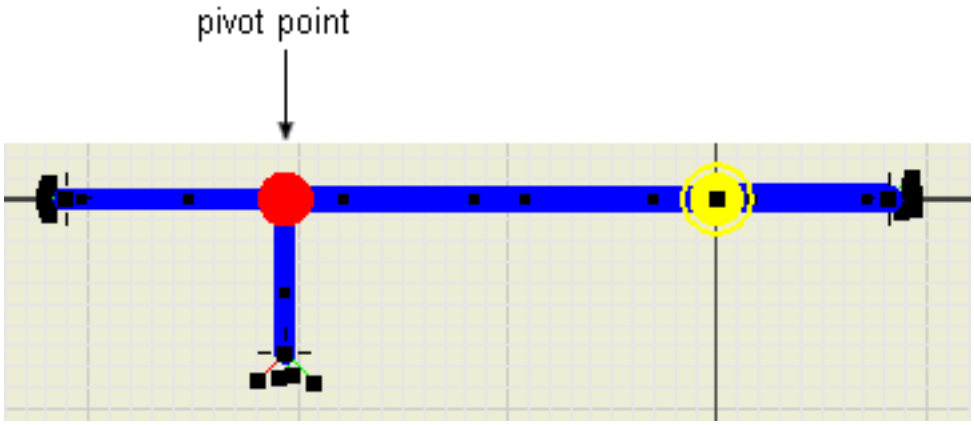
To arrange segments use Pivot to fix segment points, and then click and drag the ends of the segments around that pivot point.

Arrange the harness segments in vertical and horizontal directions. Use grid snap for precision placement.

1 Make sure that you are still editing the nailboard sketch.

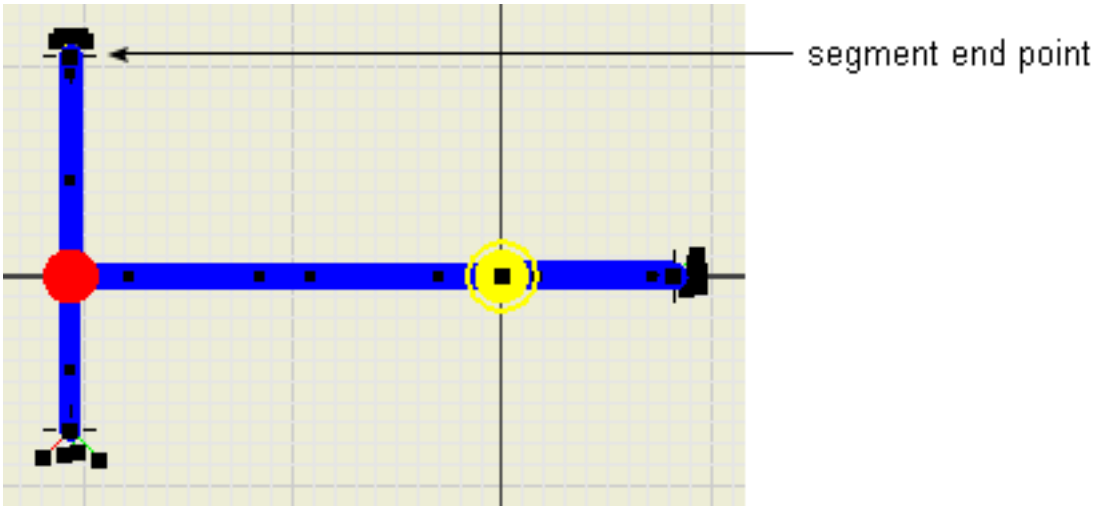
2 In the active nailboard drawing sketch, click **Nailboard tab ➤ Edit panel ➤ Pivot**.

3 In the graphics window, click the junction point for the harness branch.

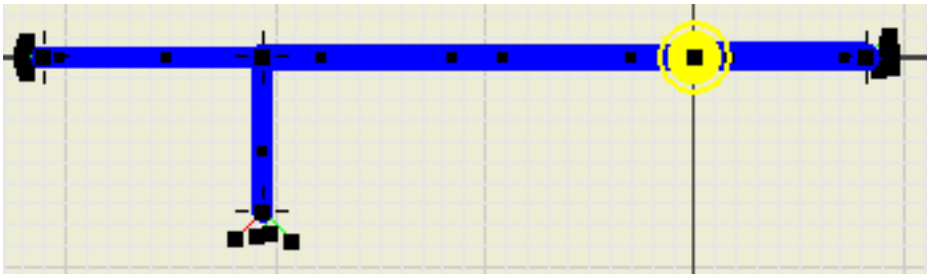


4 Click the point at the left end of the segment, and then drag until the segment snaps to the vertical position.

NOTE Snap is enabled by default. To disable snap, in the graphics window, right-click in the background, and then select **Disable Snap** from the context menu.



5 Return the segment to its original position.



6 Right-click, and select **Finish** from the context menu to remove the pivot point.

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Arrange the Wire Stubs

Manually rearrange individual wire stubs.

1 Zoom in on the end of the branched segment to see the distribution of the wire stubs.

2 Click and drag each of the wire endpoints to change their positions as shown.

A pivot point is not required when rotating wires about a segment endpoint.

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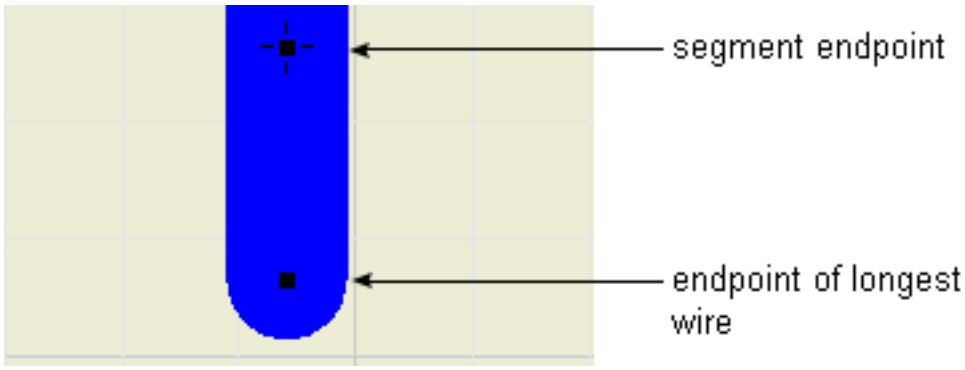
Automatically Arrange Wire Stubs

Use the Fan In and Fan Out commands to arrange wire stubs automatically.

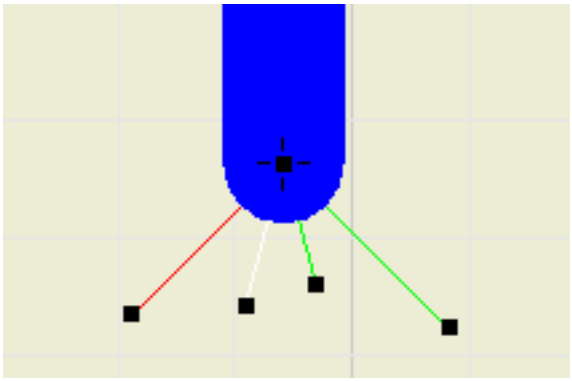
1. Right-click the segment endpoint or any of the four wire endpoints, and then select **Fan In/Out ➤ Fan In**.

   The wires are displayed using the color and diameter of the segment and the length of the longest wire.



2. Right-click the segment point or end point of longest wire, and then select **Fan In/Out ➤ Fan Out**.
3. In the Fan Out Wires dialog box, click **OK**.

NOTE Right-click a wire in the fanned out state. Select **Fan Out** from the context menu to change the sorting direction or angle between the outermost wires.



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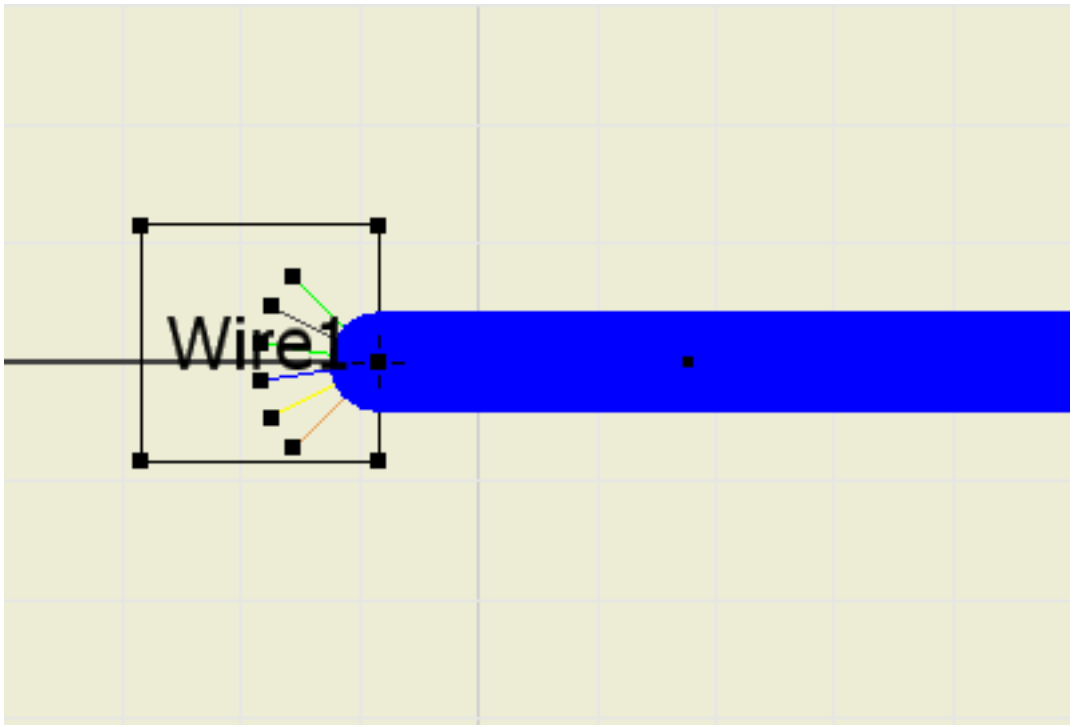
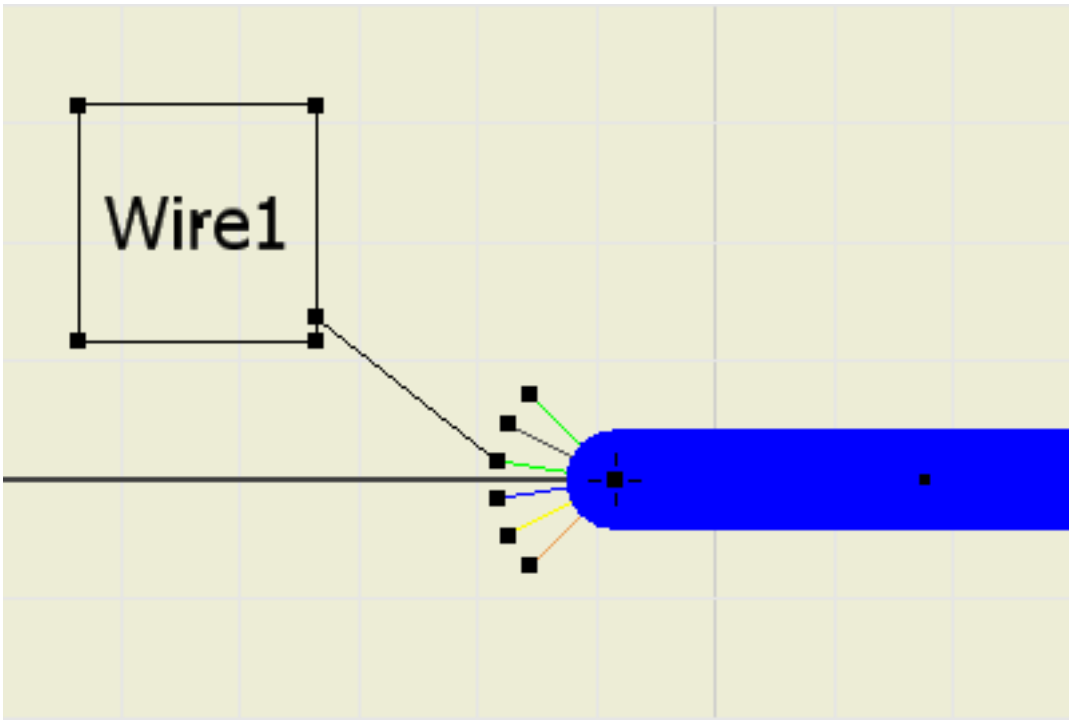
Arrange the Label

Set the label to show in the nailboard. Then manually rearrange individual labels by dragging.

1. Verify that the browser is set to Cable & Harness. If it is not, click **Model** at the top of the browser and select **Cable & Harness**.
2. In the browser, right-click **Harness Assembly 1**, and then select **Nailboard Settings**.
3. On the Nailboard Settings dialog box, under **Virtual Parts**, click the **Show Labels** check box, and then click **OK**.



4 Click and drag the text box to change its position as shown.



NOTE Label and font size can be adjusted on the label definition in the Cable and Harness Library.

5 For this exercise, change the default label display setting back to hide. Right-click the harness assembly, select **Nailboard Settings**, on the **Display** tab, clear the **Show Labels** check box.

