

Autodesk Inventor Routed Systems 2010

Getting Started

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Published by:
Autodesk, Inc.
111 McInnis Parkway
San Rafael, CA 94903, USA

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Tubes and Pipes

Part 1 of this manual presents the getting started information for Tube and Pipe in Autodesk Inventor® Routed Systems. This add-in to the Autodesk Inventor assembly environment provides the capability to create complete tube and pipe systems in mechanical assembly designs.

Get Started with Tube & Pipe

1

This chapter provides basic information to help you get started using Tube & Pipe in Autodesk Inventor® Routed Systems and the exercises in this manual.

It also provides information about the tube and pipe environment, how to get started with a tube and pipe assembly, and how to use tube and pipe data.

About Tube & Pipe

Tube & Pipe provides features for setting tube and pipe styles, adding runs and routes to mechanical assemblies or product designs, adding initial fittings to create route branches, and populating selected routes. When a tube and pipe assembly is complete, the tube and pipe information can be represented in drawings and presentations, and output to different data formats.

Tube & Pipe Features

With the tube and pipe tools you can:

- Create tube and pipe assembly files.
- Define, view, modify, copy, delete, and share tube and pipe styles that conform to industry standards.
- Create tube, pipe, or hose routes using automatic solutions the system calculates based on the style criteria.
- Create derived routes and edit the underlying base sketch.

- Use sketched tools to create parametric regions in rigid piping and bent tubing routes.
- Utilize the existing geometry and drawn construction lines to navigate the rigid piping and bent tubing routes.
- Defer updates on automatic routing for the tube and pipe runs assembly, pipe run, and individual routes.
- Modify both routes and runs by placing dimensions and geometric constraints.
- Access and use the Content Center to place conduit parts and fittings in tube and pipe assemblies. Fittings can be used to initiate route branches.
- Author custom tube and pipe iParts and standard parts for publishing to the Content Center Library.
- Populate selected routes with library components.
- Suppress memory-costly tube and pipe components in Level of Detail representations, especially in large tube and pipe assemblies.
- Place non-adaptive occurrences of master runs assemblies, runs, hose assemblies, and routes and make them adaptive for reuse.
- Swap in the needed master runs assembly member in the tube and pipe interchangeability set when a tube and pipe iAssembly factory is created.
- Recover the route centerlines in tube and pipe drawings.
- Use the bill of materials to document routes and runs.
- Move pipe runs rigidly and edit hose routes in positional representations.
- Save tube and pipe data to ISOGEN *.pcf* files or ASCII bend tables.
- Use the browser to edit tube and pipe runs and to change visibility for routes, runs, and components.

Tube & Pipe Environment

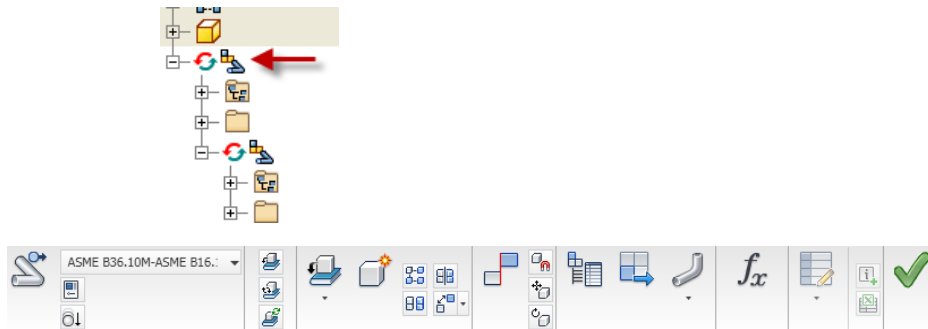
Tube & Pipe provides the familiar Autodesk Inventor® assembly environment in addition to commands for adding and editing routes and runs.

When you open an assembly in Autodesk Inventor Tube & Pipe, a command appears on the Assembly tab for adding piping runs to your design.



Once you add your first run, the commands specific to creating tube and pipe runs in an assembly are displayed.

The Tube and Pipe panel is active when the top-level (master) run is active as shown in the following image.

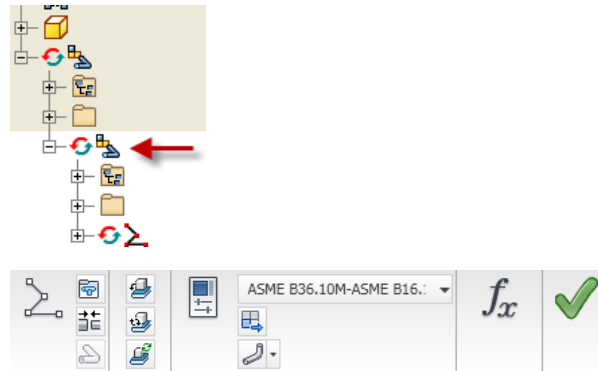


Tube and Pipe

Use the **Tube and Pipe** panel to add new individual runs to the master runs assembly, define tube and pipe styles, set Gravity, and output the ISOGEN data.

Use the tabs to switch between the Tube & Pipe and the Assembly panel bars.

The **Pipe Run** tab is active when an individual run is active as shown in the following image.



Pipe Run

Use the Pipe Run tab to add new routes, place and connect fittings, define tube and pipe styles, and output the ISOGEN data.

The Route panel is available when the tube, pipe, or hose route environment is active.



Route

Use the Route panel to create new routes, create a derived route, add or move sketch nodes and segments, define tube and pipe styles, and dimension or constrain the route.

There are several ways to enter the route environment, such as creating new routes in a pipe run, double-clicking an existing route, and right-clicking a route and selecting Edit.

There are additional tube and pipe commands contained here:

Browser toolbar	Enables switching between the Model browser, Content Center Favorites, and Representations.
Model browser	Contains the content of the tube and pipe assembly in a hierarchy, along with the main Tube & Pipe Runs subassembly. The main run acts as a container for all routes and runs and the components created or placed in the routed system.
Part Features panel	Contains the Tube & Pipe Authoring command to author tube and pipe iParts and normal parts for publishing to the Content Center.

Tube & Pipe Browser

All tube and pipe components added to an assembly are contained in the main runs subassembly. The components include individual runs and their associated routes, fittings, segments, and route points.

Each run contains an Origin folder, one or more routes, and any segments or fittings that are populated or manually inserted.

Routes contain the designated route points. To become familiar with the various components in the browser, open a sample tube and pipe assembly.

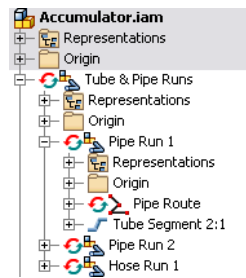
Use the Tube & Pipe Browser

- 1 Open the tube and pipe assembly, *Accumulator*. By default, it is located at: **Windows®XP** *C:\Program Files\Autodesk\Inventor<version>\Tutorial Files\Tube & Pipe\Accumulator*

Windows Vista®

C:\Users\Public\Documents\Autodesk\Inventor<version>\Tutorial Files\Tube & Pipe\Accumulator

- 2 Activate and expand the master runs assembly Tube & Pipe Runs. Pipe Run 1 contains components for a bent tubing style run.



- 3 Expand Hose Run 1 to view components for a flexible hose style run.
- 4 Continue to expand folders in the hierarchy until you are familiar with the contents, and then close the assembly.
- 5 To view components for a rigid pipe run, open the sample, *Buffer Prep Skid*. By default, it is located at: **Windows XP** *C:\Program Files\Autodesk\Inventor<version>\Samples\Models\Tube & Pipe\Buffer Prep Skid*

Windows Vista

C:\Users\Public\Documents\Autodesk\Inventor<version>\Samples\Models\Tube & Pipe\Buffer Prep Skid

- 6 Expand the folders in the hierarchy until you are familiar with the contents, and then close the assembly.

Work in Autodesk Inventor Installations

If Autodesk Inventor Tube & Pipe is not installed on the system viewing the tube and pipe data, the master runs assembly and all that it contains is read-only. This means that the outline of the tube and pipe component is

visible through Autodesk Inventor, but the component cannot be edited, and new tube and pipe components cannot be added.

When the Tube & Pipe add-in is not installed, the tasks you can perform with the tube and pipe data include:

- Open a Tube & Pipe Runs subassembly.
- Determine interferences with tube and pipe components in populated routes by selecting the entire tube and pipe runs subassembly in the browser.
- View the outline of tube and pipe run data within the context of an open assembly file.
- Turn visibility off to completely hide the Tube & Pipe Runs component in the graphics window.
- Create detailed drawings of populated routes within any file.

Prerequisites

It is assumed that you have a working knowledge of the Autodesk Inventor interface and tools. If you do not, use the integrated Help for access to online documentation and tutorials.

At a minimum, you should understand how to:

- Use the assembly, part modeling, sketch, and drawing environments, and browsers.
- Edit a component in place.
- Create, constrain, and manipulate work points and work features.
- Set color styles.
- Use Content Center.

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It is also recommended that you have a working knowledge of Windows XP, or Windows Vista, and a working knowledge of concepts for routing tube and pipe through mechanical assembly designs.

Back Up Tutorial Data Files

For each exercise in this section, you use files that contain the example geometry or parts for that task. These files are included in the *Tutorial Files* directory for the application. For Tube & Pipe, the files are located in the installation path of Autodesk Inventor Tube & Pipe by default:

- **Windows XP** *C:\Program Files\Autodesk\Inventor <version>\Tutorial Files\Tube & Pipe*
- **Windows Vista** *C:\Users\Public\Documents\Autodesk\Inventor <version>\Tutorial Files\Tube & Pipe*

Before you begin the exercises, back up the files so the originals are always available. You can revert to these files if you make any mistakes during the exercises, or if you would like to repeat an exercise.

Back up the tutorial files

- 1 Go to the *Tube & Pipe* directory and create a new folder called *Exercise_Backup*.
- 2 From within the *Tutorial Files* directory, copy the exercise data into your new folder.

Now you can use the files in the *Tutorial Files* directory as you work through the exercises in this book. Keep any files you create for an exercise in the *Tutorial Files* directory to avoid the possibility of file resolution problems.

Workflow for Tube and Pipe Assemblies

The first step in creating a tube and pipe system is to open an assembly file. The assembly file can be empty or contain an assembly model. You can then create the first pipe run and begin adding bent tubing, rigid piping, and flexible hose routes.



Create a tube and pipe runs assembly

- 1** Set up the project environment such as project type, workspace, and permissions to the style library. Content Center configuration is also especially important.
- 2** Optionally, customize your own master runs assembly template.
- 3** Within a normal Inventor assembly, create a master runs assembly.
- 4** Use the Tube & Pipe Styles tool to set style options.
- 5** Select a tube, pipe, or hose style, and then create a new route using automatic solutions and parametric regions to guide your selections.
- 6** Adjust the route to design changes.
- 7** Populate the route or insert additional fittings into the tube and pipe assembly or directly onto a route. Fittings can be used to initiate new routes.
- 8** Add additional routes and runs.
- 9** Make final adjustments to each route and run. Change the active style or add, remove, reposition, and replace fittings, route points, and segments.
- 10** Populate selected route(s).
- 11** Optionally:
 - To reuse the master runs assembly, runs, routes, or flexible hose assemblies, place secondary occurrences and use the Make Adaptive tool to transition them to new primary occurrences.
 - Create Level of Detail representations to save memory if you are working with a large assembly.
 - Create drawing views based on specific representations and configurations, create and export bill of material tables for routes and runs, and annotate drawings using parts lists, piping styles, and so on.
 - Save the file in a different format such as ISOGEN or bend table to import to other drawing applications.

Set Up Projects For Exercises

For the exercises, browse to and select the project file in the *Tutorial Files* directory.

Set up the project for Tube & Pipe exercises

- 1 Click  > Manage > Projects.
- 2  Use the Configure Content Center Libraries tool to configure libraries. If required, ensure that you have logged in to your Autodesk Vault server and the needed libraries are ready on the server.
- 3 Select the *AirSystemAssy.ipj* as the active project.
- 4 Optionally, in the Edit Project pane, right-click Use Styles Library, and then select Yes or Read Only.
- 5 Optionally, expand the Folder Options, right-click Content Center Files, select Edit, and then enter CC.
The CC folder is under the root path of the project work space. You can specify any other location you need to store the Content Center library content. You can also use the default Content Center Files location.
- 6 When you have completed all settings for the project, click Save.
- 7 Ensure the project is active. If it is not, double-click the project.
- 8 Click Done.

Define the Master Runs Assembly

When an assembly file is first opened, the assembly environment is displayed and you can begin adding pipe runs. For the first pipe run added, the system creates the master runs assembly along with an individual run. The master runs assembly is a container for all pipe runs added to the assembly. The number of runs you include depends on your design and manufacturing documentation needs.

Each run can include one or more individual routes. All routes in a run can use the same or different styles. With the capability to assign unique styles, it is possible to have all three route types, each with different size diameters

in a single run. Routes can start and end on assembly model geometry or an initial fitting dropped on the route to create a branch or fork.

Create Master Runs Assemblies

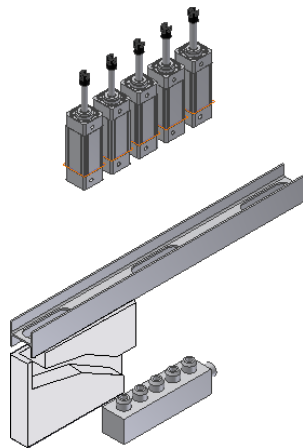
To create the master runs assembly, you click Environments tab ► Begin panel ► Tube and Pipe on the ribbon. On the Create Tube & Pipe Run dialog box that appears you provide a unique name and location for the master runs assembly and individual run respectively. By default, the files are saved to the workspace of the active project.


The Tube & Pipe Runs assembly is added to the browser along with other placed components and is arranged in the order it was added to the assembly.

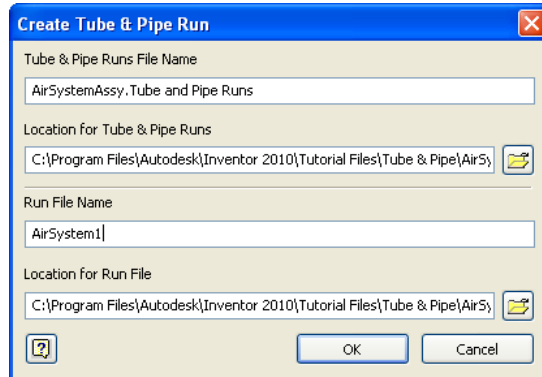
In the following exercise, you open an existing assembly and prepare to add tubing and piping in the context of that assembly. You also become familiar with the components included in the tube and pipe design environment including the Tube and Pipe tab, Pipe Run tab, Route tab, and Model browser.

Create a tube and pipe assembly

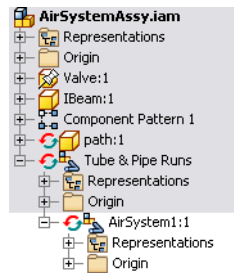
- 1 Click  ► Open.
- 2 Open the *AirSystemAssy.iam* assembly.



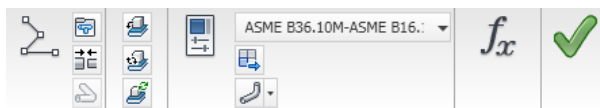
- 3  On the ribbon, click Environments tab ► Begin panel ► Tube and Pipe.
- 4 On the Create Tube & Pipe Run dialog box, enter *AirSystem1* as the name for the first run and accept the other defaults. Verify the default path in the project workspace.



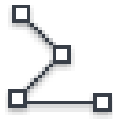
- 5 Click OK.
The individual run is automatically added and activated so you can begin defining a route and adding components.



- 6 Examine the available tools on the Pipe Run tab, as shown in the following image.



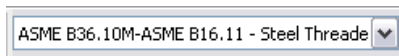
- 7 Examine the Pipe Run tab to see the New Route tool, the Display All Objects/Display Route Only tool, and the Active Style list.



New Route

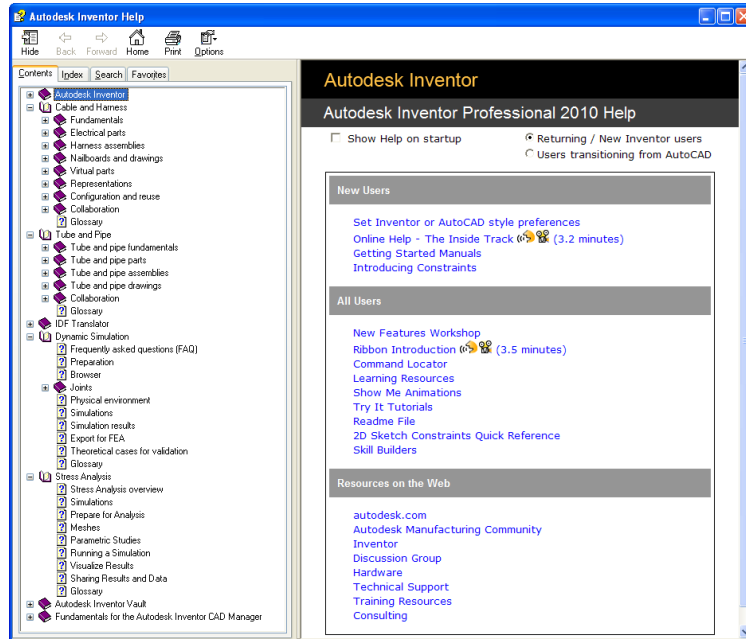


Display All Objects/Display Route Only



Active Style list

- 8 Click Help ► Help Topics to access Help, and then click the Contents tab to display the table of contents if it is not already displayed.
- 9 Click Tube and Pipe to display and examine the related Help topics.



10 Close the Help window.

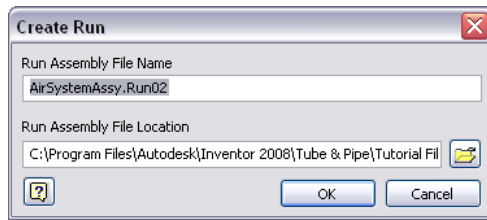
11 Activate the top-level assembly and save the file.

NOTE It is recommended that you save the top assembly regularly. Tube and Pipe components are not stored in your project workspace until the top assembly is saved.

Add Individual Runs

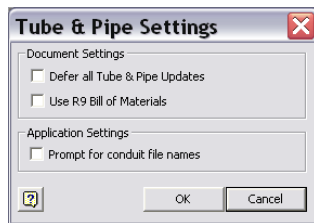
Each time you use the Create Pipe Run tool after the Tube & Pipe Runs assembly is created, an individual run is added to this container run. Individual runs are arranged in the order they are added to the tube and pipe assembly. You can name and locate each run file as it is added.

The following image shows the default settings when you create the second run:



Specify Global Settings

With the top assembly active, right-click Tube & Pipe Runs in the Model browser and use the Tube & Pipe Settings dialog box to specify the global document and application settings for the master runs assembly.



Defer All Tube & Pipe Updates

Defer all tube and pipe updates when you are editing normal parts in a standard Autodesk Inventor assembly or editing positional representations.

After the Defer All Tube & Pipe Updates check box is selected, most commands specific to Tube & Pipe are disabled. The tube and pipe runs assembly is not visible. Neither the tube and pipe runs assembly nor the associated parts in drawings automatically update. You cannot create new pipe routes and runs in the tube and pipe runs assembly.

When the Defer All Tube & Pipe Updates check box is cleared, you can defer updates for individual runs and individual routes. For detailed instructions, see [Control Individual Settings](#) on page 117 in Chapter 6.

Specify the setting for defer all tube and pipe updates

- 1 Activate the top assembly.
- 2 Right-click Tube & Pipe Runs, and select Tube & Pipe Settings.

- 3 In the Tube & Pipe Settings dialog box, ensure the Defer All Tube & Pipe Updates check box is cleared.
You can view automatic responses to edits on the tube and pipe assembly.
- 4 Click OK.

Use of Bill of Materials

The drawing manager uses the current bill of materials (BOM) to create tube and pipe drawings. When you migrate R9 or earlier tube and pipe drawings, you can specify whether to continue using the R9 BOM (default) or to use the current BOM.

The BOMs mainly differ in how they store the raw material description for library parts:

- In the current BOM, the raw material description for conduit parts is stored in the new stock number property. All BOM items with the same part number are automatically merged in parts lists. If the part number is blank, parts do not merge.
- In the R9 BOM, the raw material description for conduit parts was stored in the part number property. You had control over the merging of rows in the parts list. Even if two pipes had the same part number, you could choose not to merge them.

In the exercises that follow you use the current version of Autodesk Inventor Tube and Pipe to create tube and pipe assemblies, so the Use R9 Bill of Materials check box is cleared by default.

NOTE For more information about how to migrate legacy tube and pipe drawings to the current version, see the Help.

Prompt for Conduit File Names

When conduit parts are saved to your project workspace the first time, the default file names are used, with a suffix of a 13-digit number that is generated randomly based on your system time. The conduit file naming convention can be customized when you enable the Prompt for Conduit File Names option.

Specify the prompt for conduit file names

- 1** Activate the top assembly or master runs assembly.
- 2** In the Model browser, right-click the Tube & Pipe Runs and select Tube & Pipe Settings.
- 3** In the Tube & Pipe Settings dialog box, Application Settings pane, to accept the default conduit part file names, clear the Prompt for Conduit File Names check box. Otherwise, select the check box.
- 4** In the exercises that follow you do not need to customize the conduit file names, clear the check box.
- 5** Click OK.

Route Basics

2

A run is a collection of one or more routes with the same or unique styles that work together to make up a complete flow system. A route is the path that determines the shape of the flow system within the assembly and the intelligent placement of library components for the run along that path.

Some routes contained in a run start and end on the assembly model geometry. Other routes branch off a primary route to create a network of interconnecting rigid pipes, bent tubes, and flexible hoses required to represent a single flow system.

Once you have a route, you can populate it with the Content Center library content based on the tube and pipe style and the defined route path through the assembly.

About Rigid Routes

There are two types of rigid routes: rigid piping and bent tubing. The rigid route styles, Rigid Pipe with Fittings and Tubing with Bends, contain the rules for conduit parts and elbows. A pipe route can comprise pipe segments, couplings, 45-degree and 90-degree elbows, gaskets, gaps for groove welds, and custom bends. A tube route comprises tubing segments and tubing bends. Couplings connect straight segments and elbows or bends connect each directional change point. If a butt weld style is active and gaps are set to display, straight segments and directional change points have gaps between them for groove welds instead of fittings.

A rigid route can be a series of auto regions and parametric regions:

- An auto region is created by selecting circular openings and work points as the start and end route points and can have any number of segments that are automatically created by the system. As you select two circular openings, vertices, and work points, the system automatically generates the needed segments and route points. If more than one routing solution exists based

on the points selected, you can cycle through the solutions and select the one that best fits your needs.

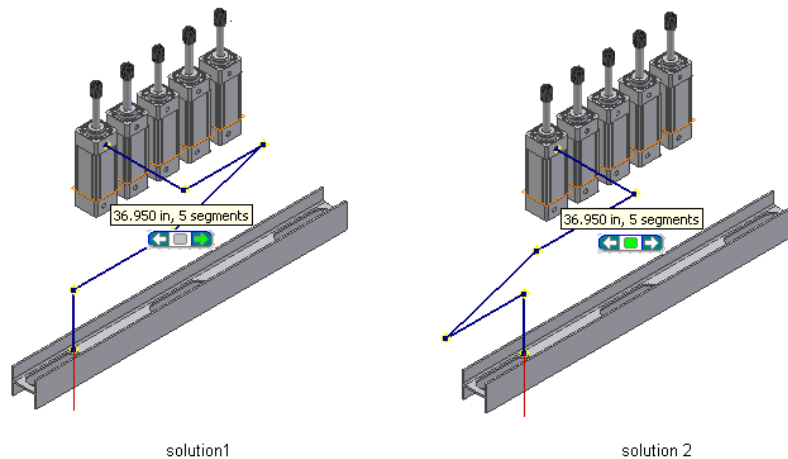
- A parametric region is created using the 3D Orthogonal Route tool along with the 3D sketch route tools, such as Point Snap, Rotation Snap, Parallel With Edge, Perpendicular To Face, Bend, and General Dimension.

Auto regions are created where geometric constraints are not important. Parametric regions are created to constrain the route to existing geometry or dimensions. An auto region can be converted to a series of continuous sketched segments at a later time.

Auto Route Cycling

As you select existing valid geometry, the system may calculate more than one auto region solution based on the points selected and display the Select Other tool. You can cycle through the solutions upon route creation or later edits, and select the one that best fits your needs.

Click the arrows to cycle through available solutions. Click the middle green button to make your selection. If there are multiple auto-route solutions in a single route, the Select Other tool cycles through the solutions in each section before advancing to the next section.



Solutions are evaluated and prioritized based on length and number of segments. The length and segment information is included in a tooltip as you consider the available solutions.

If you must switch to a new auto region solution in later edits, activate the route environment. The Alternate Auto Solution tool is available when right-clicking the auto region in the Model browser.

Parametric Regions

Along with the 3D Orthogonal Route tool, you can use geometric constraints, dimensions, custom bends, point snap, and rotation snap to manually define sketched route points.

If existing geometry such as a vertex, linear geometry, planar faces, and work features (including work points, work axes, and work planes) can help navigate through the route system, include them as reference geometry. You can then apply appropriate geometric constraints and dimension constraints to define the design.

In addition, you can draw construction lines from sketched route points and then use the General Dimension tool to position the coplanar segment accurately.

NOTE It is best to plan for route constraints before starting the design.

About Flexible Hose Routes

Flexible hose routes can contain up to three parts: a start fitting, a hose segment, and an end fitting. The start fitting and end fitting for a flexible hose must have two connection points. Flexible hose routes can also consist of only the hose, with both fittings suppressed, or a hose with one fitting suppressed.

To provide more control over hose shape, you can insert intermediate route points in the hose route as you create it. As you make your selections, a preview line appears between the selected points to help you visualize the route.

Flexible hose routes can be created between standard assembly geometry or initiated from fittings dropped onto existing routes to create a branch.

Route Points

Routes are created by selecting at least two route points. Route points can be manually defined. In rigid routes, the system may also automatically generate route points in auto regions.

A route typically starts from:

- Circular geometry such as a face, a hole, and cylindrical cuts
- Work points that reside in the assembly
- Vertices on any assembly component
- Existing free terminal route points in the active route
- Existing fittings

When you select circular geometry or work points, the route remains associative to these points. If the model geometry changes, the route automatically updates.

A variety of edit tools are available for route points on the Route tab and the right-click context menu, depending on the specific route creation mechanism and route type. For instance, when you are editing a rigid route, the 3D Move/Rotate tool, Constraints tool, and General Dimension tool are applicable to sketched route points. However, only the Move Node tool, Move Segment tool, and Edit Position tool are applicable to auto regions where appropriate.