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Change Segment and Wire Line Display

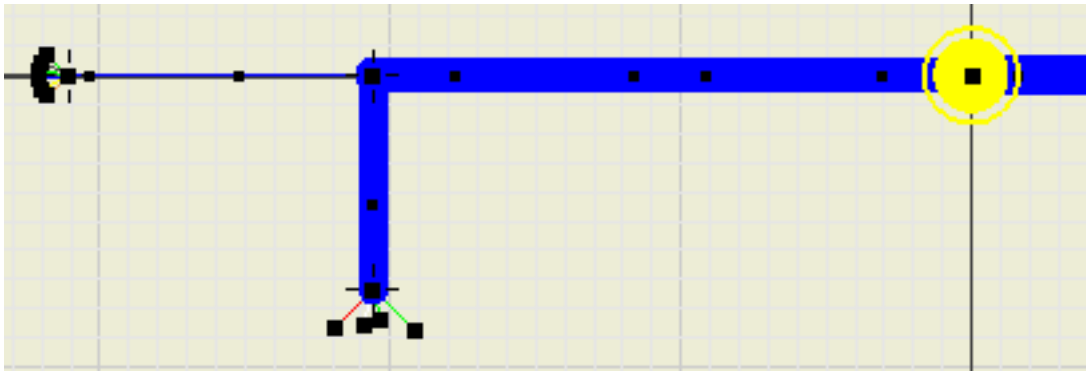
By default, nailboard wires and cable wires are set to display as thin lines. Segments are set to display as actual diameter.

Change the segment display.

1 In the active nailboard drawing sketch, select any of the segments to change.

2 Right-click, and clear the **Display as Actual Diameter** check box.

The segment changes.



3 Right-click again, and click the **Display as Actual Diameter** check box to return the display to the actual diameter.

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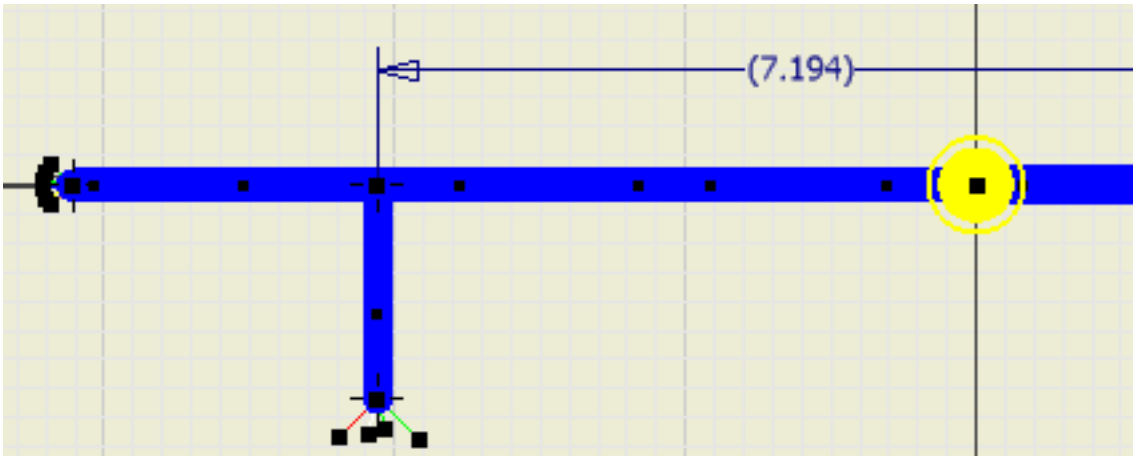
Dimension the Nailboard

1 On the ribbon, click **Nailboard tab ➤ Dimension panel ➤ Harness Dimension**.

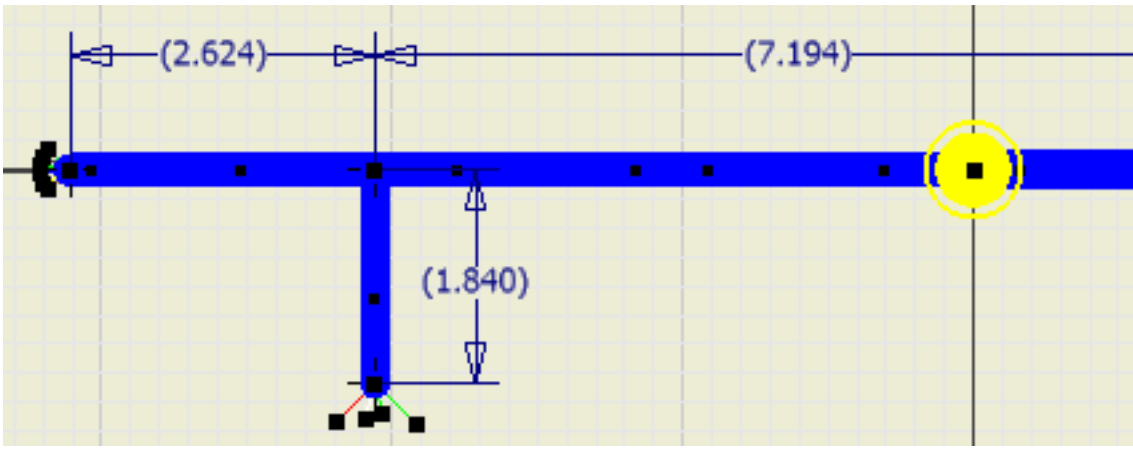
2 In the graphics window, click the point at the right end of the horizontal segment, and then click the segment point at the branch.

3 Drag to display the dimension, and then click to place as shown.

NOTE Your dimensions can differ, depending on your placement of the segment and segment points.



4 Dimension the other two segments as shown, then right-click and select **Done**.



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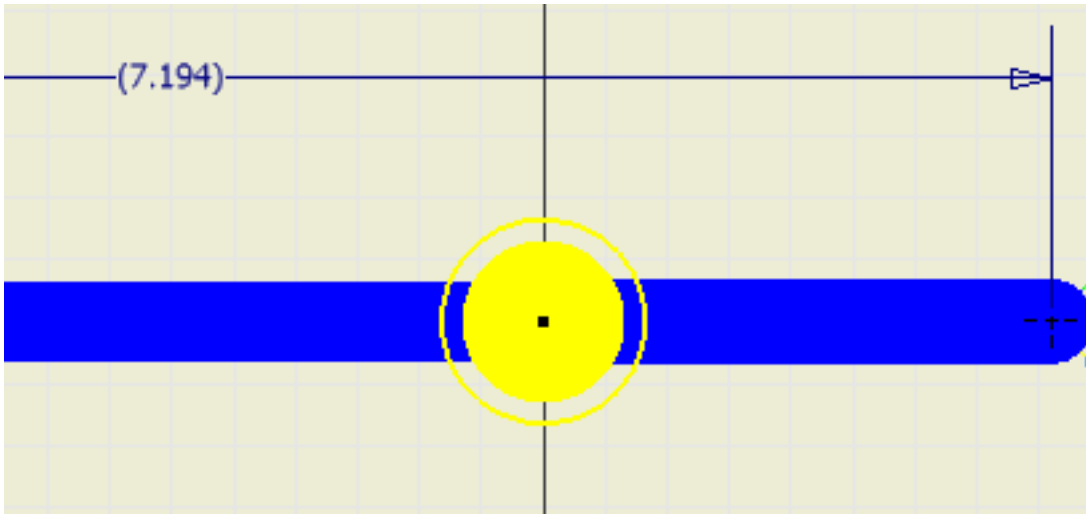
Add Properties

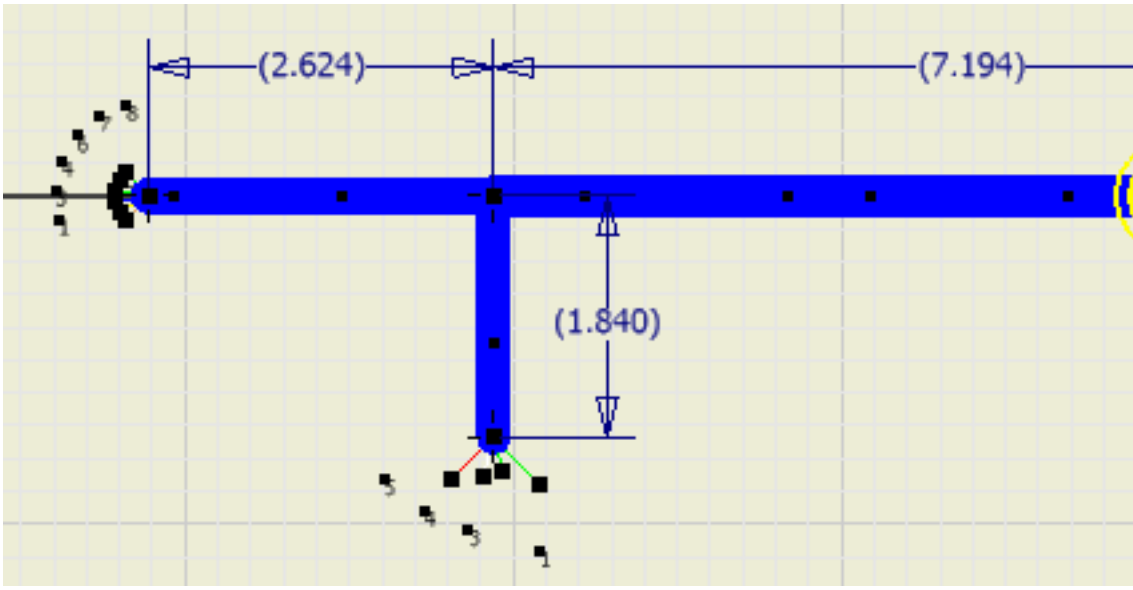
Use **Property Display** to view and display harness properties for wires, cable wires, pins, splices, and segments displayed on the drawing.

Add properties to pins, parts, and a selected wire.

1. In the graphics window, zoom in on the harness.
2.  On the ribbon, click **Nailboard tab** ➤ **Manage panel** ➤ **Property Display**.
3. On the Property Display dialog box, under **Select Filters**, click **All Pins**.
4. In the **Property Name** list, select **Pin Name**.
5. Under **Display Options**, select **Value Only**, and then click **Apply**.
6. In the graphics window, a rubber band line appears from the property text box, which attaches the cursor to the associated object. Click to place the property as shown.



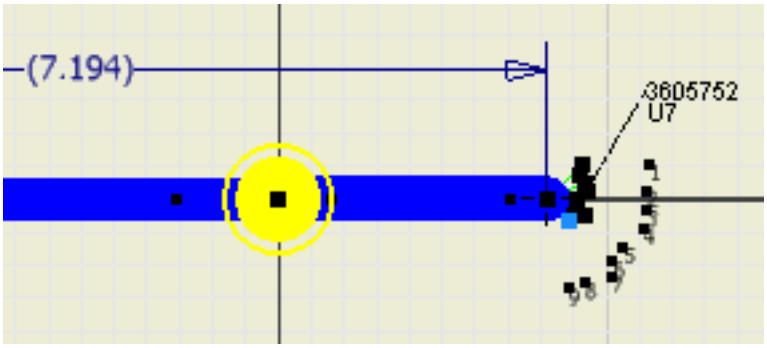
All properties are automatically placed based on this selection.



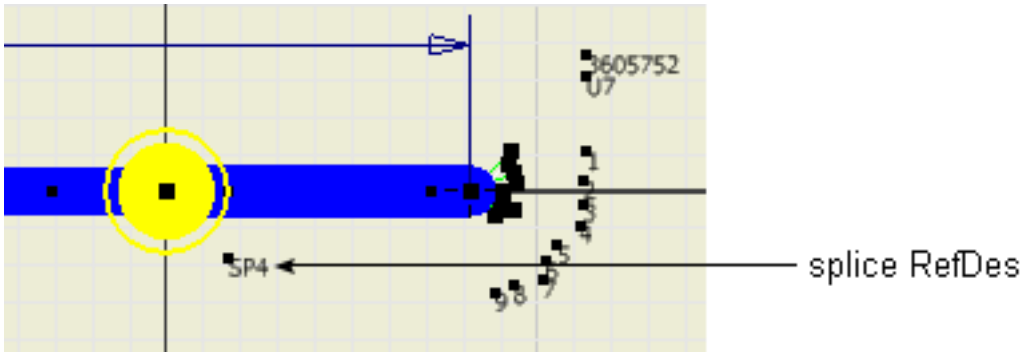
7 To place the part properties, select **All Parts**, select **RefDes** and **Part Number**, then select **Value Only**, and click **Apply**.

Use CTRL + click to select multiple items in the list.

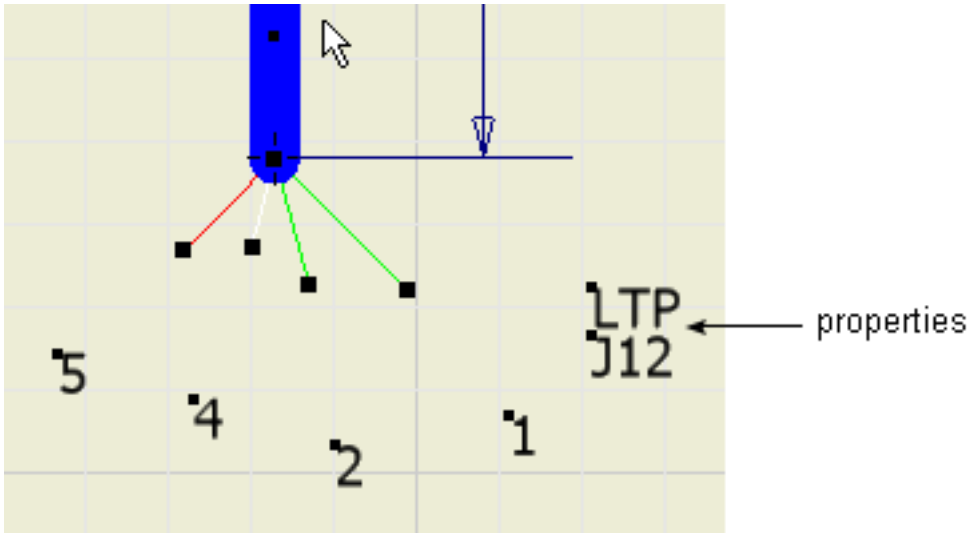
8 In the graphics window, click to place the part properties.



9 To place the splice properties, select **All Splices**, then select **RefDes** and **Value Only**, and click **OK**.



10 Use CTRL + click to select the **LTP** and **J12** properties, and drag them closer to the end of the branched segment.



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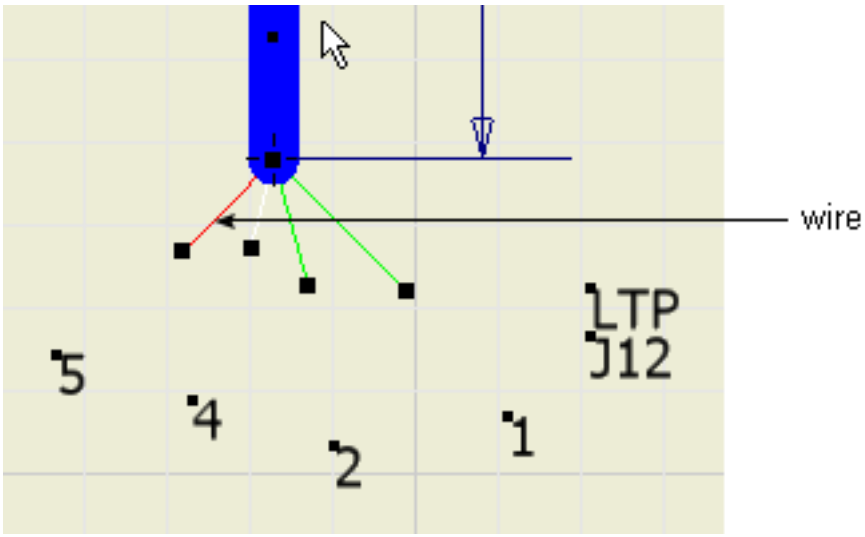
Add Properties on a Single Object

Place properties for a single object.

1 On the ribbon, click **Nailboard tab ➤ Manage panel ➤ Property Display**.

2 On the dialog box, select **Object**, and then click the wire shown in the following image.

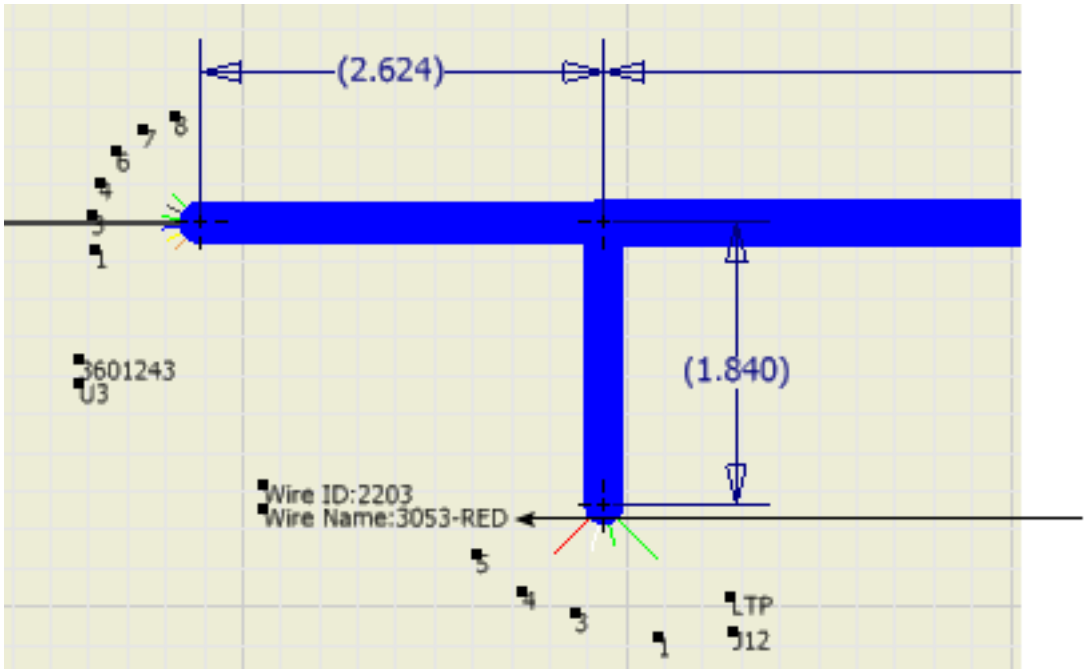


3 On the dialog box, scroll in the property list to select **Wire ID** and **Wire Name**.

4 Click **Name and Value**, and then click **OK**.

In the graphics window, the property appears on the opposite end of the segment, and not on the branch segment where the wire was selected.

5 Press the spacebar to switch the property to the branch end of the segment, and then click to place.



6 To see the associativity between the harness objects and the associated properties, drag the wire to a horizontal position.

When the wire moves, the properties associated to the wire also moves, keeping their original offset and relative position.

7 Return the wire to the previous position.

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Add Tables

1 If the nailboard sketch is active, right-click in the graphics window, and select **Finish Sketch**.

2 On the ribbon, click **Nailboard tab** ➤ **Create panel** ➤ **Create Table**.

3 On the Table dialog box, under **Select View**, click **Browse**.

4 On the Open dialog box, browse to and select the **Part_Splice_WireCable_BOM1.csv** file, click **Open**, and then **OK**. The default location is: *Tutorial Files**Cable & Harness*\.

5 To place the table, click in the upper left corner of the drawing.

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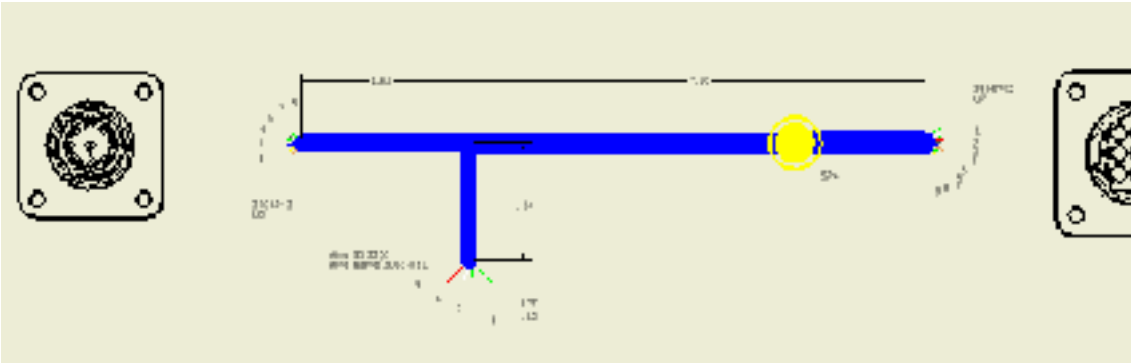
Place Connector Views

Place two connector views using default display settings and then override the default display settings for the other connector.

1  In the drawing view, click **Nailboard tab ➤ Create panel ➤ Place Connector Views**.

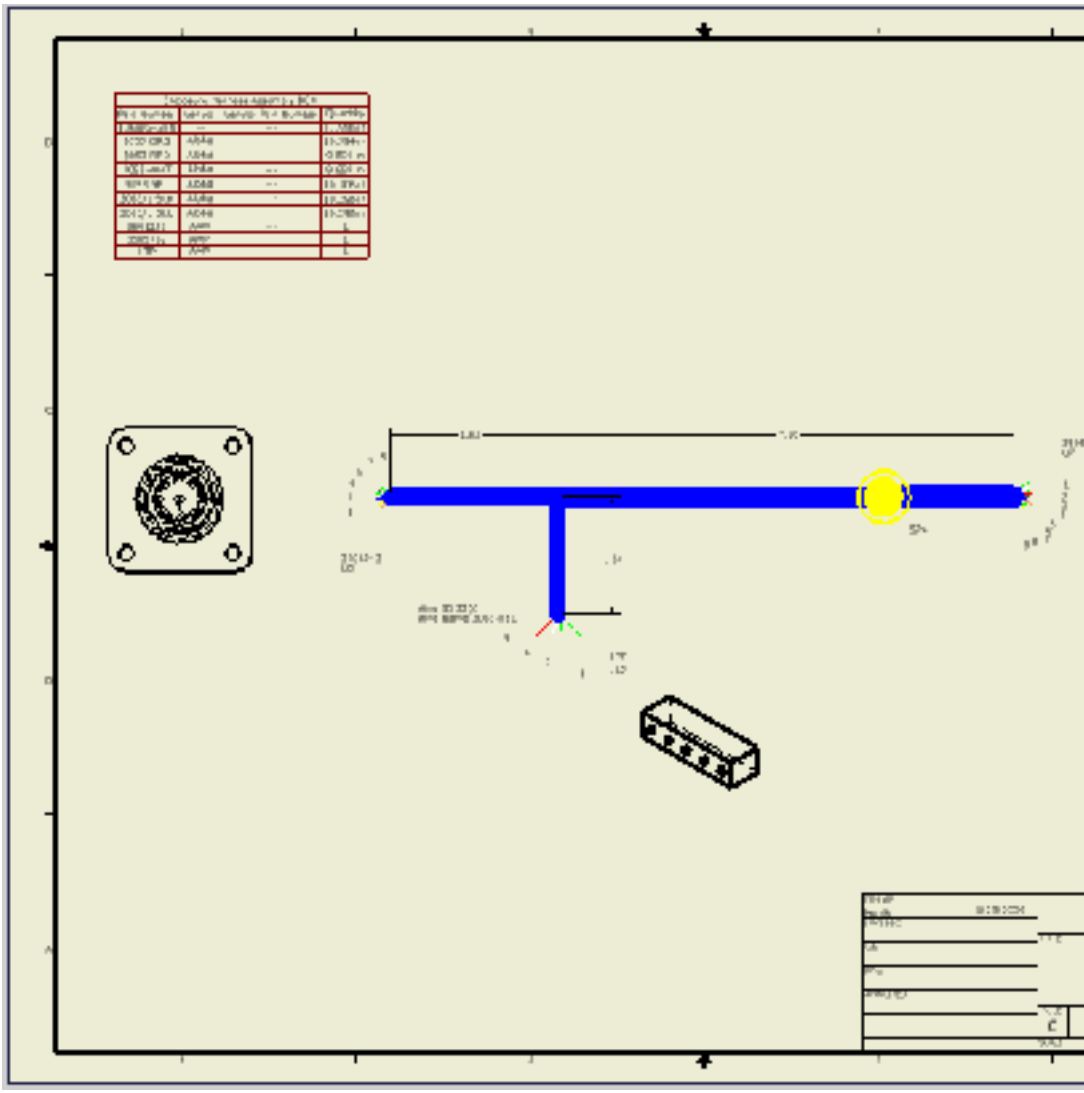
2 In the connector list, clear the check box for **LTP:1**, and then click **OK**.

The selected connectors are placed using the default display settings.



3 Click **Place Connector Views** again.

4 Under **Orientation**, click the arrow to select **ISO Top Right** from the list, and then click **OK**.



5 Save your work.

To change the views after placement, right click the view to change, and select **Edit View** or **Edit View Orientation**. To delete a connector base view, right-click the view and select **Delete**.