Rigid Route Points

Each route point in rigid routes is typically associated to a fitting with an exception of free terminal route points. Rigid routes with a butt weld style are also an exception. In this case, gaps can be displayed between route components. Valid point selections are controlled by styles, connection data, and the action being performed.

To better control the direction of a route, you can manually define any number of intermediate route points using the 3D sketch route tools. Appropriate tools are available to edit route points in both auto regions and parametric regions.

System-generated route points always automatically update to changes made to the route during editing. Sketched route points in the parametric region may also dynamically update unless they are fully constrained.

Hose Route Points

In hose routes, intermediate route points are used to control the shape of the splines. They are not associated to any fittings. Depending on how the hose route style defines the start fitting and end fitting, intermediate route points can be inserted at an appropriate time.

To reposition the hose route points, you can place geometric constraints or adjust the offset distances from existing geometry using the Redefine tool. Editing the hose length does not impact the position of the adjacent hose route points.

3D Orthogonal Route Tool

The 3D Orthogonal Route tool appears as soon as you begin selecting route points in the graphics window. It contains several elements that guide selection of valid route points.

When the 3D Orthogonal Route tool first appears, only the line extender is displayed. With the line extender you can select points that are offset from a selected edge. Once you select a point along the line, other elements of the 3D Orthogonal Route tool appear at the selected point.

The elements displayed are dependent on what is selected, the connection geometry, and set styles. For example, the 3D Orthogonal Route tool includes different elements depending on whether you are creating a pipe route with fittings or a tube route with bends. Some elements are common to both styles.

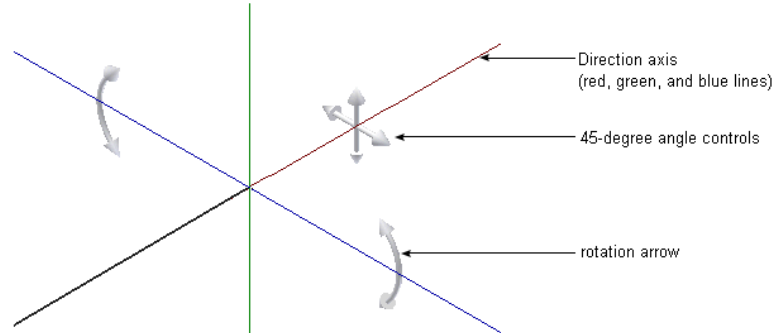
Valid points, those that will make a connection of the allowable length, are highlighted with a green dot as you move the cursor over the lines in the tool. Points that do not fall within the range set by the style criteria are displayed as a yellow x. The size of the tool can be increased if the line is not long enough.

Tool Elements for Pipe Routing

When a rigid piping style is active and all elements are displayed on the 3D Orthogonal Route tool, you can:

- Rotate freely around the local axis.
- Change direction in 90- or 45-degree increments.
- Create points from referenced geometry.

- Select points offset from selected edges.



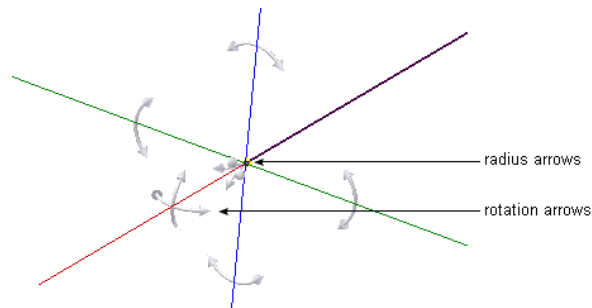
The different elements in the 3D Orthogonal Route tool for rigid pipes with fittings include:

- | | |
|--------------------------------|--|
| Direction axes | Shows valid direction for the next route point. Click the line to add a node in that direction. Together with the Point Snap tool, you can define a work point on the axis direction from referenced geometry. |
| Rotation arrows | Shows the rotation possibilities for the next route point and enables the free rotation. Together with the Rotation Snap tool, you can rotate the direction axis to an orientation from referenced geometry. |
| 45-degree angle control | Changes direction in 45-degree increments. This is available only when 45-degree route direction is set for the active style. |

Tool Elements for Tube Routing

When a Tubing with Bends style type is active the line extender, direction axes, and rotation arrows display along with the elements specific to creating bent tube runs. In addition to the ability to rotate freely around the local axis, create points from referenced geometry, and select points offset from selected edges, you can also:

- Change direction at any angle between 0 and 180 degrees.
- Make precise adjustments to the included angle using the rotation arrows.
- Change the bend radius using the radius arrows.



Change Tool Displays

Both tool color and size can be changed. Tool size is changed using the plus (+) or minus (-) keys on the keypad. Use plus (+) to increase the size, and minus (-) to reduce it. To adjust the colors in which the direction axes, line extender, or tooltip are displayed, set the colors as you would other color format styles.

Set colors in the display of the 3D Orthogonal Route tool

- 1 Open a tube and pipe assembly file containing at least one route. You can use the Tube & Pipe sample files, which are installed in *Autodesk\Inventor <version>\Samples\Models\Tube & Pipe* by default.
- 2 Activate the top-level assembly or Tube & Pipe Runs subassembly.
- 3 On the ribbon, click Manage tab ► Styles and Standards panel ► Styles Editor .
- 4 In the Styles and Standards Editor dialog box, expand Color in the left pane and select a *Route_UI_Tool_(toolname)* color style.
- 5 Set the appropriate color attributes.
- 6 Click Save to apply color changes immediately. Otherwise, colors become effective when you click Done to close the dialog box.

Define Angular Position and Rotation Snap

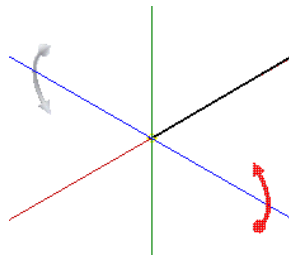
The rotation arrows and direction axes can be displayed when defining routes and placing fittings, and then again when editing and repositioning routes

and fittings. With the Rotation Arrows and Direction Axes displayed, you can view the rotation possibilities for the current selection.

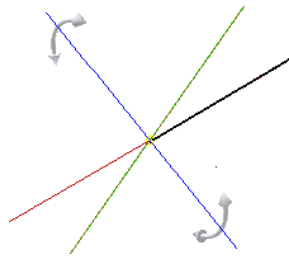
To rotate freely around the axis, click and drag the Rotation Arrow as needed. To snap the rotation in 90-degree increments, click on a line of the Direction Axis.

The tool snaps to all four quadrant border angles as you rotate. The tool also snaps to edge or face geometry. In this case, a dashed line and preview point show how the snap is applied.

Click and drag a rotation arrow.



Release the cursor in the new location.

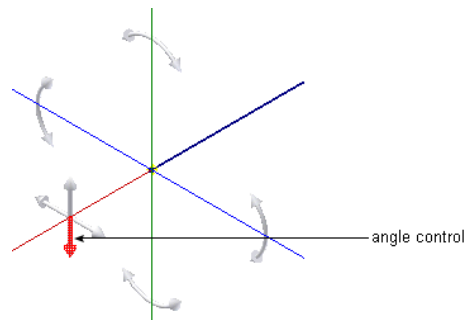


Define 45-degree Angles

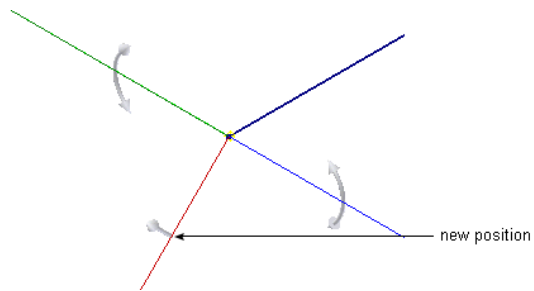
When the 45-degree route direction is set in the active style, the Angle Control can be displayed in the 3D Orthogonal Route tool. When it is displayed, you can rotate the route position in 45-degree increments.

To use the Angle Control, click the arrow pointing in the angular position you need. When you are satisfied with the position, select a point on the line to create a segment at the new angular position.

Click an arrow to direct the path to the angle you want.



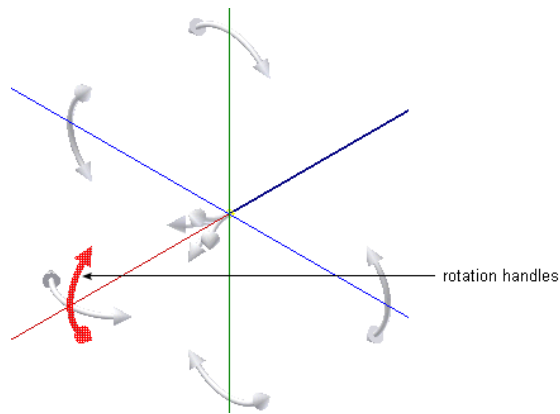
The route path changes to the new angular position. If you select the wrong direction, click the single arrow displayed on the selected axis to revert to the previous angular position.



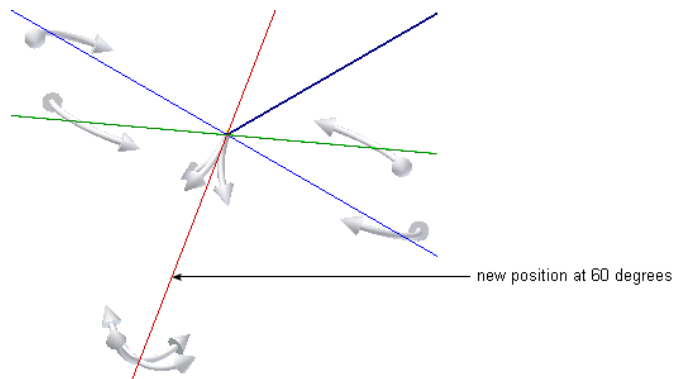
Define Bent Tubes Angles

When a tubing with bends style is active, the Rotation Handles are displayed on the 3D Orthogonal Route tool. When they are displayed you can create a bend at any angle.

To use the Rotation Handles click the arrow pointing in the angular position you need, and drag to the required position. The tool snaps in regular increments based on the 3D Angle Snap value. This value is set on the Tools tab ► Options panel ► Document Settings ► Modeling tab.



When you are satisfied with the new position, select a point on the line to create a segment at the new location. The route path changes to the new angular position. Using the Rotation Handles, you can also create a series of bends to achieve a compound bend.

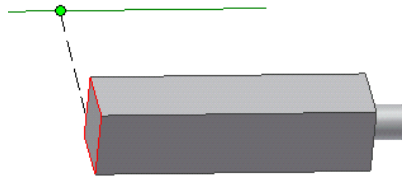


Use Point Snap to Define Points

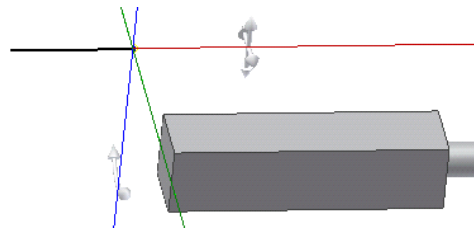
When the 3D Orthogonal Route tool is active and Point Snap is checked in the context menu, you can define points by snapping to other model geometry. Pause your cursor over faces, edges, or work points, a dashed line and preview point are displayed at the intersection of the line and the plane of the highlighted edge or face.

The dashed line represents the snap point in relation to the highlighted geometry. When the preview point is displayed at the needed location, click the selected geometry and the point is created.

A dashed line shows the point snapped from the face.



Click the face to create the route point at the intersection with the tool.



Enter Precise Values

Although all route points can be selected interactively, sometimes exact values are needed to create the required route. You can enter precise values for both angles and distances. The values entered are the distance or angle from the active position to the current node. If a point was snapped onto a line of the 3D Orthogonal Route tool, the value entered is the distance from the snap point to the desired node to add. The entered values must comply with set rules for segment length; otherwise, you are prompted to reenter the value.

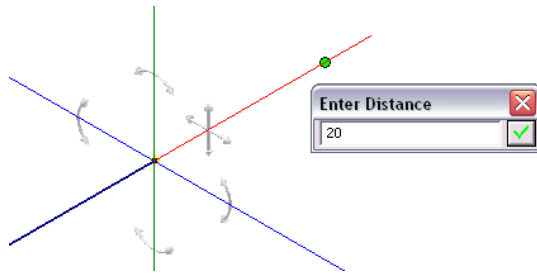
For tubing with bends styles, you can also enter a precise bend radius.

To enter an exact distance, start typing the value while your cursor pauses over the direction axis of the 3D Orthogonal Route tool. You can also right-click and select Enter Distance to enter a value.

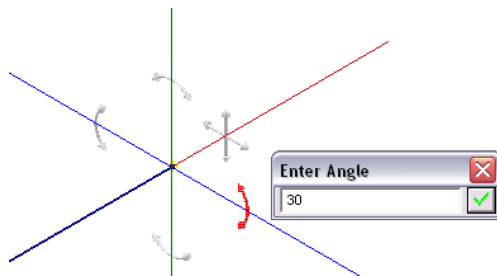
Entering angles is very similar. To enter an angle, start typing the needed value while your cursor pauses over a rotation arrow or bent tube rotation arrow. You can also right-click and select Enter Angle to enter an angle.

Enter exact distances while your cursor pauses over the rotation arrows, rotation arrows, or direction axes of the 3D Orthogonal Route tool.

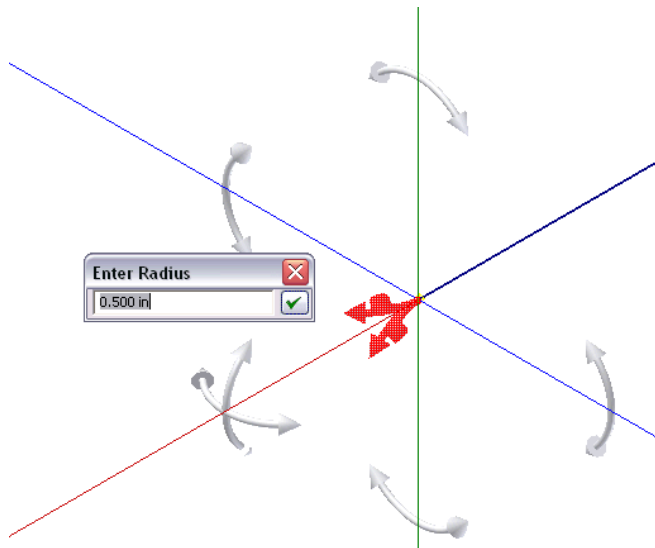
Enter the value for the distance.



Enter the value for the angle.



To view the current bend radius, pause your cursor over the radius arrow. Click the tool to enter a different value for the bend radius. You can also start typing the new value while your cursor pauses over the radius arrow.



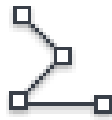
The new radius affects only the next bend. All subsequent bends use the default radius set on the Tube & Pipe Styles dialog box.

Route Tools

To start creating routes, you must activate an individual pipe run, and then enter the route environment. Along with the 3D Orthogonal Route tool, a variety of sketched route tools are available and can assist in the route design. You can create a route by connecting two or more points and directing the route through circular openings and around existing geometry in its path.

Basic Tools

To enter the route environment, use the Pipe Run tab ► Route panel ► New Route tool.



New Route tool

In Route mode, the Route tab displays and you can begin creating a path for your route. Use the Route tool to add new routes or continue existing ones.



Route tool

As you select points for a route, Autodesk Inventor® Routed Systems gives you visual feedback about what is happening in the graphics window and text messages on the status bar. The messages change based on what is selected and the action you are performing.

When deciding on the design of your route, you can:

- Decide whether you must manually define the route direction or allow the system to automatically calculate solutions.

- Identify the circular geometry (or work points) that are used as the start and endpoints.
- Analyze where directional change points are needed to route through or around existing geometry.
- Create in-line work features, such as a work point at the intersection of work planes, to guide the route path.
- Place constraints or dimensions to sketched segments.
- Convert auto regions to parametric regions so that you can make more edits.

You can define your routes as close to the appropriate results as possible, and then adjust them later, or you can develop them using precise distances and dimensions as you go. To speed creation and plan for dynamic editing and updating, allow the system to automatically create route points whenever geometric constraints are not important. Define route points manually where it is critical for a route to navigate through a particular direction and constrain to existing geometry.

If using in-line work features, consider simplifying the route display by selecting the Auto-hide in-line features option on the Tools tab ► Options panel ► Application Options ► Part tab. In-line work features are hidden as soon as they are consumed by a feature. The auto-hide option is enabled by default.

Bend Tools

Use bent tubing and flexible hosing styles to create curved routes conforming to the minimum bend radius. Typically, use the following tools to manually create bends in the rigid route:

- Custom Bend tool on the right-click context menu when the 3D Orthogonal Route tool is active. It is applicable when creating a pipe route.
- Bend tool on the Route tab. It is applicable when editing parametric regions in a finished route.



Bend tool

In pipe routes, when a bend is required where it has not been manually defined, the default bend radius of two times the pipe nominal diameter is used. For instance, applying the Parallel With Edge tool to a certain linear geometry may also require a bend if directional fittings in the Content Center such as 45-degree or 90-degree elbow are not appropriate.

NOTE When an existing bend is deleted by mistake, use the Bend tool to create a new bend where appropriate.

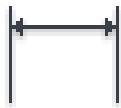
Dimension Tools

In rigid routes, there are three typical types of dimensions pertaining to the route sketch:

- Linear dimension, such as the length of route segments
- Radial dimension, such as bend radius for custom bends in piping routes and normal bends in tubing routes
- Angular dimension, such as angles between directional turns

Dimensions only apply to the route sketch in parametric regions. Auto regions always dynamically update to assembly changes so the number of segments typically varies from the new solution. They do not involve dimensions until the Convert to Sketch command is applied.

The Auto-Dimension tool on the right-click context menu enables you to switch whether to automatically dimension the subsequent route sketch. To place and edit dimensions manually, click Route tab ► Constrain panel ► Dimension or double-click an existing dimension in place.



General Dimension tool

Similar to Autodesk Inventor, dimensions on the route sketch can be categorized into two types: normal sketched dimension (driving dimension) and driven dimension. Normal sketched dimensions are used to drive the route geometry. For instance, sketched route segments are manually created using a specified dimension. Driven dimensions are enclosed in parentheses as displayed in the graphics window and allow route geometry to dynamically respond to associated changes.

NOTE The General Dimension tool cannot create a bend. After you use the Bend tool to create a bend between two coplanar segments, you can edit the bend radius using the General Dimension tool.

Parallel and Perpendicular Tools

During forward creation of sketched route segments, use the Parallel With Edge and Perpendicular To Face tools to reorient the axis of the 3D Orthogonal Route tool against existing geometry. When the next route point is defined on this axis, a parallel or perpendicular constraint is added to the resultant route segment. You can also pick up route points in the other two axes.

Applying the two tools may request an irregular angle (neither 45-degree nor 90-degree) at the preceding route point. Consequently, a custom bend is created. Radius arrow and rotation arrow are available on the 3D Orthogonal Route tool. Edit the bend radius and rotation angle as needed.

Other Tools

In addition to the 3D Orthogonal Route tool, Point Snap, Rotation Snap, Custom Bend, Bend, General Dimension, Parallel With Edge, and Perpendicular To Face tools that are discussed in the preceding sections, Tube & Pipe in Autodesk Inventor Routed Systems Suite provides the following tools to define the route sketch manually:

Constraints tool	Applies the 3D sketch constraints between route points, segments, and included reference geometry, such as Perpendicular, Parallel, Tangent, Coincident, Collinear, and Fix.
Include Geometry tool	Includes the reference geometry from the source geometry to constrain the route sketch, such as vertices, linear edges, planar faces, and work features (work points, work axes, and work planes).

Draw Construction Line tool Creates any number of construction lines from the sketched route point and places dimensions, typically the included angle between the construction line and the adjacent segment.

For detailed instructions on how to use these tools, see the Help.

Set Styles

3

Tube and pipe styles describe the characteristics for tube, pipe, and hose routes. These styles are key to controlling the design of the routed system as it evolves from prototype to manufacturing.

This chapter provides basic information about the available options, how to set them, how to modify and change them, and how to add them to a template.

About Tube and Pipe Styles

Tube and pipe styles affect most aspects of route design from route creation and editing to populating the route. They are used to ensure consistent application of tube and pipe components. For example, conduit parts and fittings in a pipe run often have certain requirements for size, route direction, and materials. With tube and pipe styles, you can set these requirements once, and then apply them to the design.

When defining a style, you are specifying the conduit part and fittings from the Content Center libraries that make up the pipe run and establishing rules to be followed during routing.

There is a list of system-supplied tube and pipe styles in Autodesk Inventor® Routed Systems. You can use one of these styles, create your own style based on one of them, or create your own style based on published conduit parts and fittings.

NOTE It is possible to define a style with which you are unable to create a route, such as if you select fittings that do not have compatible end treatments. While the Tube & Pipe Styles tool allows styles to be defined in this way, routes cannot be created using such a style.

To correctly define new materials for tube and pipe styles, you must enable the Use Styles Library setting for your project. For detailed instructions, see [Set Up Projects For Exercises](#) on page 11 in Chapter 1.

WARNING Do not confuse tube and pipe styles with styles and style libraries in Autodesk Inventor®. They are two separate, unrelated entities.

Access Tube and Pipe Styles

Before you begin working with routes and runs, examine the style settings, and then select the style you need.

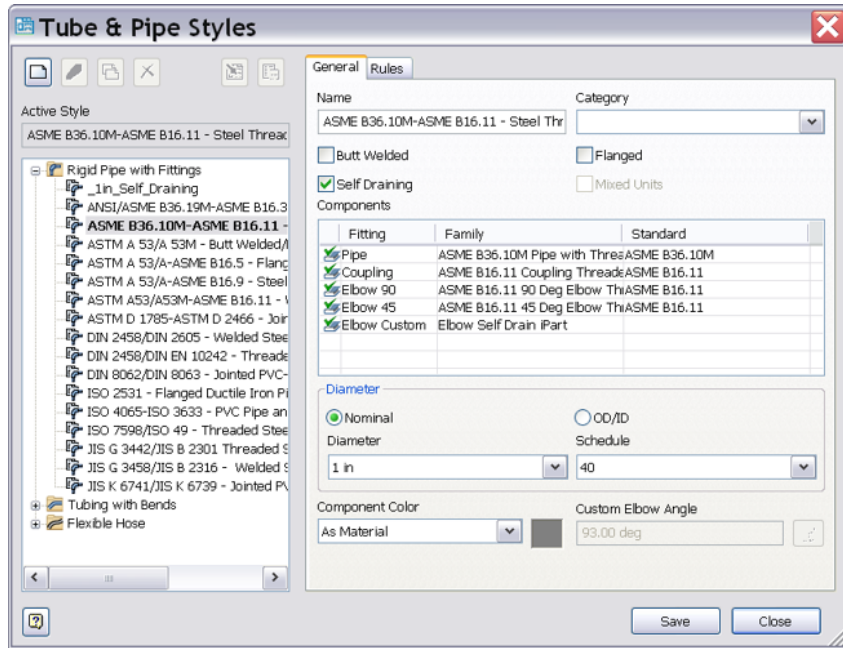
To work with styles, activate the tube and pipe environment and start the Tube & Pipe Styles tool to display the Tube & Pipe Styles dialog box. The Tube & Pipe Styles tool can be accessed on the ribbon or right-click menu when you activate:

- The master runs assembly
- A run
- A route




Tube & Pipe Styles tool

Existing styles are also available from the Active Style list on the ribbon.



The features in the Tube & Pipe Styles dialog box include:

- Toolbar** The tool set for the library that you use to create, edit, copy, and delete one or more defined styles. Import and export functions are also provided to bring style definitions in and out of the local or master runs templates. Pause your cursor over the toolbar to view the  tooltips.
- Active Style** Displays the style used when automatically creating a tube or pipe route and when populating a route with library content. The style displayed is based on the active tube and pipe environment.
- Styles browser** Displays the list of available style types, categories, and individual styles. By default, the browser is expanded to the location of the active style. Expand the list to view and select other style types, categories, and style definitions.
- General tab** Sets or displays the name and category, style type, default components, and other general characteristics of

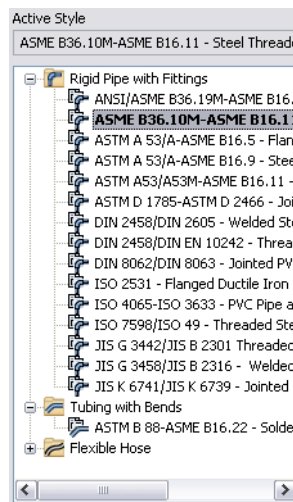
Rules tab

how routes are created for new or existing style definitions.

Sets the parameters that specify the size range for creating route segments between identified route points. For butt welded styles it also sets gap size and display. For flanged and butt welded styles, it indicates the type of coupling to use.

View and Select Styles

When you first open the Tube & Pipe Styles dialog box, the browser list automatically expands to the location of the active style. The active style displayed is dependent on the current tube and pipe environment. The active style is displayed above the styles browser in a read-only input box. It is also highlighted in bold in the browser list.



There are three types of styles in the list:

Rigid Pipe with Fittings

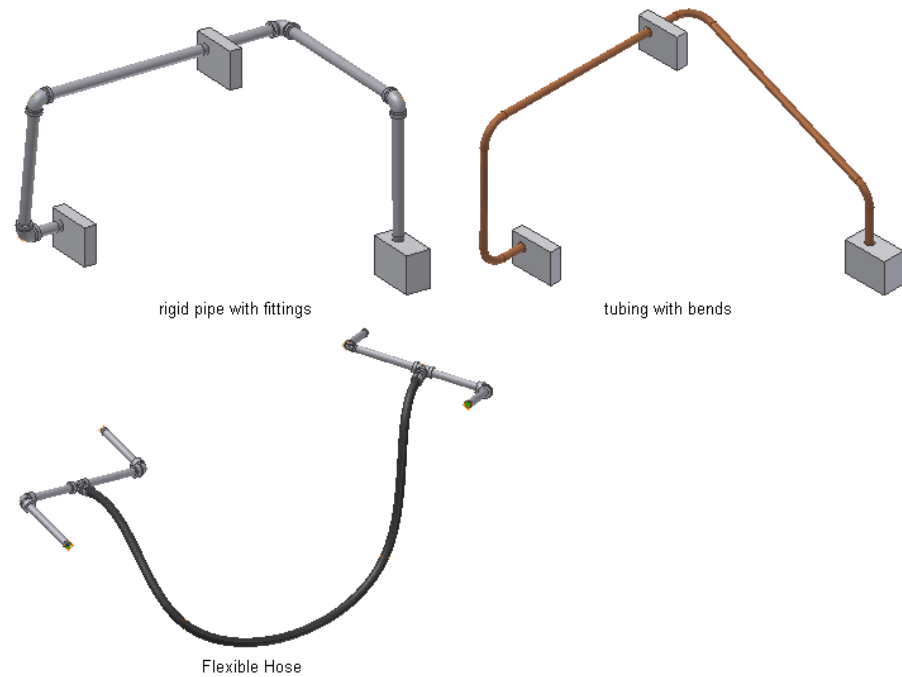
Creates a series of straight pipe segments connected with specified fittings. The required and optional fittings vary depending on the type of rigid pipe style being created.