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Create an Assembly Drawing

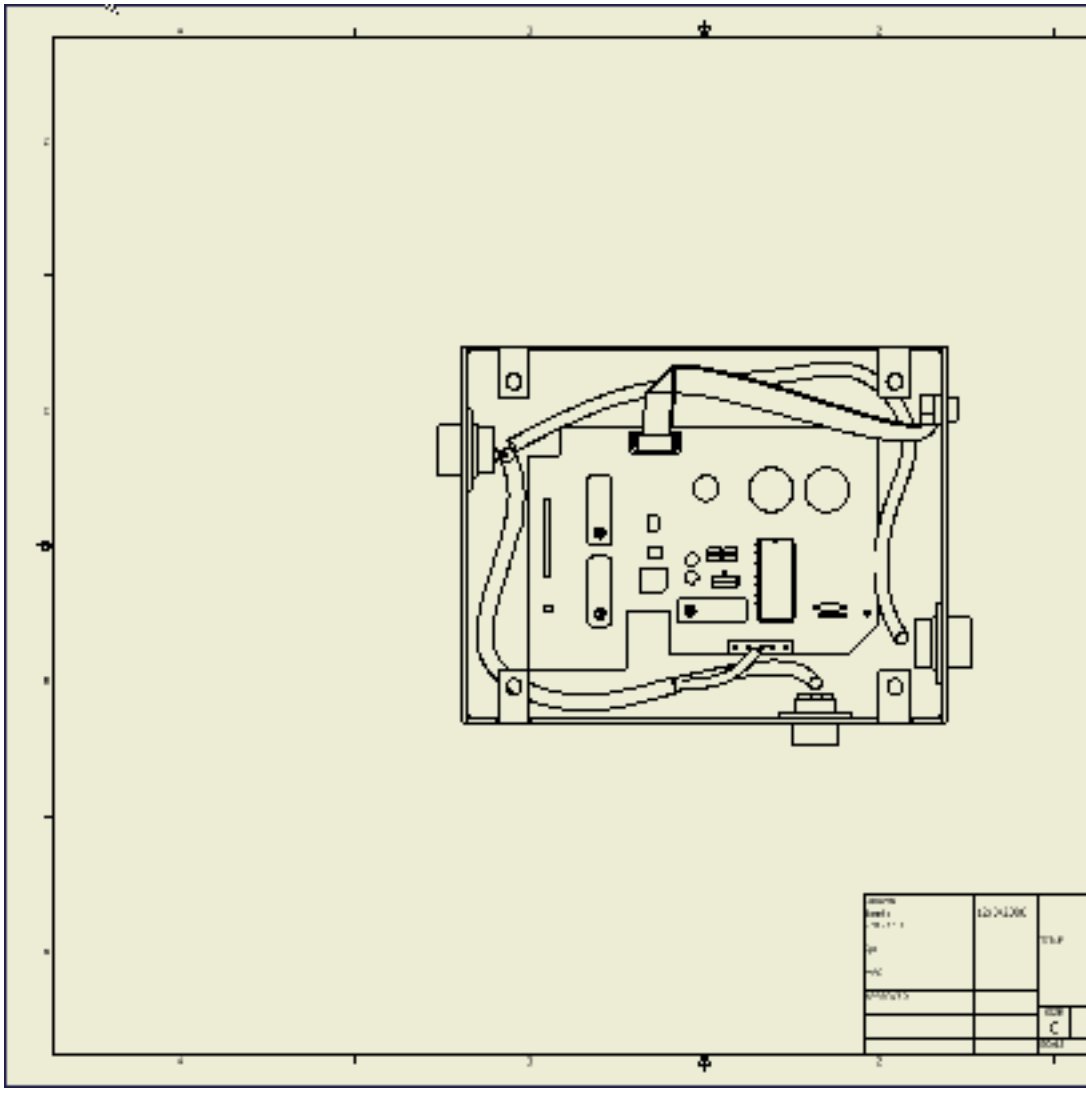
1 Click  ➤ **New**.

2 In the Create New File dialog box, click the Templates folder and select **Standard.idw** under the Drawing category, and then click **Create**.

3  In the drawing view, click **Place Views tab ➤ Create panel ➤ Base**.

4 On the Drawing View dialog box, ensure that **Enclosure_assembly.iam** is selected. The file is located in *\Tutorial Files\Cable & Harness*.

5 Click to place the view as shown, then right-click and select **Done**.

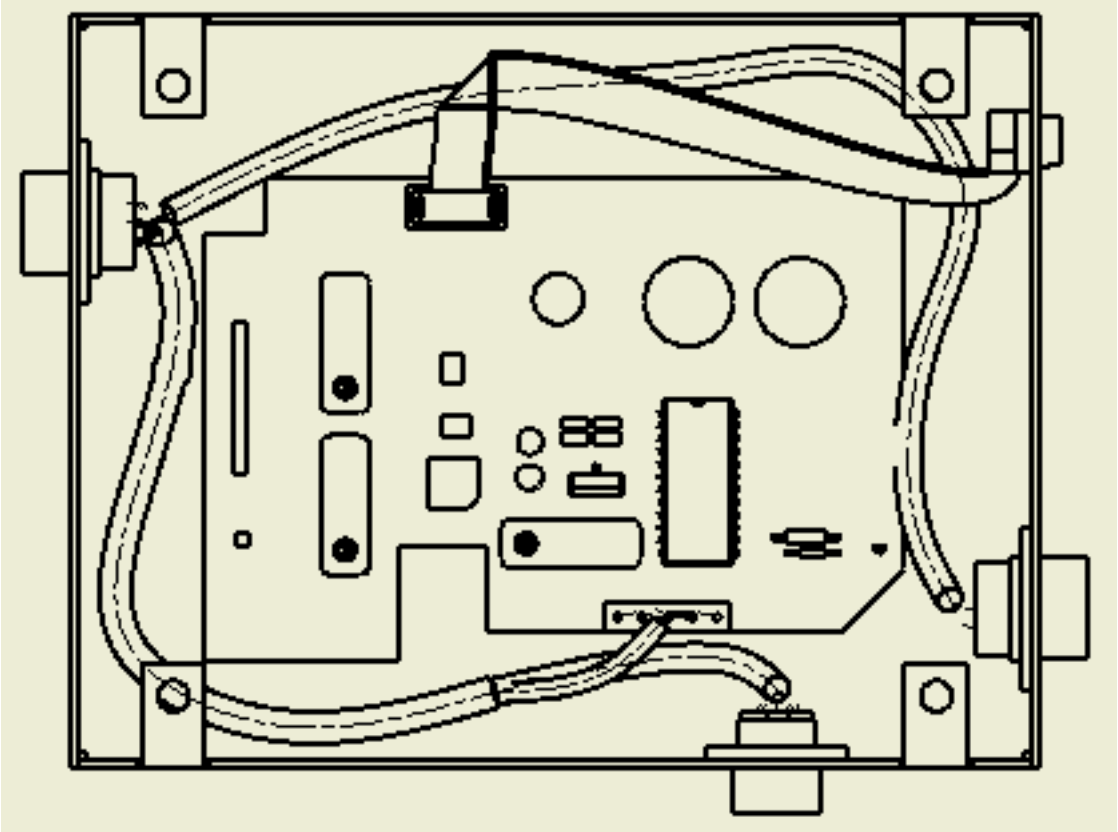


6 In the browser, expand **VIEW 1** and **Enclosure_assembly.iam**.

7 Right-click **Harness_Assembly1**, and then select **Include Centerlines ➤ Wires**.

The centerlines for all wires are displayed.

8 Right-click **Harness_Assembly2**, and then select **Include Centerlines ➤ Cables**.



9 Optionally, add dimensions and other annotations.

10 Save the file as Enclosure Assembly1.idw.

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Change Fan State and Displays

Create a nailboard view of the harness assembly from an open drawing, and change the fan state and display behavior.

1 Click  ➤ **New**.

2. In the Create New File dialog box, click the Templates folder and select **Standard.idw** under the Drawing category, and click **Create**.

3.  On the ribbon, click **Place Views tab ➤ Create panel ➤ Nailboard**.

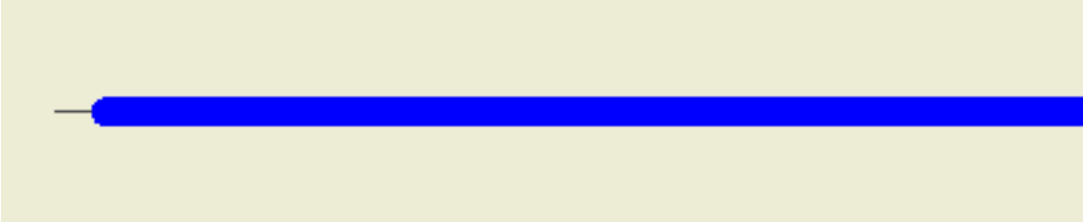
4. On the Nailboard View dialog box, ensure that the **File** is set to **Enclosure Assembly.iam** and **Nailboard View** is **Harness Assembly2.iam**.

5. Click the **Display** tab, and select **Fan In**.

6. Under **Appearance**, click **Display as Longest Wire**.

7. Click **OK** to create the nailboard view.

8. Right-click and select **Finish Sketch** for a better view of the cable wire stubs.



9. On the ribbon, click **Nailboard tab ➤ Edit panel ➤ Edit**.

10. In the browser, right-click **Harness Assembly2**, and select **Nailboard Settings**.

11. On the Nailboard Settings dialog box, click **Display as Segment** in the Fan In Display area.

12. Click **OK**.

NOTE This technique is the only way to change the fan in the display once the nailboard drawing is created.

13. Close the drawing file without saving changes.

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Summary

In this tutorial, you:

- Generate reports.
- Create nailboards.
- Manipulate the harness.
- Change nailboard displays.
- Annotate nailboard drawings.
- Print nailboards and drawings.

You have completed the successive tutorials for the Cable and Harness functionality. Go on to the next tutorial to learn about interoperability between Autodesk Inventor and AutoCAD Electrical.

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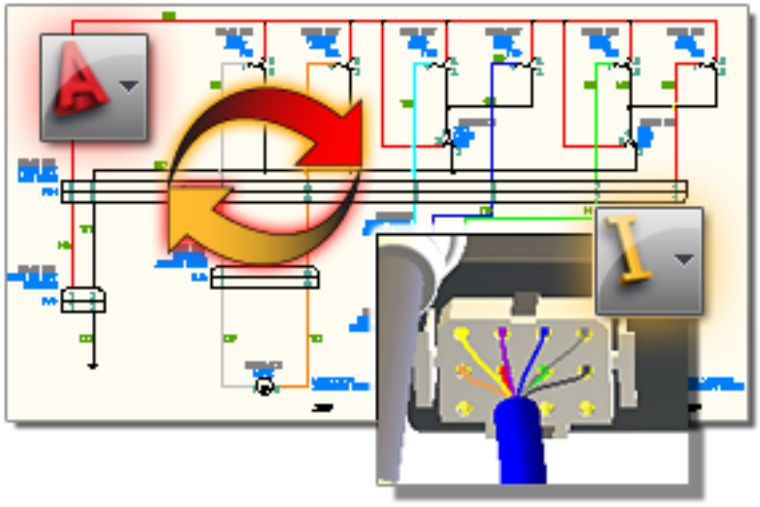
Interoperability: Inventor and AutoCAD Electrical

16

About this tutorial



Interchange data between AutoCAD Electrical and Inventor.

Category	**Routed Systems**
Time Required	35 minutes

Tutorial File Used 100500.iam

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

Objectives

- Use interoperability to digitally prototype and document your electrical designs.
- Apply the combined power of AutoCAD Electrical and Inventor to your designs.
- Exchange data between the two programs to explore new avenues of digital prototyping.

Prerequisites:

- Familiarity with both AutoCAD Electrical and Inventor is a plus but not necessary. This tutorial is designed to work whether you have only AutoCAD Electrical or Inventor, or if you have both programs.
- Navigate the model space with the various view tools.

Navigation Tips

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

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About this tutorial (continued)

In the first half of this tutorial, the data exchange direction is from AutoCAD Electrical to Inventor.

In the second half, the exchange direction is from Inventor to AutoCAD Electrical.

You do not need both programs to derive benefit from this exercise. The two XML files generated in this workflow are also included in the tutorial sample files. Therefore, if you have only one program, you can still perform the XML import operation.

If you have only Inventor, you can review the AutoCAD Electrical portion of the tutorial. Then you can perform the import steps and subsequent steps on the Import the AutoCAD Electrical data (page 330) page. You are directed to perform this import at the appropriate point.

NOTE This tutorial was created using AutoCAD Electrical 2011 and Inventor 2011 Professional (you need the Professional or Routed Systems versions for the Cable and Harness functionality).

Before proceeding, ensure the sample DWG file is accessible. For Windows XP, copy all files located in:

Documents and Settings\{username}\My Documents\Acade {version}\Aedata\Tutorial\Aegs\Interoperability

to

Documents and Settings\{username}\My Documents\Acade {version}\Aedata\Proj\Aegs

For Windows Vista, copy to:

Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Interoperability

For Windows 7, copy to:

Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

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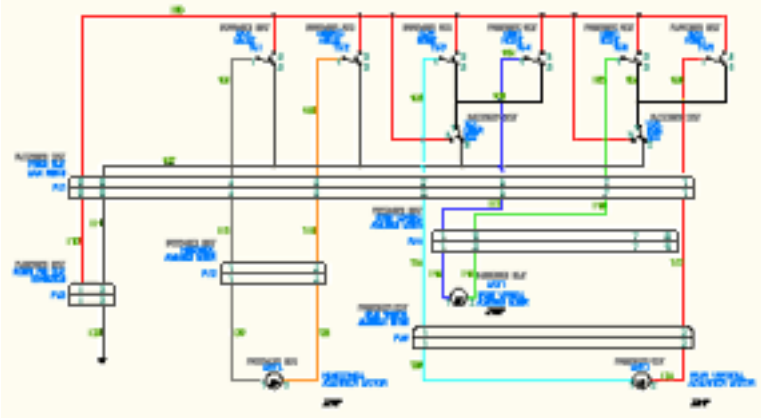
Part 1: 2D to 3D

In this half of the tutorial, you learn how to export your data from AutoCAD Electrical to Inventor. The sample DWG file is a wiring diagram used for a seat assembly. The assembly uses electric motors to provide adjustments to the seat position.

Open DWG

1. Start AutoCAD Electrical.
2. Open the Project Manager. If this window is closed select **Project** tab ➤ **Project Tools** panel ➤ **Manager**. 
3. Select **Open Project** from the project drop-down menu.
4. Select the project **ace_inv.wdp** then click **Open**.
5. Expand the *ACE_INV* project and then double-click **900501.dwg**.



Ensure that the drawing is in Model Space.

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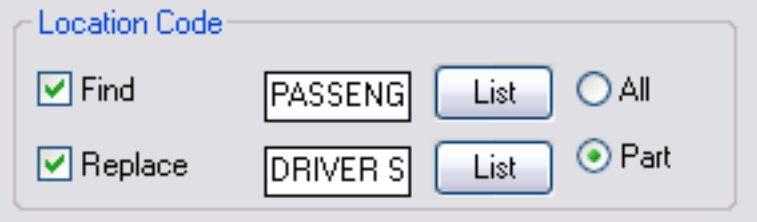
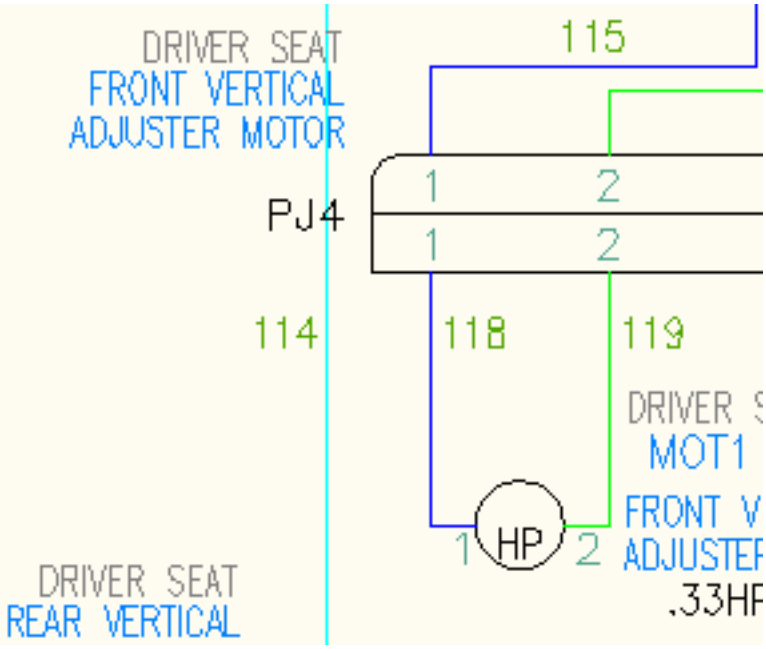
Rename Component Tags

Before you export this data to Inventor, you update the PASSENGER SEAT component tags. Assume that this drawing was copied forward and is now ready to modify and use for the driver seat.

1. Select **Schematic ➤ Edit Components ➤ Find/Edit/Replace Component Text**. 
2. In the Find/Edit/Replace Electrical Component Text dialog box, select **Active drawing (all)**. Click **OK**.
3. In the Find/Edit/Replace - this Drawing (all) dialog box, set the following options and take the following actions.
   - In the **Location Code** group, select **Find**.
   - In the Loc values dialog box, click **List** and then select **PASSENGER SEAT**. Click **OK**.
   - Select **Replace**, and then enter **DRIVER SEAT**.



- Click **Start Search,** and review the results in the Match 1 of 16 dialog box.
- Click **Replace All**, and then click **Yes, Make Changes**. The component tags change to DRIVER SEAT.



- Click **Cancel**.

4 Save the drawing.

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Export to XML

Next, you export the electrical data contained in your AutoCAD Electrical digital prototype to an XML file. You use this XML file later to import the data into Inventor.

1. Select **Import/Export Data ➤ Export ➤ Inventor**.
2. In the Autodesk Inventor Professional Export dialog box, ensure that **Active Drawing** is selected. Click **OK**.
3. In the Autodesk Inventor Professional XML File Export dialog box, specify the **Automotive** folder. This folder is located in the Inventor Tutorial Files directory, within the Inventor 2011 location. For the file name, use **driverseat_from_ace**. Click **Save**.

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Set the Project

1. Start Inventor.
2. Select **Get Started ➤ Launch ➤ Projects**.
3. Click **Browse**.
4. Browse to *Tutorial Files/Automotive* folder and select **interop.ipj**. Click **Open**.
5. In the Projects editor, click **Done**.

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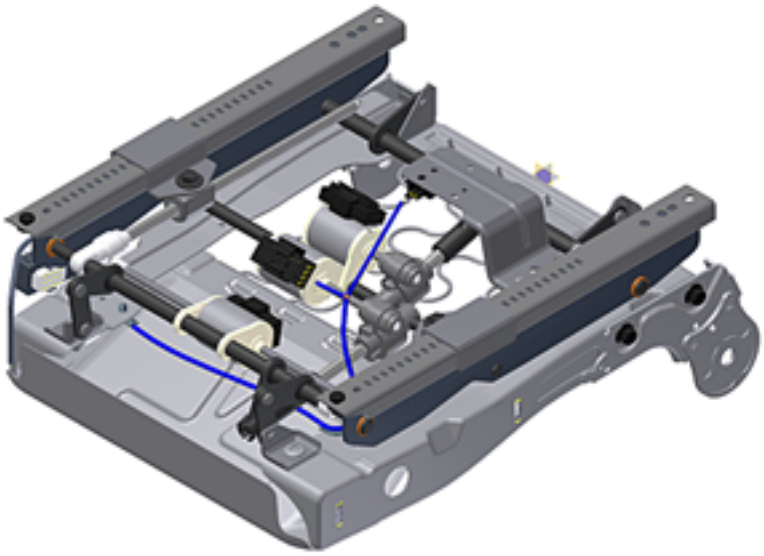
Open the Dataset

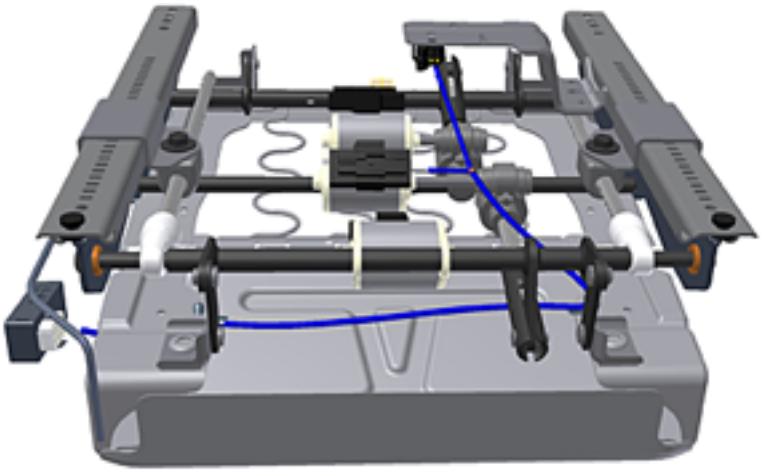
Open **100500.iam**. The file is contained in the 1000 folder. The model opens in the Default design view representation.

This sample has been stripped down to reduce data size. The complete seat looks like the following image.

Switch to the **Electrical** design view representation.



Orbit and zoom your view as you progress through the workflow, as needed. It can be helpful to approximate the following view as you get started with the workflow.



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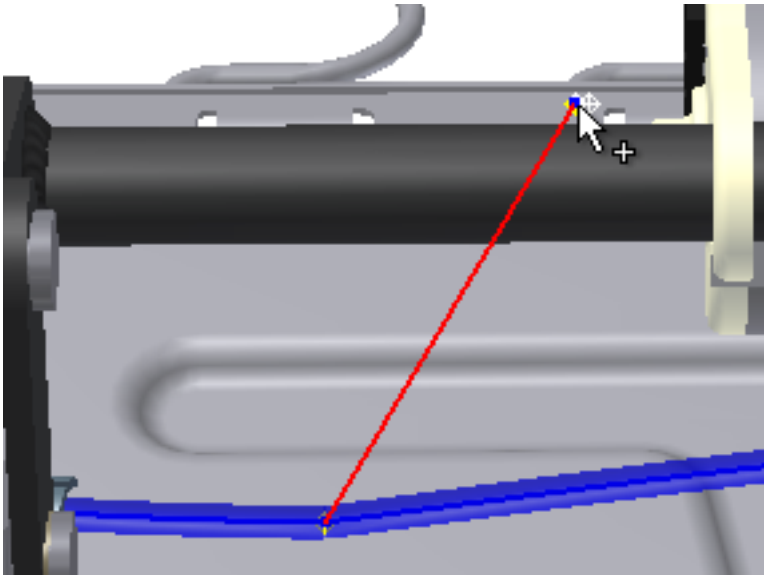
Add Harness Segments

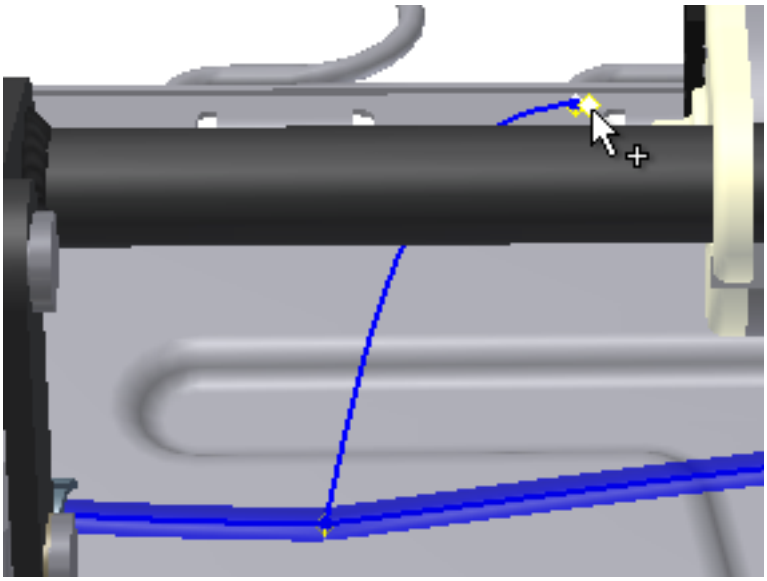
Now you add the two harness segments.

1. In the browser, double-click **Harness1** to edit the harness. Be careful to edit the harness assembly rather than the harness part.
2. Select **Cable and Harness ➤ Create ➤ Create Segment**. 
3. Select the existing segment near the front of the seat to place the first point. The exact selection location is not critical for this exercise.

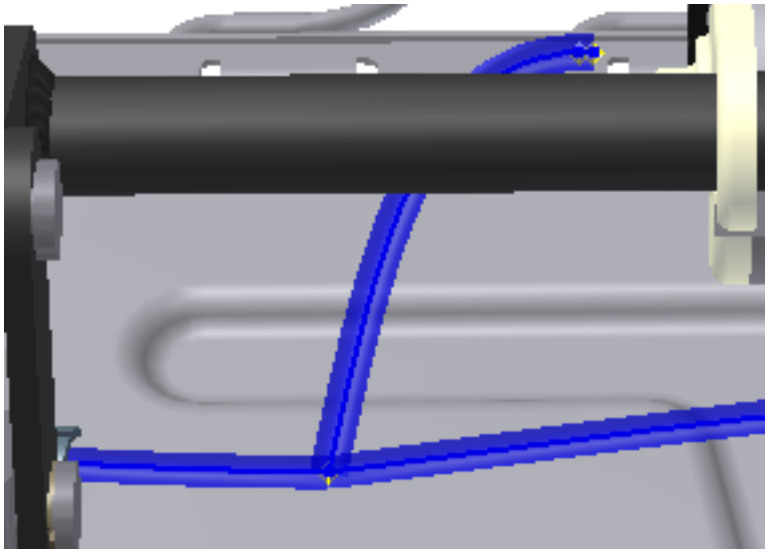
4 Select the existing work point to set the next segment point.



5 Select the other work point to set the final point. These two points are very close together. Zoom in to see them clearly.



6 Right-click and select **Continue**. The first segment is created.



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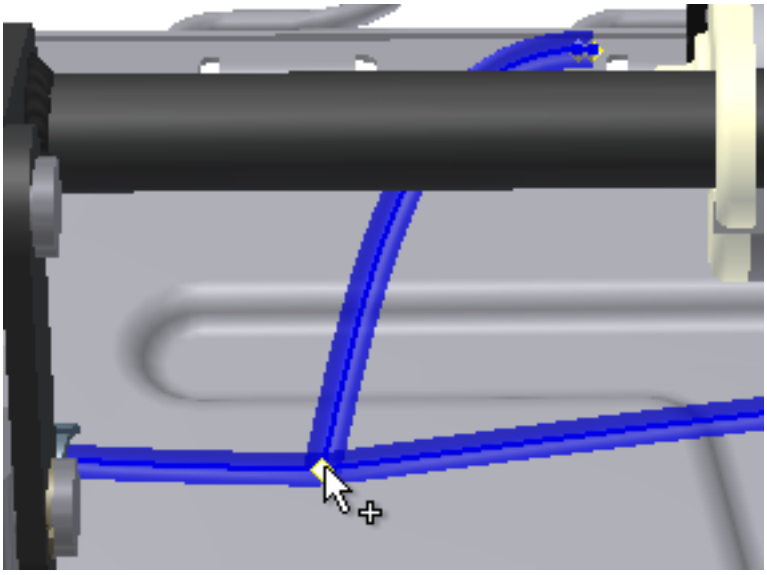
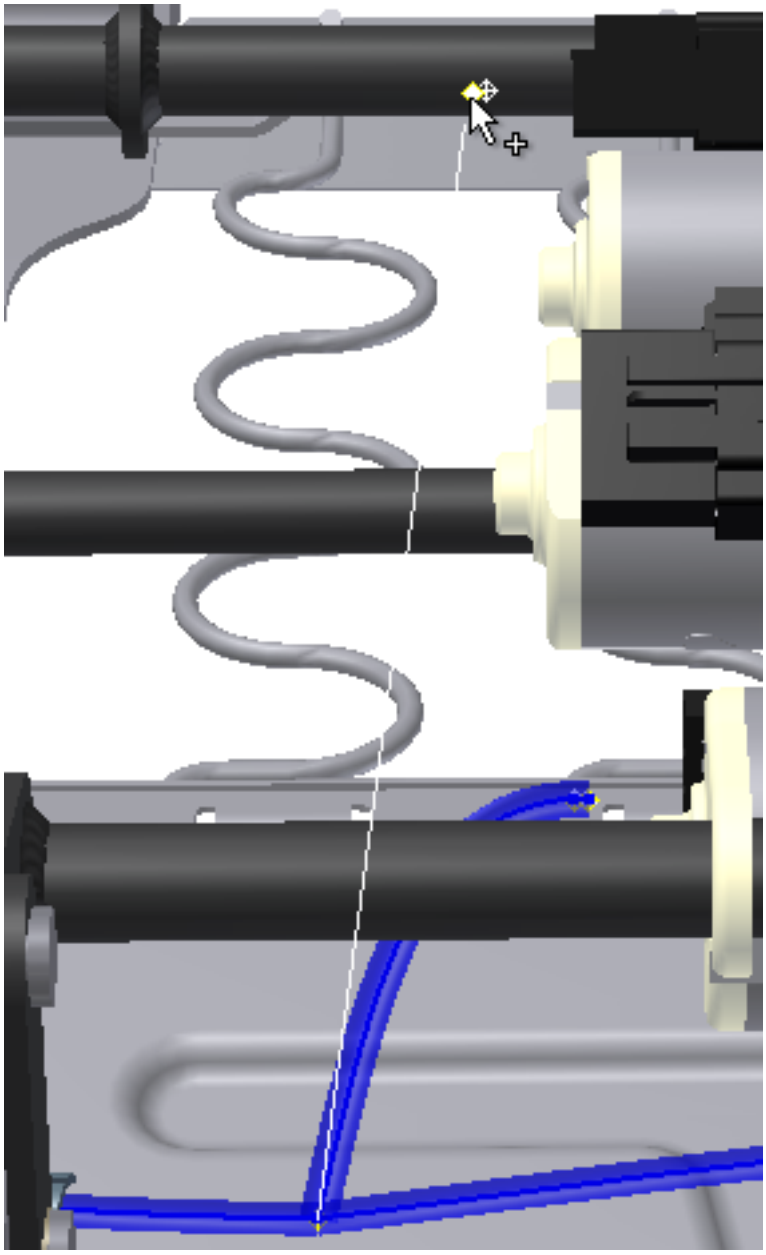
Add Harness Segments (continued)