Tubing with Bends

Creates a single route with bends instead of fittings at the directional change points.

Flexible Hose Creates a route consisting of a single hose segment that can connect up to two fittings

An example of each is shown in the following image.



The style browser also provides access to the list of all defined styles. You can use this list to select one or more styles, and then click a tool on the toolbar to perform various operations. You can also right-click a node to display a context menu with additional options.

Set Style Options





The Tube & Pipe Styles dialog box has two tabs. The nodes you select in the browser list and certain selections you make on the General tab act as filters to drive not only the available nominal diameter values and the rules, but also the components from which you can choose.

The parameters on each tab are described in this section.

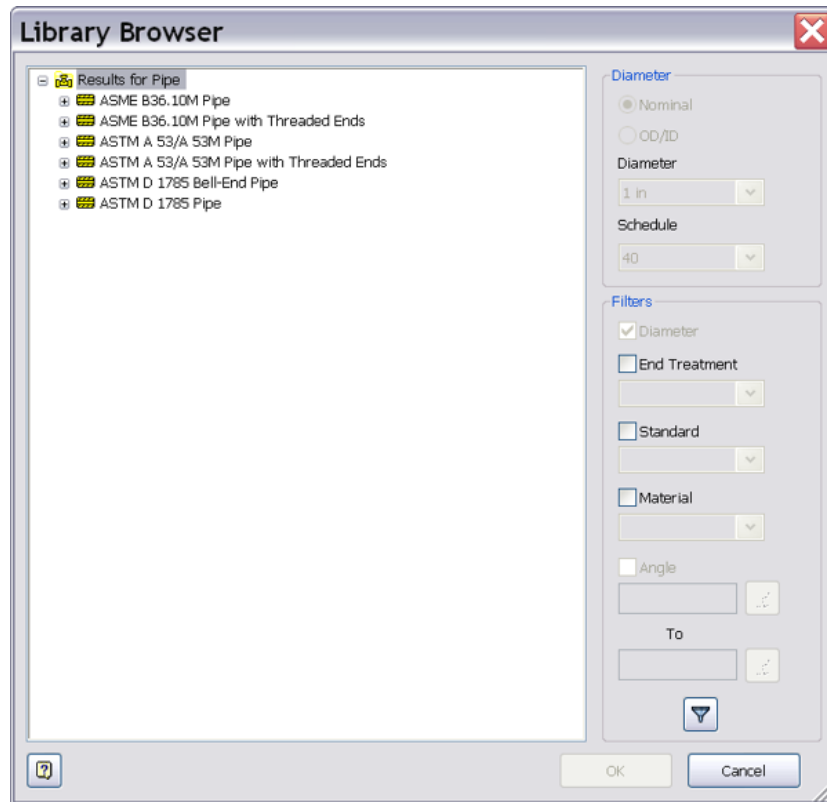
General Tab

On the General tab you can set a new, unique name for a style or modify and rename an existing one. You also specify the general characteristics of how routes are created and select components to include in the style definition.

The components table lists a basic set of components for the style being viewed or created. Symbols indicate whether the component is optional or required and if there are any issues with the component. Pause the cursor over the symbol for a description.

Fitting	Family	Standard
 Pipe		
 Coupling		
 Elbow 90		
 Elbow 45		

Right-click a row in the table to clear existing components, suppress or unsuppress fittings, or browse for components. When browsing for components, the Library Browser dialog box is displayed. The parts available for selection on the Library Browser dialog box are filtered from the Content Center based on the node selected in the styles browser and the style criteria set on the General tab.



You can also set additional filtering, such as material and industry standard to further refine the list. The standards and materials of published parts are listed. Use the lists to make your selections. Once the standard is selected, the available materials for the standard are listed. When an asterisk is displayed, the system returns all content for that setting. For example, the system allows all (* /any) materials to appear in the results.

Rules Tab

The Rules tab sets parameters that specify the size range for creating route segments between identified route points. It includes the minimum and maximum values, as well as the increment round-off value. Other settings are specific to the type of style being created:

- For bent tube routes, you can set the default bend radius for the bends.

- For flexible hose routes, you can set a hose length round-up value and minimum bend radius.
- For butt welded styles, you can set the gap size for the groove welds and whether to display the gaps in the graphics window and drawings.
- For a combination butt welded and flanged style, indicate the style to use at coupling points. The fitting connections are determined by the end treatment that is set for the fitting. All other end treatment types use a gap to join segments and fittings.

WARNING It is recommended that the minimum segment length is at least 1.5 times Nominal Diameter. Otherwise, it is more likely to cause a minimum segment length violation if conduit segments are too small compared to Nominal Diameter.

Work with Styles

Although it is best to set styles before creating routes or placing fittings, styles can be created at any time and style changes can be applied to new and existing routes throughout the design process. With styles you can:

- Set style defaults for all new routes you create.
- Change the active style for the tube and pipe assembly.
- Change the style for the active route.
- Modify settings for all routes that use the same style.

NOTE You cannot apply a rigid type style to an existing flexible hose route and vice versa. To change between a rigid style and flexible hose style, you must delete the route and create a new one using the flexible hose style.

Before creating a new style, author necessary conduit parts and fittings and publish them to the Content Center. They are not created automatically. Once custom parts are authored and published, define the new style to match the properties of published parts. For detailed instructions about authoring and publishing, see [Author and Publish](#) on page 155.

You can modify any of the available style definitions including those that are provided with the system. The change is saved with the style. The style change is also applied to any route currently using that style. In the following exercises you create new styles based on existing ones.

NOTE Before you make changes to a system style, make a copy and give it a new name. You can then select the system style from the browser and make the modifications.

To create a new style from a blank one, select the style type to create, and then click New. This leaves the basic requirements for the style type as a guide, but clears all values.

Create Rigid Pipe with Fittings Styles

When creating rigid pipe styles, the required components depend on the type of rigid style being created.

- Typically three compatible parts are required: a pipe, a coupling, and an elbow. If you require both 45-degree and 90-degree elbows, four parts are required.
- Self Draining styles require five components: a pipe, a coupling, a 45-degree elbow, a 90-degree elbow, and a previously published custom elbow or tee that matches the slope angle.
- Welded tube and pipe styles typically need two part types: a pipe and a 90-degree elbow. Butt welded styles require you to set a gap size for the groove welds and determine whether to display the gaps in the graphics window and drawings.
- Flanged styles require: a pipe, an elbow, a flange instead of a coupling, and an optional gasket.



In this exercise, you create two new rigid piping styles using existing styles as the base.

NOTE When you switch between styles or create new styles during edits, you are prompted whether or not to save edits. Click Yes to save edits to the current style before proceeding or click No to proceed without saving changes to the current style.

Create rigid piping styles

- 1 With the *AirSystemAssy.iam* assembly open, activate the master runs assembly Tube & Pipe Runs or the AirSystem1:1 run.



- 2 On the ribbon, click the Tube & Pipe Styles tool.
- 3 In the style browser, select ASTM A53/A53M - ASME B16.11 Welded Steel Pipe as the basis for the new style.
- 4  Click Copy.
- 5 From the styles browser, select the copy you just created.
- 6  Click Edit.
- 7 On the General tab, enter *Welded Steel Pipe - ASTM A53/A53M - ASME B16.11 (1/4 ND)* in the Name input box.

NOTE As you create new style definitions, you can also create categories in which to organize them. Categories are optional.

- 8 Under Components, notice that there are 4 components included in this style that have all been successfully selected.
- 9 Under Diameter, click the arrow to select a nominal diameter of 1/4 in. from the list.
- 10 If desired, click the Rules tab to view the settings for minimum and maximum values and the increment round-off.
- 11 Click Save.
The new style is added to the browser list, but is not set as the active style for forward route creation.
- 12 To set the style as active, right-click the style in the browser list and select Active.
- 13 To define the second style, make a copy of ISO 7598/ISO 49 Threaded Steel Pipe with Iron Fittings.
- 14 Select and then edit the copy you just created.
- 15 Name the new style: *Threaded Steel Pipe with Iron Fittings (1/2 ND, 90 Only)*.
- 16 Under Components, right-click the Elbow 45 row and select Clear.
- 17 Under Diameter, select a nominal diameter of 1/2 in.

- 18 Click Save.

Tubing with Bends Style

- 1 In the style browser, expand the Tubing with Bends style type.
- 2 Select ASTM B 88-ASME B16.22 Soldered Copper Tubing as the base for the new style.
- 3 Copy and edit the style as you did for the previous styles.
- 4 Name the style: *Soldered Copper Tube (1/2 ND,.5 Bend)*.
- 5 Click the Rules tab and enter a bend radius of *0.500 in.*
- 6 Click Save.

Flexible Hose Styles

Flexible hose styles have some additional options that the other styles do not. You can set the Use subassembly option to determine how the flexible hose route is structured. It can be created in a flat structure, or with route components grouped in a subassembly under the pipe run (default). Once you create a flexible hose route, the route structure cannot be changed.

As with rigid route types, one of the first things you must do before creating a route is to select a flexible hose style using the Tube & Pipe Styles tool. You can create your own flexible hose style using custom fittings published to the Content Center or select a predefined style.

Flexible Hose Style Parameters

A flexible hose style includes several parameters that are common to all three route types. The parameters specific to a flexible hose style include:

- Hose Round Up Value** (On the Rules tab) Rounds the hose length up to the first larger value, based on the specified increment. For example, if the Hose Round Up Value is set to 0.5 in., the hose length of 1.4 in. is rounded up to 1.5 in.
- Fitting Types** (On the General tab under Components) Includes a hose part, a start fitting, and an end fitting. The start and end fittings can be suppressed using the context

menu from the appropriate rows in the components table. Suppressed fittings are not included in the flexible hose route. If you suppress the start fitting, the end fitting is automatically suppressed.

Use subassembly Determines the structure of the hose as either a flat structure or a subassembly.

For more information about parameters, see [Set Style Options](#) on page 41 in this chapter.

NOTE You cannot apply a rigid type style to an existing flexible hose route and vice versa. To change between a rigid style and flexible hose style, you must delete the route and create a new one using the flexible hose style.

Structure Flexible Hose Routes

When defining the flexible hose style you must decide if you want the fittings placed into either a flat structure or a subassembly.

Flat structure All parts are independent components placed along with all other components under the pipe run. There is no subassembly.

Subassembly structure All parts are grouped into a subassembly under the pipe run.

The route structure used for your design is typically determined by how the hose route parts are purchased, assembled, and represented in manufacturing documentation such as Parts Lists and Bills of Materials (BOM).

Create Flexible Hose Styles

In the following exercises, you create several hose styles based on the predefined Hydraulic Hose - Female Thread - Swivel style.

Create a Flexible Hose style with both fittings

- 1 In the style browser, select Hydraulic Hose - Female Thread - Swivel.
- 2 Copy and edit the style as you did previously.
- 3 Enter the new style name, *Hydraulic Hose- Female Thread - Swivel (1/2 ND 2)* to indicate the nominal size and two hose fittings, and click Save.

- 4 On the General tab, select 1/2 in from the Diameter list.
- 5 On the Rules tab, verify that .125 in is the Hose Round Up value.
- 6 Click Save.

Create a Flexible Hose style with start fitting only

- 1 Create another style based on Hydraulic Hose - Female Thread - Swivel and save it as *Hydraulic Hose- Female Thread - Swivel (1/2 ND 1)*.
- 2 Under Components, select the End Fitting row, right-click and select Suppress Fitting.
- 3 Select 1/2 in from the Diameter list.
- 4 Accept the other default settings.
- 5 Click Save.

Create a new Flexible Hose type style with no fittings

- 1 Create a third style based on Hydraulic Hose - Female Thread - Swivel and save it as *Hydraulic Hose- Female Thread - Swivel (1/2 ND 0)*.
- 2 Right-click the copy and select Edit.
- 3 Under Components, right-click the Start Fitting row and select Suppress Fitting.
If the end fitting is not suppressed, you are prompted that the end fitting will also be suppressed. Click Yes.
- 4 Select 1/2 in from the Diameter list.
- 5 Accept the other default settings.
- 6 Click Save.

Change Styles for Existing Routes

Design needs change as the routed system evolves. With styles, you can make subtle or dramatic changes quickly and efficiently.

For example, it is common to create routed systems using less expensive manufacturing methods at the early design stages, then switch to the actual, more expensive manufacturing methods at the end.

Change the style of an existing route

- 1 Activate the route for which you want to change the style. If desired, open a tube and pipe assembly from the Samples directory to perform this operation.
- 2 On the Route tab, Manage panel, select a new style from the Active Style list.

The model is updated to conform to the new style. The changes are also reflected in the browser.

NOTE When switching a route from a Tubing with Bends style to a Rigid Pipe with Fittings style, the results may be unpredictable. You cannot switch an existing route between the rigid piping and flexible hose styles.