Next, you add a second segment. This segment begins in the same location as the previous segment. Because you selected Continue on the context menu, Create Segment is still active and ready to create another segment.

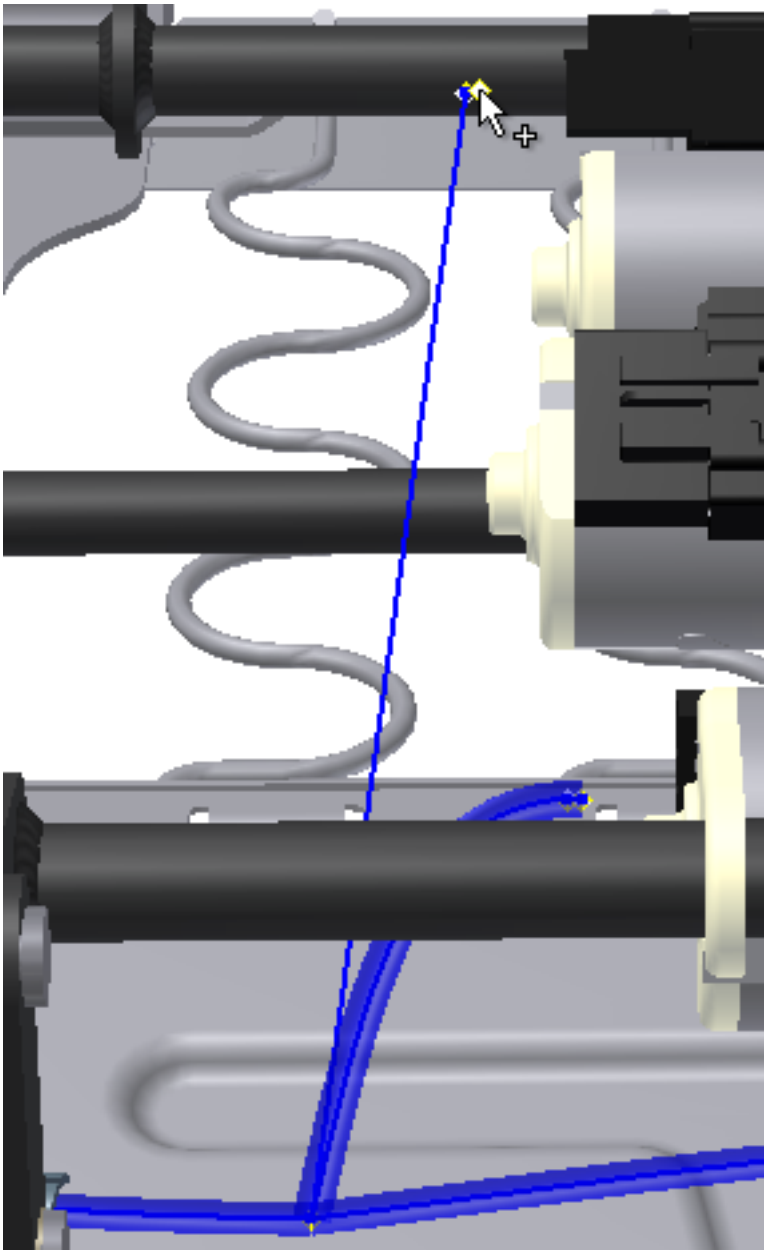
1 Select the segment point that you added previously.



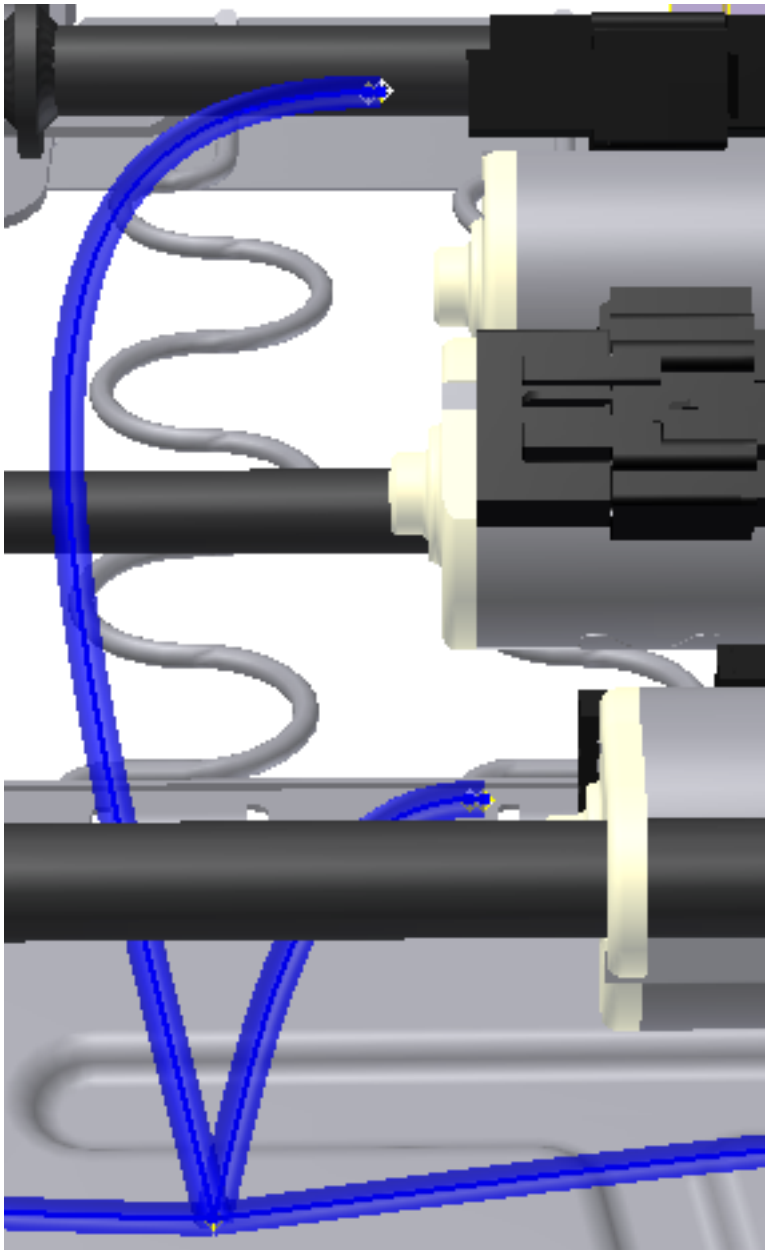
2 Select the existing work point.



3 Select the other work point.



4 Right-click and select **Finish**. The segment is created.



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Import AutoCAD Electrical Data

Apply the AutoCAD Electrical data to the Inventor 3D model.

1. Select **Cable and Harness ➤ Manage ➤ Import Harness Data**.

2. Click the Browse button next to the Harness Data File field.
3. In the Select Wire List Data File dialog box, locate and select **driverseat_from_ace.xml**, the XML file you exported from AutoCAD Electrical. The file is in the Automotive folder. Click **Open**.

   NOTE If you do not have AutoCAD Electrical, use **driverseat_from_ace.xml** provided in *Tutorial Files\Automotive\XML_delivered*.

4. Click **OK** in the Import Harness Data dialog box.

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Issues

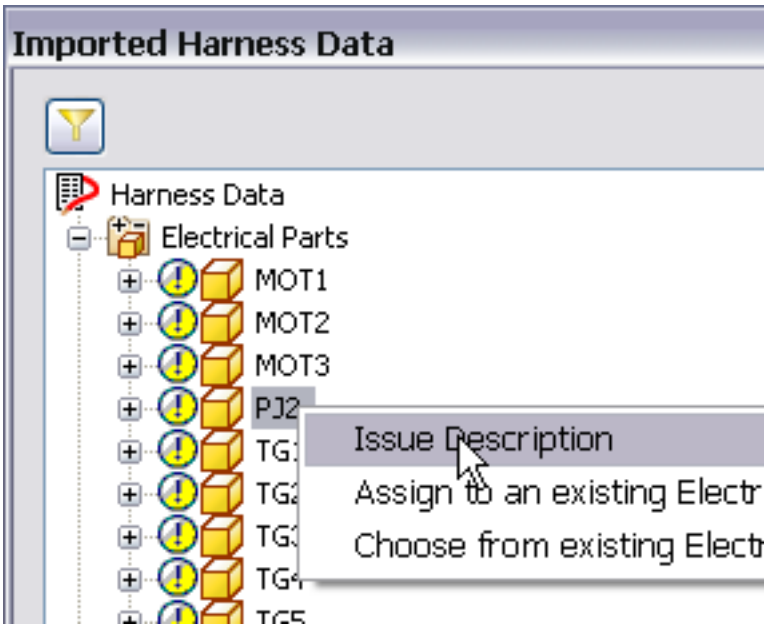
In the Imported Harness Data dialog box, the browser nodes contain the electrical components and wires imported from AutoCAD Electrical.

1. In the Imported Harness Data dialog box. click **Filter**  and then select **Show Issues Only**.

Only items with issues display in the dialog box. The items are identified with the Issue icon. 

There are many issues because the Inventor sample assembly does not contain many of the components contained in the AutoCAD Electrical drawing, as reflected in *driverseat_from_ace.xml*. For many of the RefDes in AutoCAD Electrical, there are no matching RefDes in Inventor. The fact that various components and RefDes are not present means that connecting wires also have issues. Although RefDes are not present, you can successfully complete the exercise. Data can be missing or incomplete in a real-world design process, but it is acceptable for a given point in the workflow.

Scroll to the top of the item list, right-click **PJ2**, and then select **Issue Description**.



The issue description describes the problem and offers solutions. Review the information and then close the issue description.

TIP Click Help in the *Imported Harness Data* dialog box to open a reference topic that describes various elements and features in the dialog box.

Next, you use functionality on that same context menu to assign the missing RefDes.

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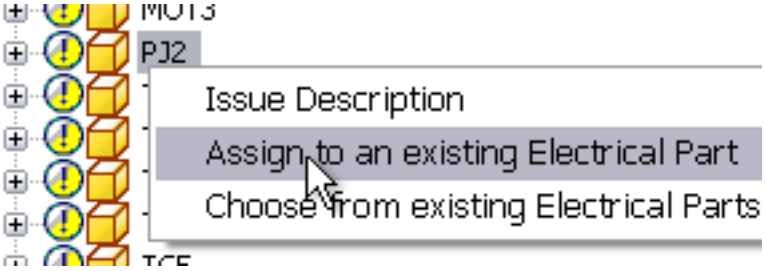
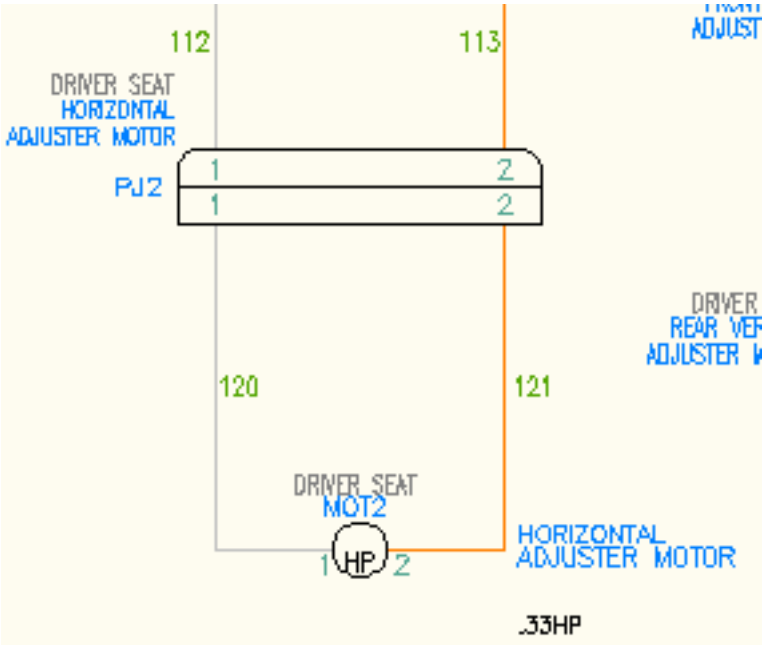
Assign Missing RefDes

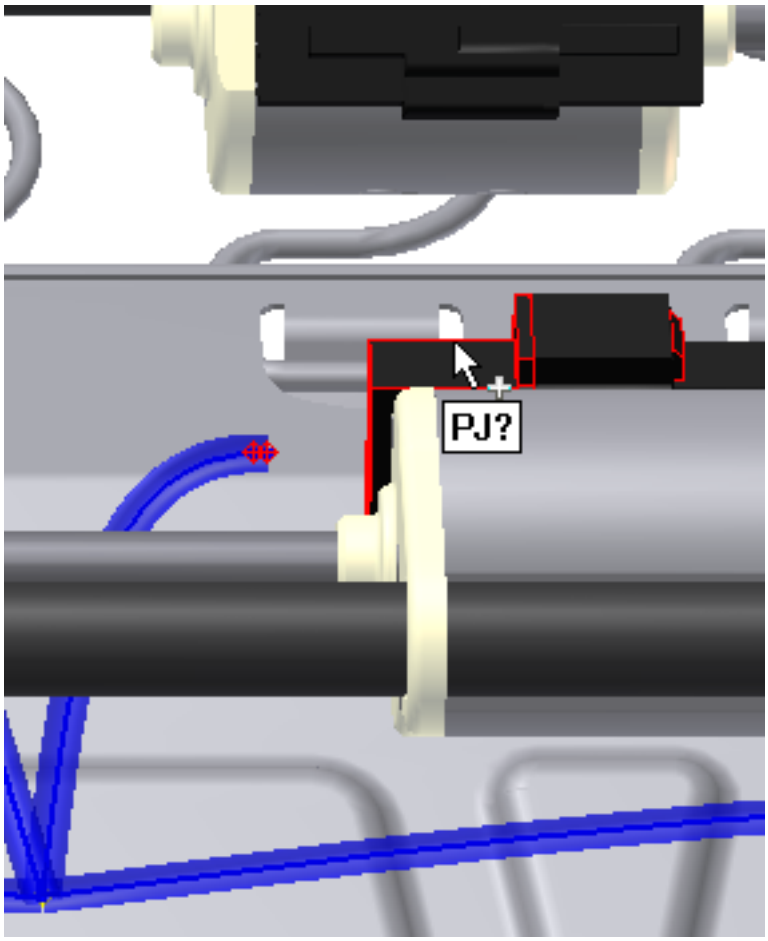
1 Right-click **PJ2** and select **Assign to an existing Electrical Part**.



About the AutoCAD Electrical drawing, *PJ2* is the RefDes specified for the connector that connects to the motor MOT2, the motor for horizontal adjustment.



2 Pause the cursor over the connector occurrence **900575:2** in the graphics window and note the tooltip.



You can select the occurrence in the browser. However, when you use the graphics window, Inventor displays a tooltip. The tooltip shows the RefDes for that component. The Question Mark (?) indicates that the RefDes is not yet assigned.

3 Select the connector.

4 In the Select Electrical Part dialog box, click **OK**.

5 The RefDes, as specified in AutoCAD Electrical, is assigned to the Inventor connector and the issue associated with *PJ2* is removed. Because the

dialog filter is set to show only items with issues, *PJ2* is not included in the list.

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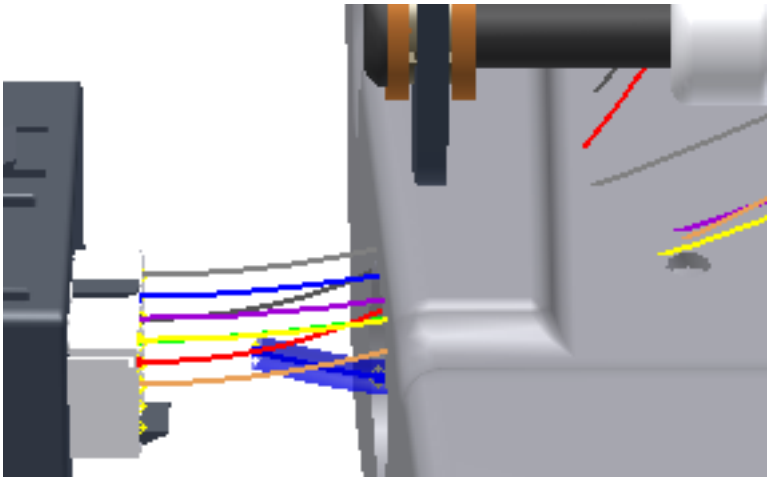
Finish the Import

Next, you finish the import operation.

1. In the Imported Harness Data dialog box, click **OK**.
2. Close the message dialog box. For this exercise, we accept the remaining issues without making further changes.

   The data from AutoCAD Electrical is imported. You see eight imported wires in the graphics window and in the browser.



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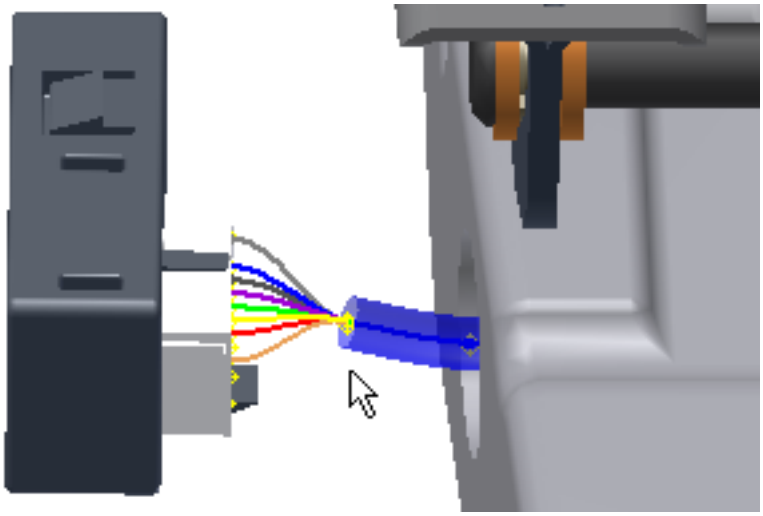
Route Wires into Harness Segments

1. Select **Cable and Harness ➤ Route ➤ Automatic Route**.
2. In the dialog box, select **All Unrouted Wires**. The **Selected** field indicates eight wires are selected.
3. Click **OK**. The wires are routed into the segments.



4. Select **Cable and Harness ➤ Exit ➤ Finish Cable and Harness**.

Save the assembly.

You completed Part 1, the 2D to 3D portion of the tutorial.

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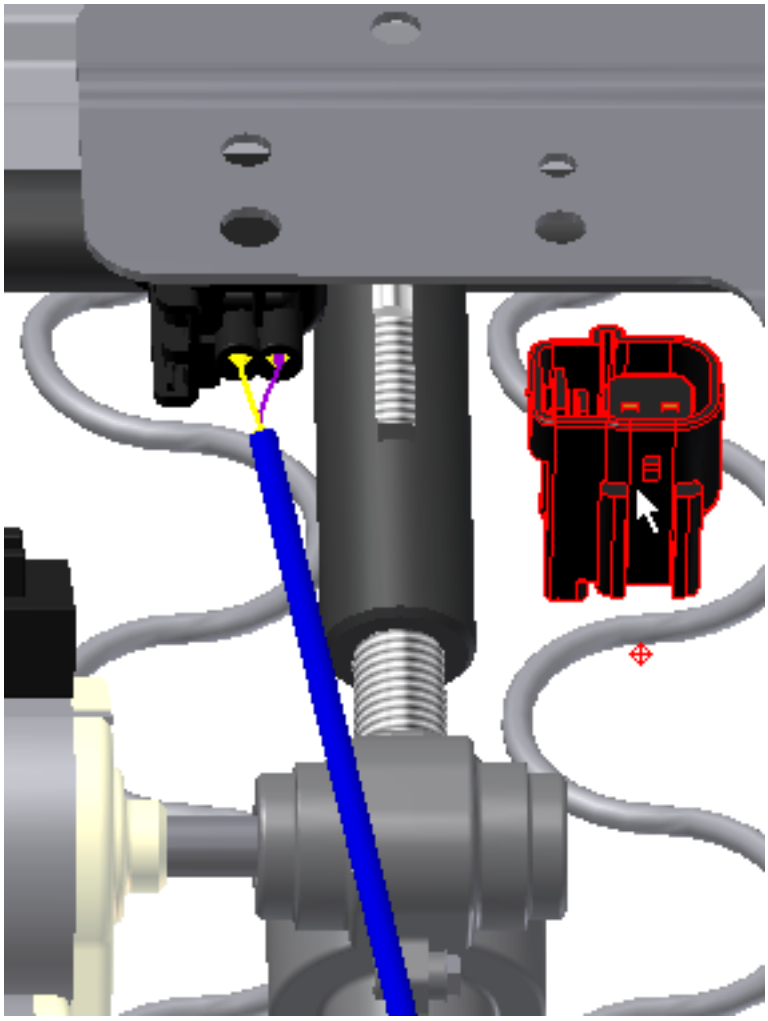
Part 2: 3D to 2D

In this half of the tutorial, you reverse the workflow and export electrical data from Inventor to AutoCAD Electrical.

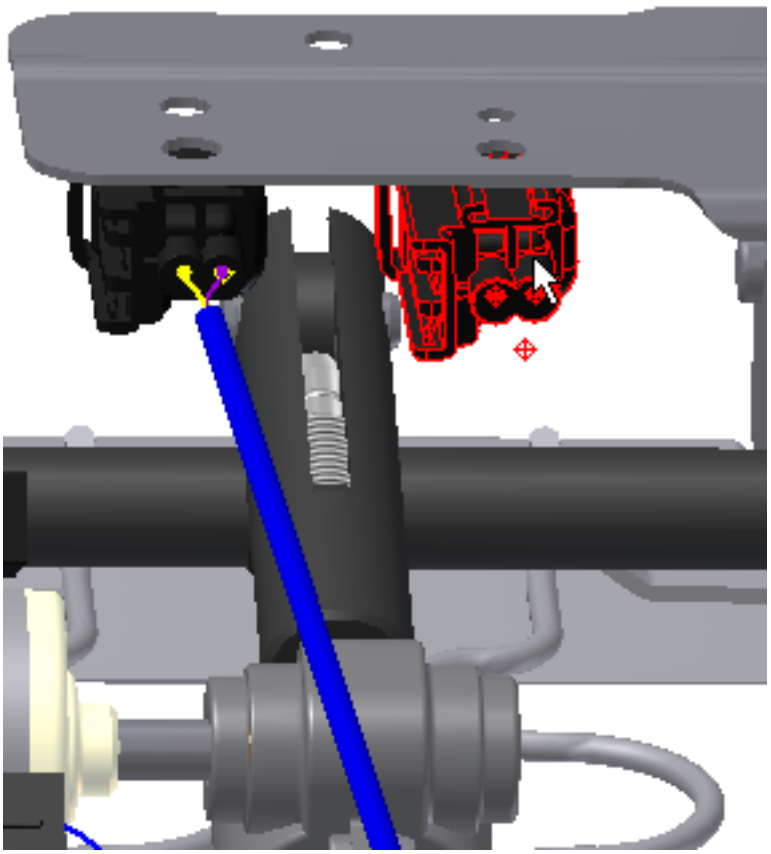
Add a Connector

1. Place an occurrence of the connector **900356.ipt**. Place the occurrence approximately, as shown.

   TIP Drag one of the existing occurrences from the browser into the graphics window to create another occurrence, instead of using the Place command.



2 On the **Assemble** tab, **Position** panel, use **Grip Snap** or **Move** and **Rotate.** Approximate the position of the connector as shown in the following image. The intent is that the pins on the connector face toward the front of the seat.



Normally you use assembly constraints to position and constrain the component, but it is not necessary for this exercise.

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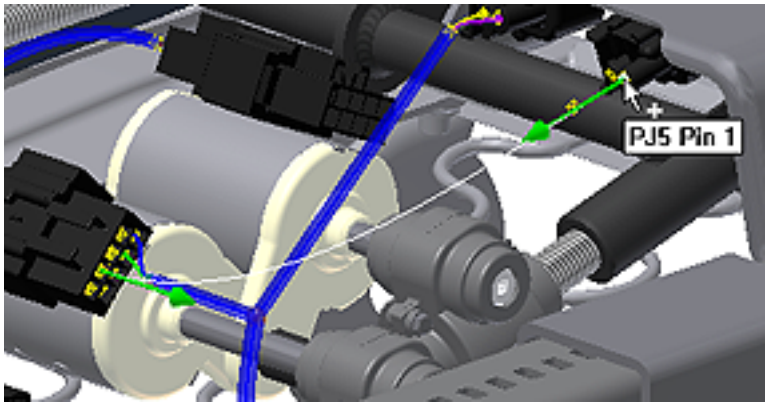
Create Wires

1 In the browser, double-click **Harness1** to edit the harness. Edit the harness assembly, not the part.

2 Select **Cable and Harness ➤ Create ➤ Create Wire**. 

3 Select the pins on the connectors, as shown in the following image. The identification for the first selected pin is **PJ4 Pin 7**.