Change Active Styles for New Routes

Routes in the same system often have different uses and different requirements. To accommodate these changes, you can select a new style with the settings needed for each different route. If you change or edit a style that is in use by one or more routes, the style change or edits will affect all routes that use that style.

Change the active style for new routes

- 1 Activate the master runs assembly or an individual pipe run.
- 2 On the Tube and Pipe or Pipe Run tab, Active Style list, select Welded Steel Pipe - ASTM A53/A53M - ASME B16.11 (1/4 ND).

Add Styles to Assembly Templates

Within an Autodesk Inventor assembly, the tube and pipe runs subassembly is created from the master tube and pipe runs assembly template, *pipings runs.iam*. This file is stored in the installation path of Autodesk Inventor Professional Tube & Pipe, by default, **Windows® XP** *C:\Program Files\Autodesk\Inventor <version>\Design Data\Tube & Pipe* and *C:\Users\Public\Documents\Autodesk\Inventor <version>\Design Data\Tube & Pipe* for **Windows Vista®**.

As with Autodesk Inventor assembly templates, Autodesk Inventor Professional Tube & Pipe provides two types of units for the template: English and Metric.

When you install Autodesk Inventor Professional, your selection of default units of measurement sets the default template used to create standard Inventor assemblies and the default template used to create tube and pipe runs subassemblies. For example, if you set English (Inches) as the default unit of measurement during installation, the template in the English subdirectory is copied to the *Design Data\Tube & Pipe* directory.

When you customize a set of tube and pipe styles, they are saved in the local piping runs assembly that you name and locate when you first start your tube and pipe assembly. To reuse them in future tube and pipe assemblies, you can export the styles from the local piping runs assembly, and then import them into a (blank) master runs assembly template. Once added to the master runs assembly template, your styles can be used in other tube and pipe assemblies.

NOTE You must check the Use Styles Library option for the project. If it is set to No, you may receive a warning message in this exercise. You must set the option to either Yes or Read Only. Otherwise, you cannot continue defining tube and pipe styles and populate routes in the exercises that follow.

Add custom styles to an assembly template

- 1 Create a new, empty assembly file.



- 2 On the ribbon, click Environments tab ► Begin panel ► Tube and Pipe.

- 3 When prompted, save the assembly file. On the Create Tube and Pipe Run dialog box, click OK to save the file using the defaults.

The main tube and pipe runs assembly is automatically named and located, and the first run is activated.



- 4 With the master runs assembly or the first run active, click the Tube & Pipe Styles tool. Create new styles, modify existing styles, or import styles as required. Save the changes and then click OK.
- 5 Right-click and select Finish Edit.
- 6 Activate the master runs assembly, right-click the individual run Run01, and then select Delete Run.
- 7 Click OK to confirm.

- 8 With the top assembly active, save the file.
- 9 In the graphics window right-click and select Generate Tube & Pipe Template.

The *pipng runs.iam* file is created in the *Design Data\Tube & Pipe* directory along with a backup of the original template. The file name for the backup is *pipng runs-old(1).iam* where 1 is an incremental number starting with 1. The default installation location for **Windows XP** is *C:\Program Files\Autodesk\Inventor <version>\Design Data\Tube & Pipe*. The default installation location for **Windows Vista** is *C:\Users\Public\Documents\Autodesk\Inventor <version>\Design Data\Tube & Pipe*.

Create Rigid Routes and Runs

4

Rigid routes are defined by selected route points. The rigid route styles, Rigid Pipe with Fittings and Tubing with Bends, contain the rules for route components such as conduit parts and elbows. Couplings, welds, or flanges connect straight segments. Elbows, bends, welds, or flanges connect each directional change point. A populated route is created based on the tube and pipe style and the defined route path through the assembly.

In this chapter, you learn about the methods and tools for creating rigid piping and bent tubing routes with auto regions and parametric regions, and populating selected routes with library components.

Workflow for Rigid Routes

The following is a typical workflow of creating a rigid route with a mixed set of auto regions and parametric regions.

Create a rigid route

- 1 Start with a tube and pipe assembly.
- 2 With the top assembly or master runs assembly active, right-click the master runs assembly, Tube & Pipe Runs by default and select Tube & Pipe Settings to set the global document settings and application settings for all routes and runs.

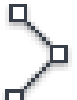


- 3 On the ribbon, click Environments tab ► Begin panel ► Tube and Pipe to add new runs.

- 4 If pipe run is not activated, right-click the pipe run and select Edit.



- 5 Select an active tube and pipe style. Optionally, you can create new styles, modify existing styles, and import styles from other routes or tube and pipe assemblies using the Tube & Pipe Styles tool.



- 6 With the pipe run active, click Pipe Run tab ► Route panel ► New Route. Enter a new file name or modify the default file name and location.



- 7 On the ribbon, click Route tab ► Create panel ► Route to start defining the start point on valid geometry.
- 8 Use the 3D Orthogonal Route tool to continue selecting valid geometry to create auto route regions, or use geometry constraints, point snap, or specific dimensions to create sketched route segments.
- 9 Right-click and select Finish Edit.
- 10 Populate the route, and set the conduit part file names if you have enabled the Prompt for Conduit File Names option on the Tube & Pipe Settings dialog box in Step 2.
- 11 To continue editing the route, if you are working with a large tube and pipe assembly, use the Display/Update Settings and Defer All Updates tools to prevent automatic updates and save memory cost.
- 12 Optionally:
 - Add additional route points as you create or edit a route.
 - Adjust route points and segments dynamically or by placing geometric constraints and dimensional constraints.
 - Change the active tube and pipe style.
 - Place fittings and conduit parts from the Content Center using AutoDrop or from the active project workspace.

- Connect fittings, routes, runs, tube and pipe subassembly, and Autodesk Inventor® components.
- Use the Model browser to visualize the tube and pipe run structure or change visibility for conduit parts and fittings.

13 Save the top assembly.

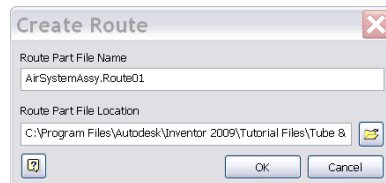
It is recommended that you save the top assembly regularly. Tube and pipe components are not stored in your project workspace until the top assembly is saved.

Create Auto Route Regions

In this exercise, you quickly define a route by selecting points using circular geometry and allowing the system to generate the rest.

Create an auto region

- 1 In the *AirSystemAssy.iam* assembly, activate AirSystem1:1.
- 2 On the Pipe Run tab active style list, verify that Welded Steel Pipe - ASTM A53/A53M - ASME B16.11 (1/4 ND) is the active style.
- 3 On the ribbon, click Pipe Run tab ► Route panel ► New Route.
- 4 On the Create New Route dialog box, accept the default route file name and location, and then click OK.



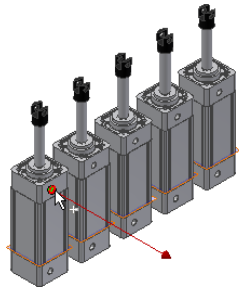
A pipe route is added to the active run and activated in place. The Route tab is displayed.



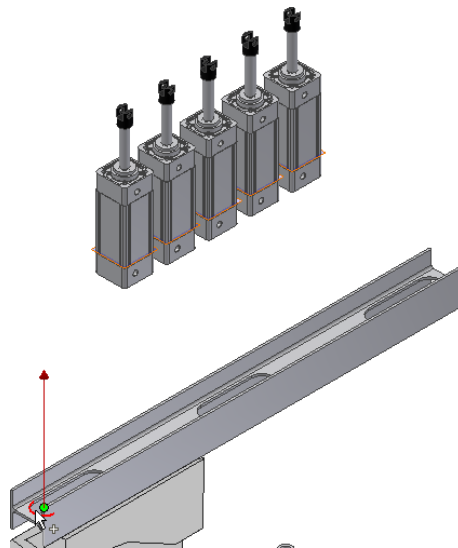
- 5 On the ribbon, click Route tab ► Create panel ► Route.

- 6 In the graphics window, pause the cursor over the circular geometry as shown in the following image. Zoom in and use the Select Other Direction tool if needed to make your selection.

Once preselected, the circular geometry highlights and the line extender displays, showing the direction of the route so you can create valid points along the line. Press the spacebar to change the direction of the line extender if needed.



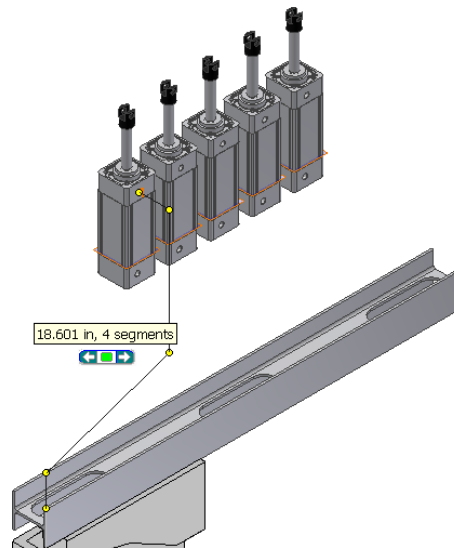
- 7 Click the circular geometry to select it as the start point. This point is associated with the component that contains the selected circular edge. A work point is added to the piping route.
- 8 For the second route point, you can select a point on the line extender or existing geometry. To route through the IBeam, pause the cursor over the left arc of the IBeam opening.



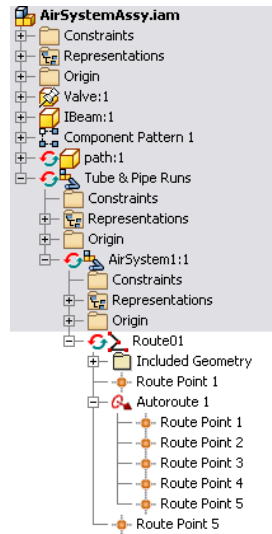
Rotate and zoom in to view the arc. Make sure the line is pointing to the approach direction of the route.

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

- 9 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.



- 10 Click the arrows to cycle through the solutions. Click the green middle button to select the first solution as shown in Step 9.
- 11 In the Model browser, verify that the Autoroute browser node is added and contains the indented auto route points.



- 12 Right-click the Autoroute browser node and select Alternate Route Solution.
You can use the Select Other tool to switch the auto region solution as needed.
- 13 Ensure that you click the green middle button on the Select Other tool to go back to the previously-set solution before you continue.

Manually Create Parametric Regions

Many runs involve more complex routing, which requires the creation of additional route points. Tube & Pipe in Autodesk Inventor® Routed Systems provides a variety of route tools to dimension or constrain the route against existing geometry and construction lines.

In this chapter, you learn about how to use the General Dimension tool, Parallel With Edge, Perpendicular To Face, Point Snap and Rotation Snap, Bend and Custom Bend, and 3D Orthogonal Route tool.

For detailed instructions about other parametric route tools, refer to the Help.

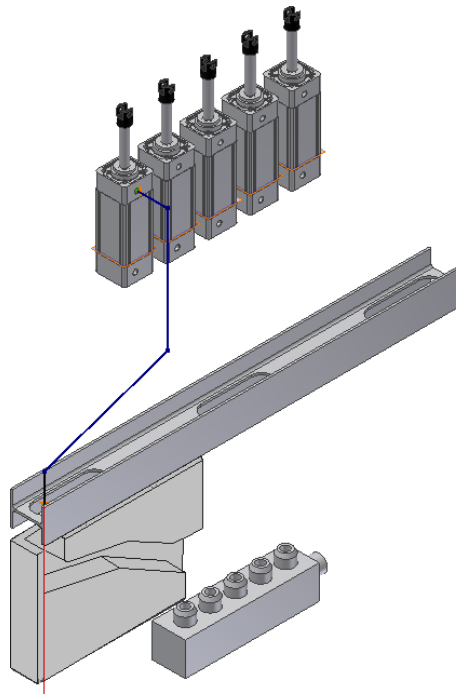
Automatically Dimension Route Sketches

When you manually define the next route point, the system automatically adds dimensions to the route sketch by default. You can disable the Auto-Dimension option on the right-click menu.

In the exercises that follow, you allow the system to automatically dimension the route sketch in parametric regions so you accept the default setting.

Automatically dimension the parametric route sketch

- 1 In the Route1 environment, ensure the 3D Orthogonal Route tool is active for forward creation. If the line extender is not displayed, right-click the last route point, and click the Route tool.



- 2 Right-click in the graphics window and verify that Auto-Dimension is enabled.

Create Segments With Precise Values

When the 3D Orthogonal Route tool is active, you can define the segments with precise values using any of the following ways:

- Move the cursor over the direction axis of the 3D Orthogonal Route tool, right-click, and then select Enter Distance.
- Pause the cursor over the direction axis, and then directly type the value from the keyboard. The Enter Distance dialog box is automatically displayed.