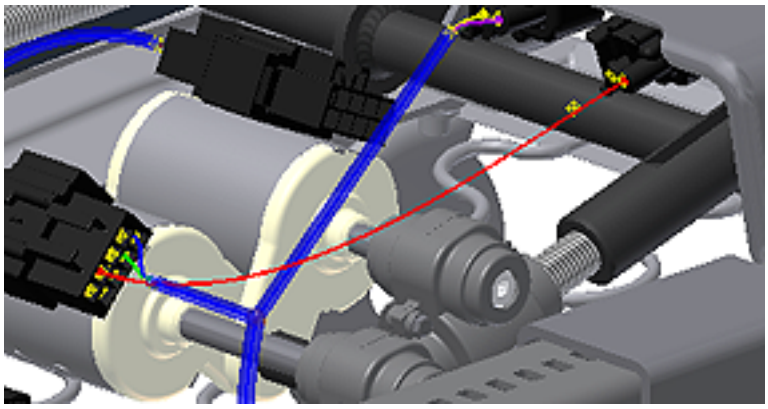


Before you apply the selections, use the Create Wire dialog box to specify **Wire ID, Category, Name** and other properties for the wire you are currently creating.

4 Specify the following settings:Wire ID:

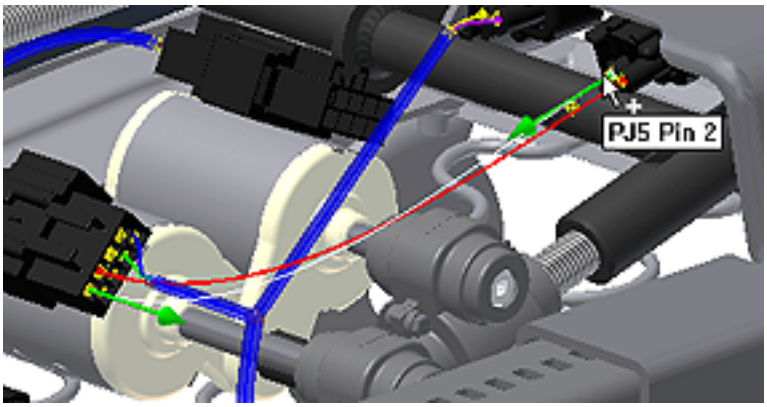
- 125
- Category: **Belden**
- Name: **9916-RED**

5 Click **Apply** to create the first wire.



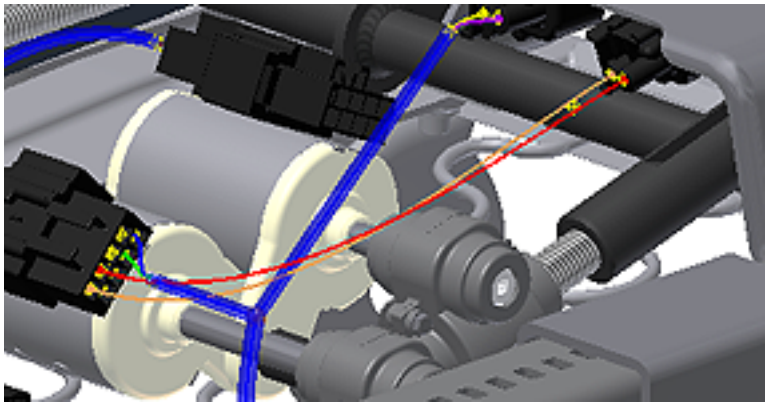
Create Wires (continued)

1 Select the pins on the connectors for the second wire. The identification for the first selected pin is **PJ4 Pin 8**.



2 Specify the following settings in the Create Wire dialog box:Wire ID:
- **126**
- Category: **Belden**
- Name: **9916-ORG**

3 Click **Apply**. Alternatively, right-click and select **Apply**.



4 Cancel the Create Wire dialog box.

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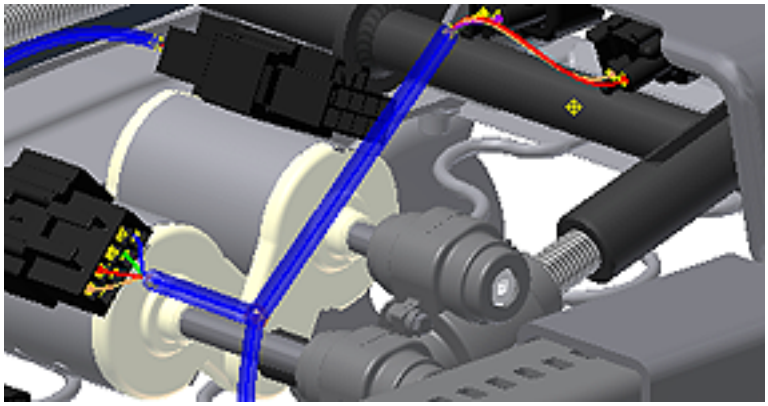
Route Wires

1 Use **Automatic Route** as you did previously to route the wires into the segments.



Save the file.

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Export to XML

Next, you export the electrical data contained in your Inventor digital prototype to an XML file. You use this XML file to import the data into AutoCAD Electrical. If you do not have AutoCAD Electrical you can skip this step.

1 Select **Cable and Harness ➤ Manage ➤ Export Harness Data**.

2 Name the file **driverseat_from_inv_exported.xml**. In Windows XP, save to: My Documents\Acade 2011\AeData\Tutorial\Aegs\Interoperability. In this folder, there is the supplied XML file as well. Dismiss the Cable and Harness message dialog box.

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Import Inventor Data

Switch to AutoCAD Electrical.

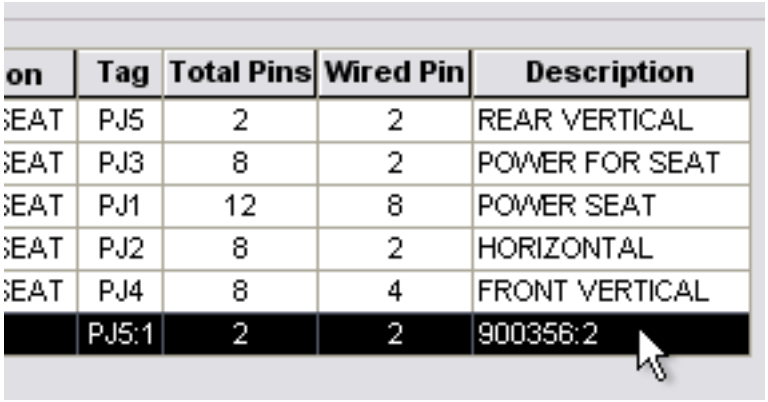
Apply the added connector and wires to the 2D model.

1 Select **Schematic ➤ Insert Components ➤ Insert Connector (From List)**.

2 In My Documents\Acade 2011\AeData\Tutorial\Aegs\Interoperability, select **driverseat_from_inv_exported.xml**. Click **Open**.

NOTE If you do not have Inventor, use the supplied file **driverseat_from_inv.xml**.

3 The connector you added in Inventor is listed in the Connector Selection dialog box. Select the row for the connector.

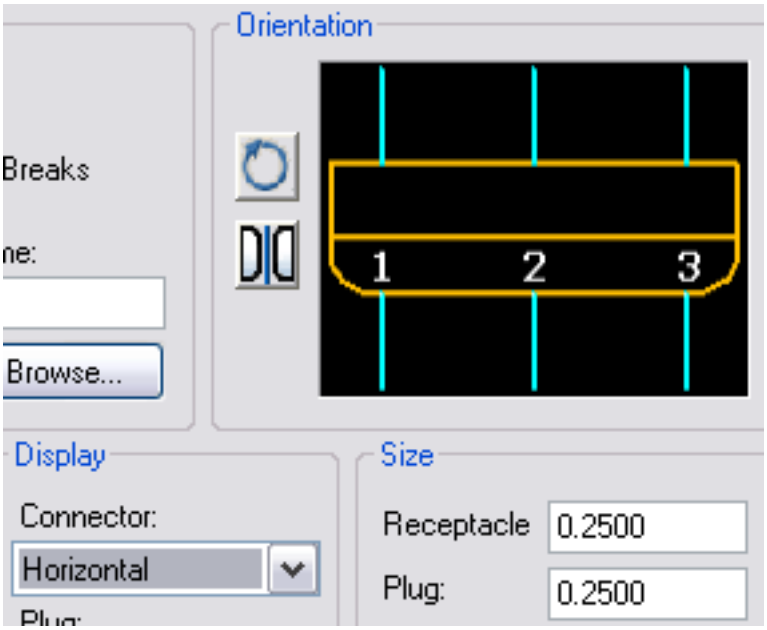


on	Tag	Total Pins	Wired Pin	Description
SEAT	PJ5	2	2	REAR VERTICAL
SEAT	PJ3	8	2	POWER FOR SEAT
SEAT	PJ1	12	8	POWER SEAT
SEAT	PJ2	8	2	HORIZONTAL
SEAT	PJ4	8	4	FRONT VERTICAL
	PJ5:1	2	2	900356:2

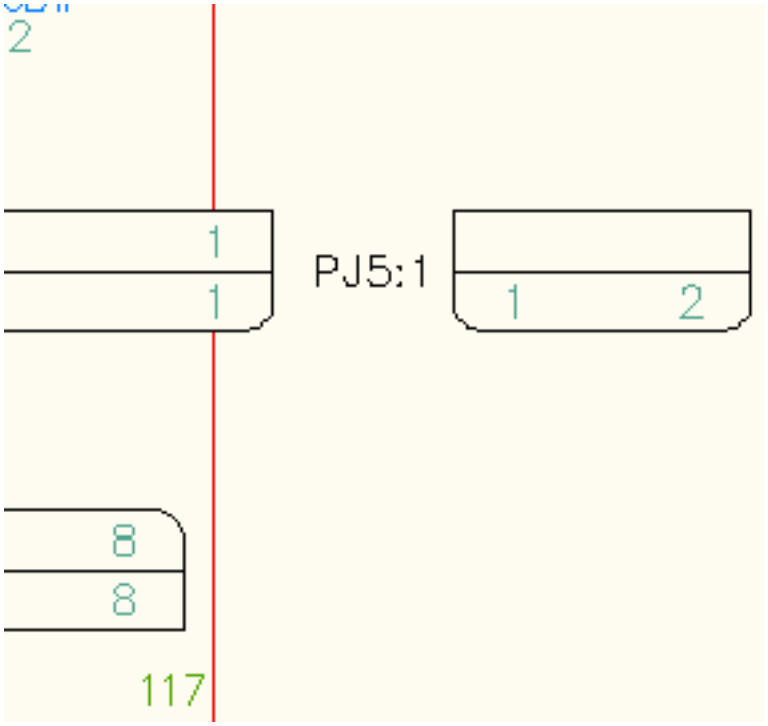
4 At the bottom of the dialog box, click **Details**.

5 On the Display > Connector menu, click **Horizontal**.

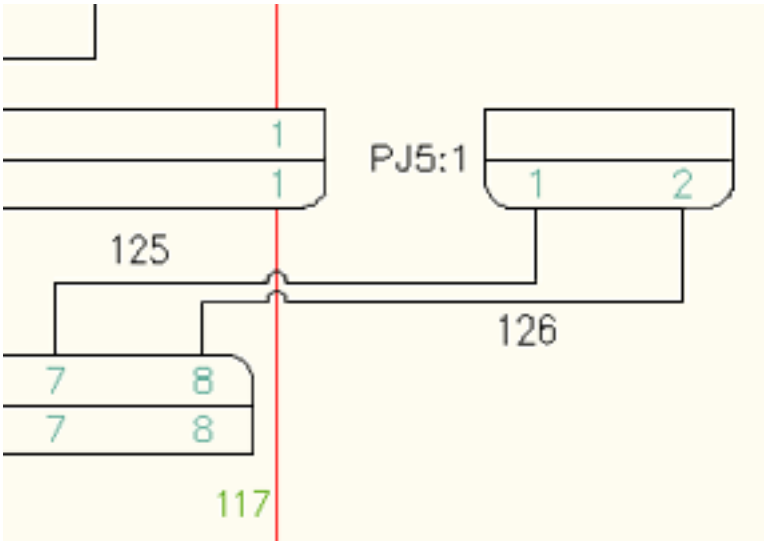
The display in the dialog box switches to horizontal.



6 Click **Insert**. Place the connector to the right of the power seat main switch.



7 In the Connector Selection dialog box click **Wire It**. AutoCAD Electrical connects the pins as you specified in Inventor.



You completed the tutorial.

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Summary

In this tutorial, you used interoperability between AutoCAD Electrical and Inventor Professional or Inventor Routed Systems to develop your digital prototypes. This tutorial showed you how you can:

- Define electrical data in AutoCAD Electrical and then apply that data to the related Inventor 3D prototype.
- Define electrical data in the Inventor 3D prototype and then apply that data to the related AutoCAD Electrical drawing.

This tutorial was meant to provide various details and to serve as an overview of a particular workflow. For further details, information, and options, see the related Help topics in AutoCAD Electrical and Inventor.

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