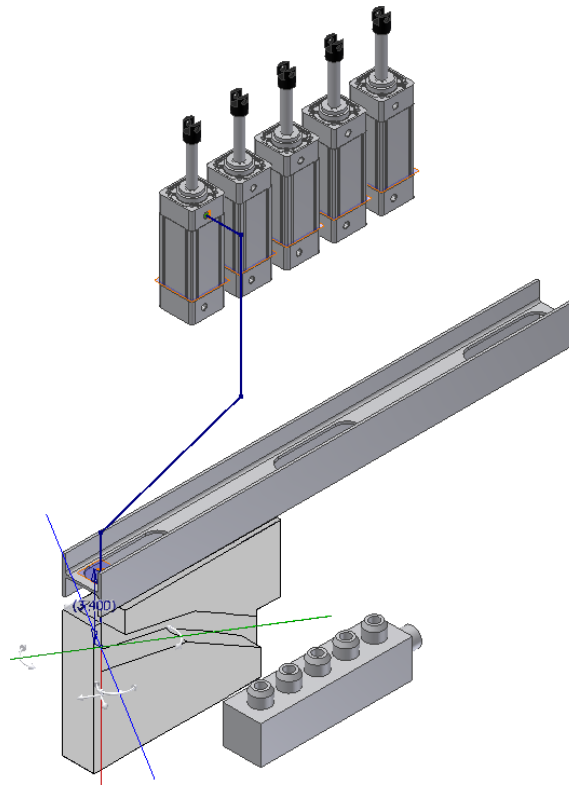
When you manually define the next route point, the system automatically adds dimensions to the route sketch by default. You can disable the Auto-Dimension option on the right-click context menu.

In the exercises that follow, you allow the system to automatically dimension the route sketch in parametric regions so you accept the default setting.

Create segments using precise values

- 1 Ensure the 3D Orthogonal Route tool is active on the last route point.
- 2 Pause the cursor over the line extender, right-click and select Enter Distance.
- 3 Enter a distance of 3.4 inches for the next point, and then click the green check mark.

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



Define Parallel and Perpendicular Segments

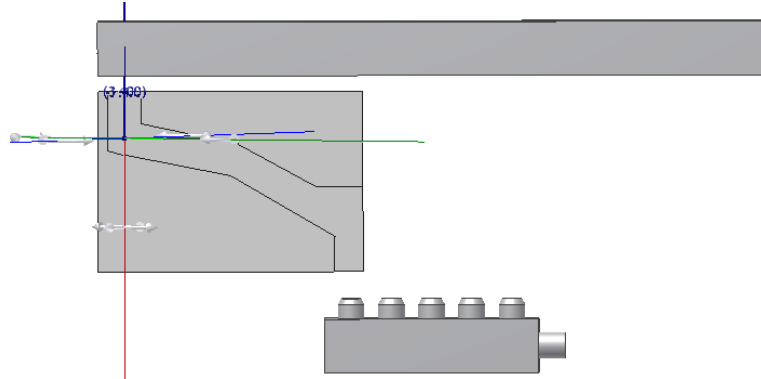
In this exercise, you reorient the direction axes of the 3D Orthogonal Route tool parallel and perpendicular to the edges and faces on the concave path part. The next segments can then fit in the path part.

Applying the Parallel With Edge tool to a certain linear geometry may require a bend if directional fittings in the Content Center such as 45-degree or 90-degree elbow are not appropriate.

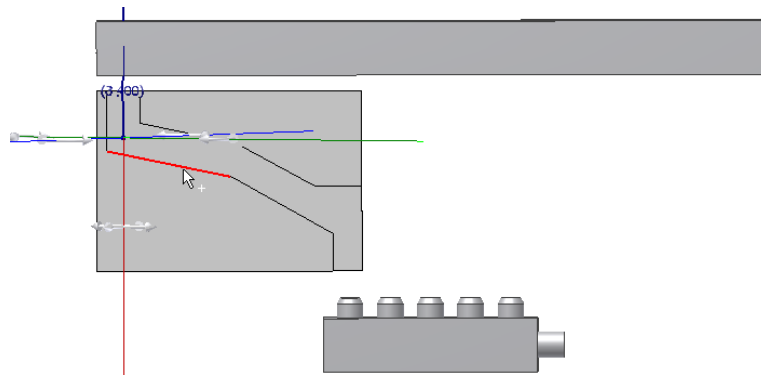
Create segments using existing geometry

- 1 Rotate your view to look at the front of the assembly.
- 2 Pause the cursor over the red direction axis of the 3D Orthogonal Route tool, right-click and select Parallel With Edge.

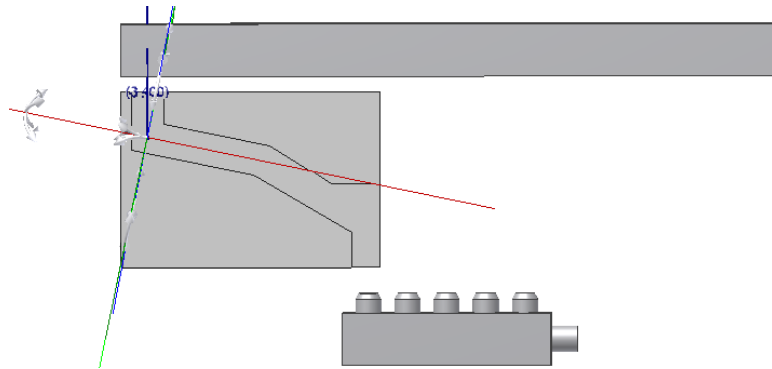
The system prompts you to select a linear edge.



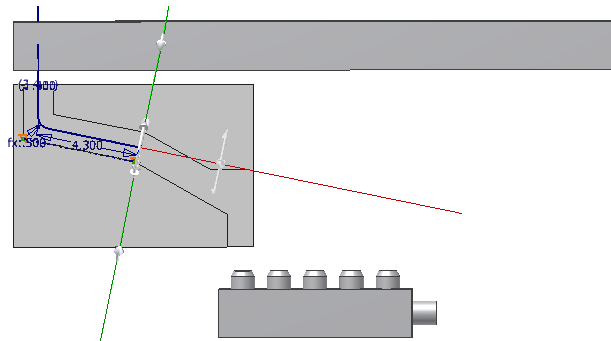
- 3 Select the linear edge on the path part as shown in the following image, and click to set the selection.



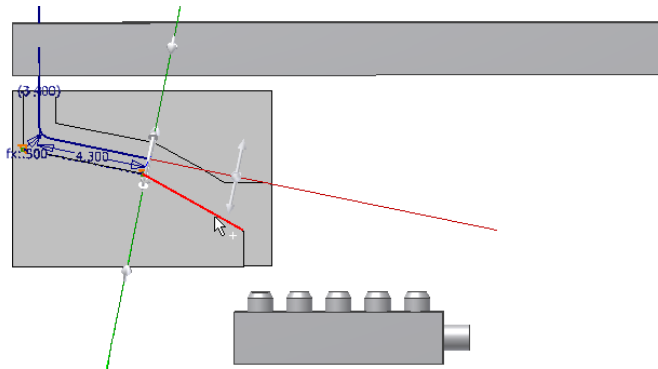
The red direction axis is reoriented to the direction of the selected edge. The included reference geometry is added to the Included Geometry folder in the Model browser.



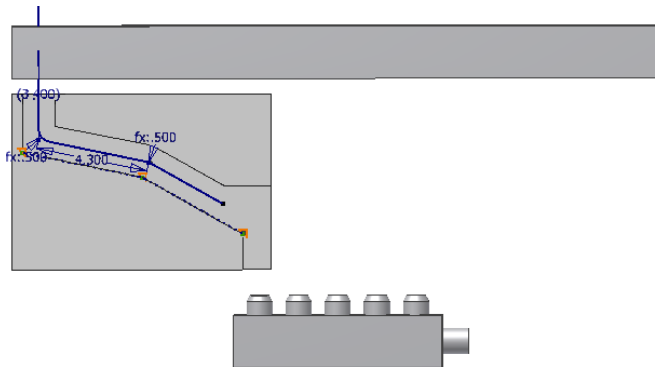
- 4 Pause the cursor over the red axis, and then enter 4.3.



- 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.



In the exercise that follows, you make the last route point coincident with a newly created route point. The last segment must not be dimensioned so that it can dynamically update to a new segment length. Otherwise, the route sketch may be over-constrained.

NOTE If you did not turn off the Auto-Dimension option, the last segment is dimensioned. To dynamically update to later edits, you can also manually delete the linear dimension. Right-click the dimension, and select Delete.

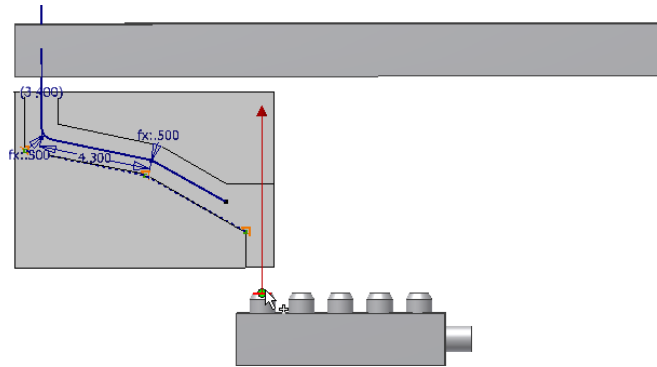
Snap Route Points to Existing Geometry

The Point Snap tool helps you to create route points by snapping to faces, edges, work points, or vertices.

In this exercise, you continue defining the route point by snapping to the path part.

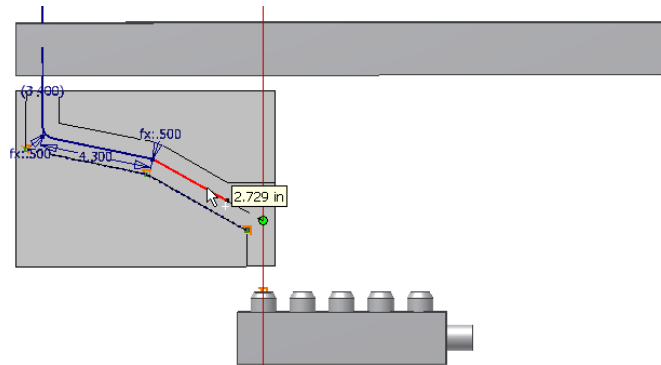
Snap route points to existing geometry

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

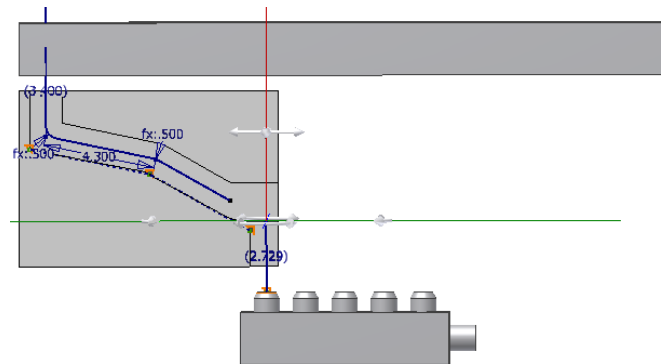


- 3 Right-click in the graphics window and select Point Snap.
- 4 Move the cursor over the last segment you just created in the preceding exercise.

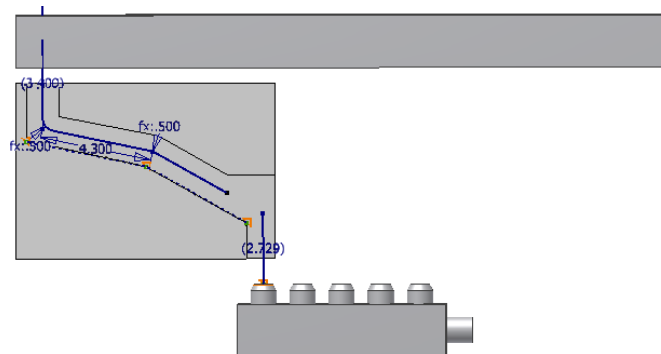
A dashed line representing the snap point is displayed from the cursor to the preview point. A preview point is displayed at the intersection of the line and the axis.



5 When the lines appear, click the segment to set the snapped route point.



6 Right-click and select Done.




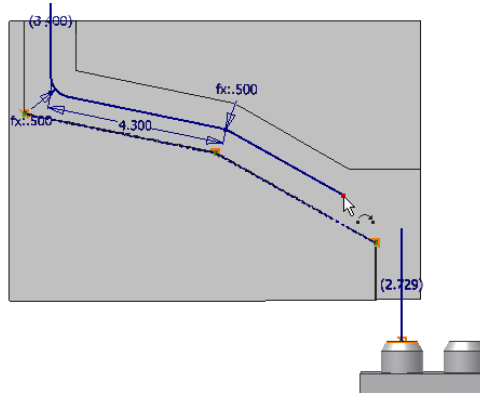
Place Constraints On Route Sketches

As with the 3D sketch in Autodesk Inventor, you can place geometric constraints on the route sketch to constrain route points and segments. Available 3D sketch constraints include Perpendicular, Parallel, Tangent, Coincident, Collinear, and Fix. They help create your tube and pipe system in the appropriate direction and orientation and constrain it to assembly changes, join route points, and make route points fixed where dynamic updates are not allowed.

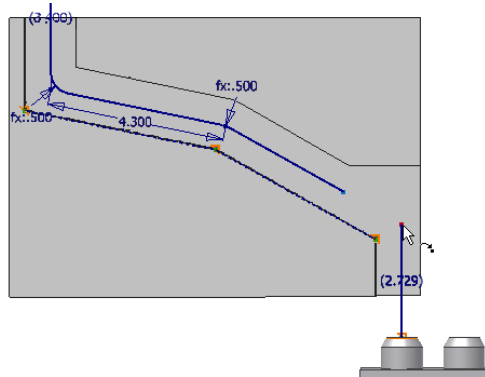
In this exercise, you place a coincident constraint on the two separate route points to join two route segments.

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 images.

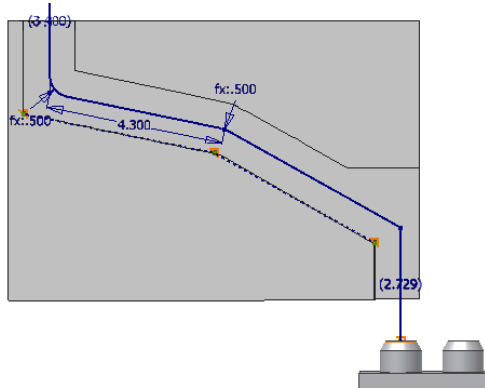


First point to select

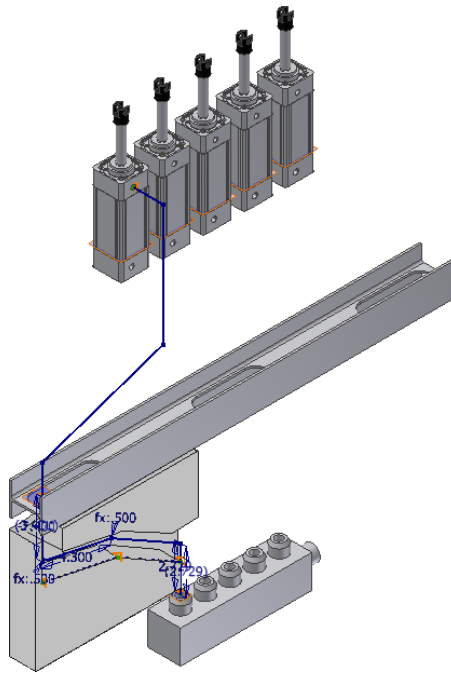


Second point to select

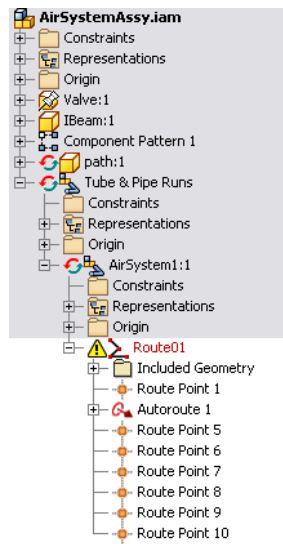
The separate sketches form a closed route.



- 3 Right-click in the graphics window and select Done.
- 4 Right-click again and select Finish Edit.



Notice that an error icon displays beside the route.



Placing the coincident constraint causes an odd (not 90-degree, 45-degree or combined 135-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.

Create Bends Between Existing Pipe Segments

The rigid piping styles allow only the 90-degree, and 45-degree angles (when 45-degree is allowed in the style) and also the combined angles of 135-degrees. When the Parallel With Edge or Perpendicular To Face tool is applied, bends are automatically created where an odd angle is required.

When an odd angle exists between the two joint pipe segments, violations occur. Use the Bend tool to create a bend so that the route is valid.

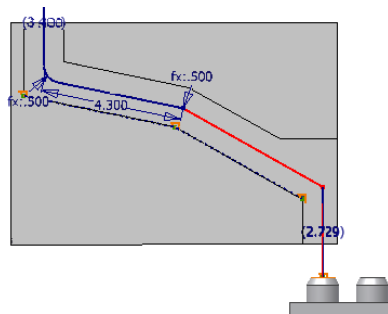
You can manually define the bends using the Custom Bend tool. It is applied before you create the next segment.

In this exercise, you use the Bend tool to fix the invalid angle violation in the preceding exercise.

Join separate route points with the coincident constraint

- 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 an error on the Show Violations dialog box, and then view the error in the graphic window.

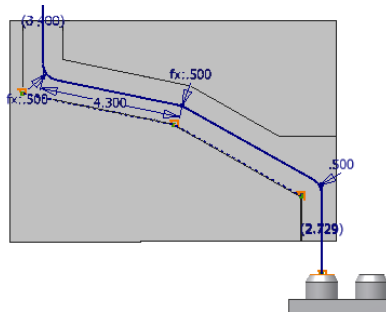
The invalid angle is between the two segments as highlighted in the following image.



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



- 4 On the ribbon, click Route tab ► Create panel ► Bends.
- 5 Accept the default settings on the Bend dialog box, and then select the two segments that include 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 again and select Finish Edit.

Create Pipe Routes With Custom Bends

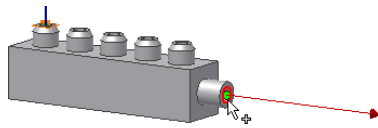
Rigid pipe routes with fittings may require a custom bend to satisfy special manufacturing situations. To add a custom bend, use the Custom Bend option in the right-click context menu while creating a pipe route. The 3D Orthogonal Route tool changes to include the tube routing tools so you can create a bend with the radius and angle you need. You can create multiple custom bends in any given route.

In this exercise, you add a second route in the AirSystem1 run, change the active style, and then create a route that is offset from the circular edge. You also use the 3D Orthogonal Route tool to create additional points, including custom bends.

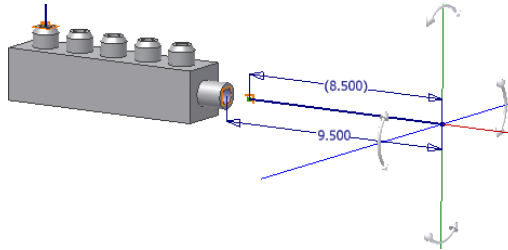
Add custom bends

- 1 In the *AirSystemAssy.iam* assembly, activate the AirSystem1:1 run.

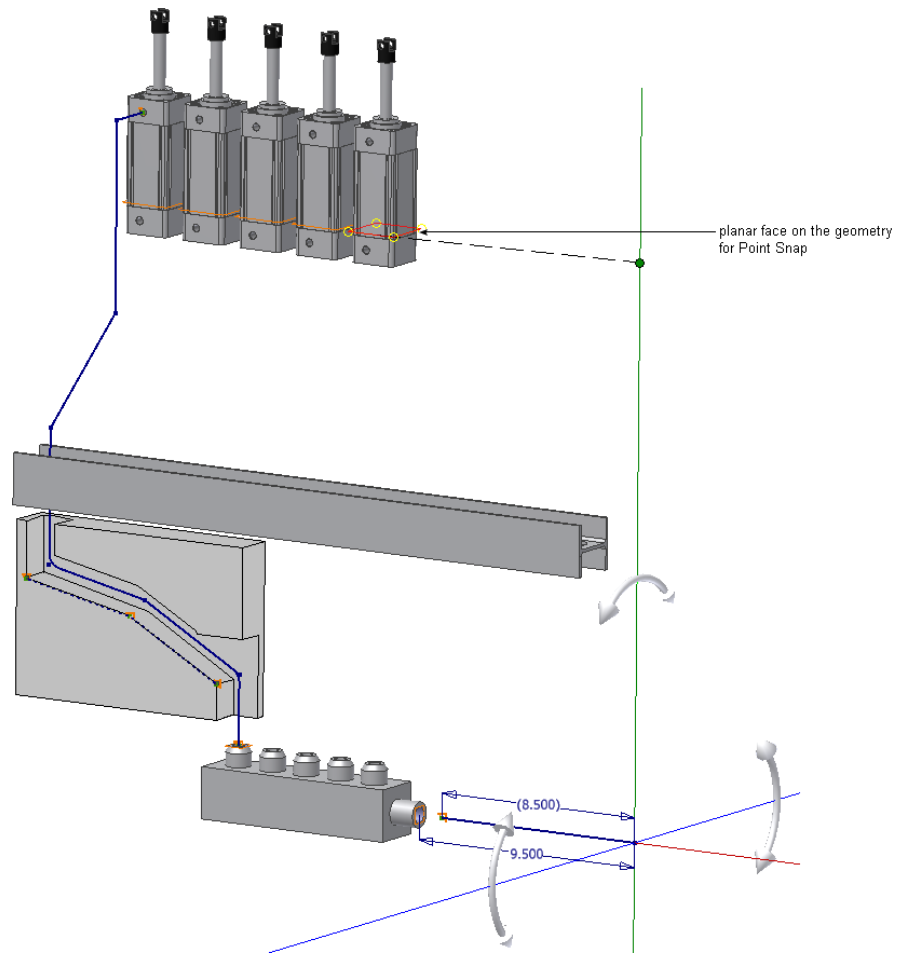
- 2 On the Pipe Run tab, Manage panel, Active Style list, set the Active Style to Threaded Steel Pipe with Iron Fittings (1/2, 90 Only).
- 3 On the ribbon, click Pipe Run tab ► Route panel ► New Route, and then accept the default route file name and location to continue.
- 4 On the ribbon, click Route tab ► Create panel ► Route to start defining the start point on valid geometry.
- 5 Pause the cursor over the circular edge as shown in the following image, enter a distance of 1 inch, and then click the green check mark.
The point is created offset from the circular edge by that distance.



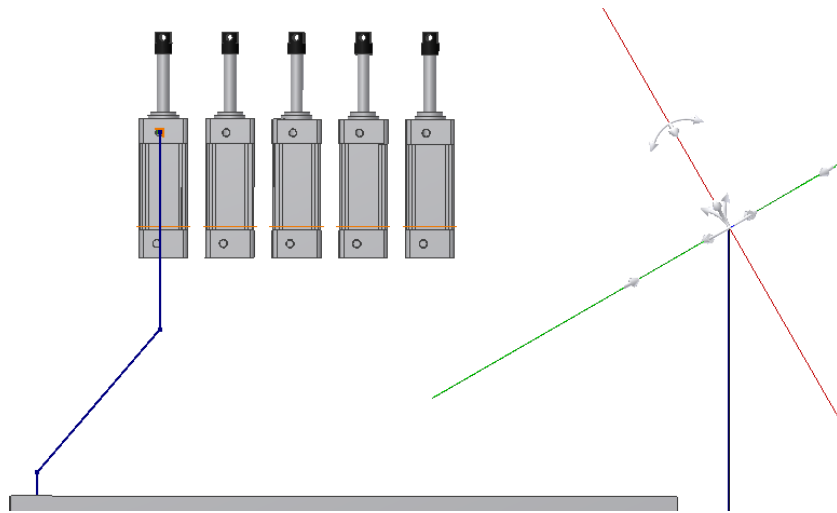
- 6 Move the cursor along the line extender and enter 8.5.
You can enlarge or reduce the 3D Orthogonal Route tool by pressing plus (Shift and +) and minus (-) on the keypad.



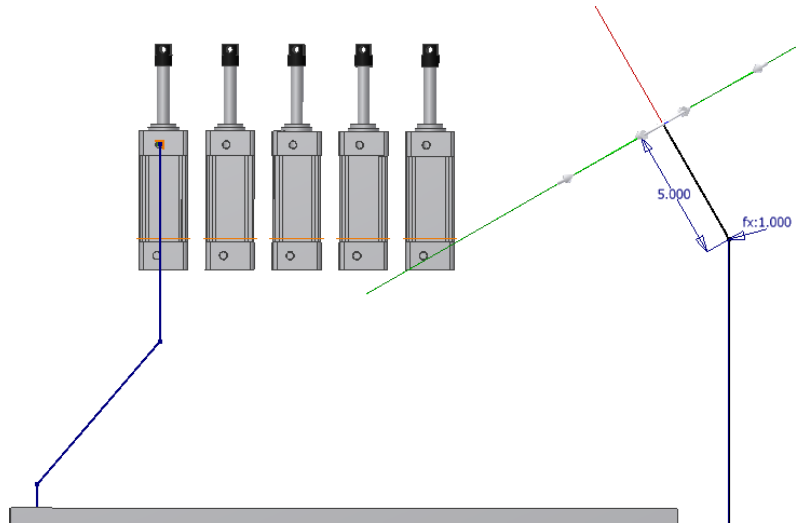
- 7 Enlarge the 3D Orthogonal Route tool until the line reaches the top of the model.
- 8 Make sure Point Snap is on, and then move the cursor over the work plane to snap the point onto the 3D Orthogonal Route tool. Click to create the point.



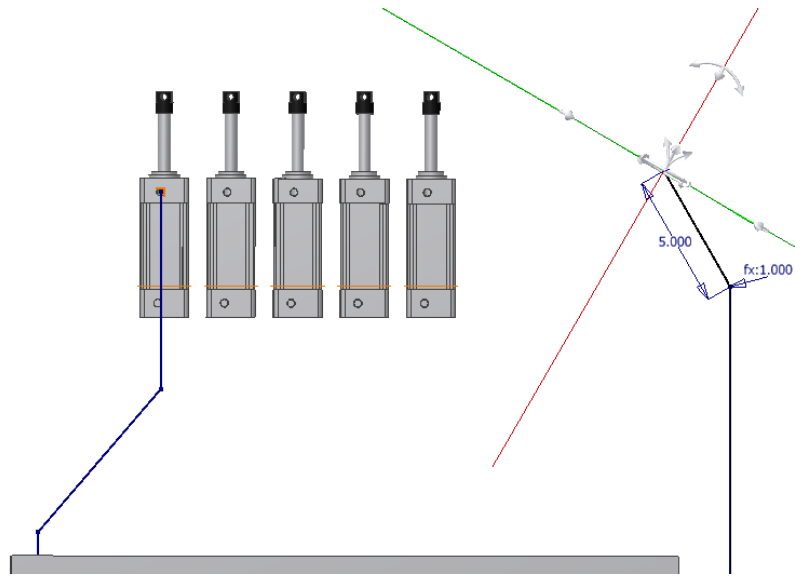
9 Rotate the view to the front of the assembly.



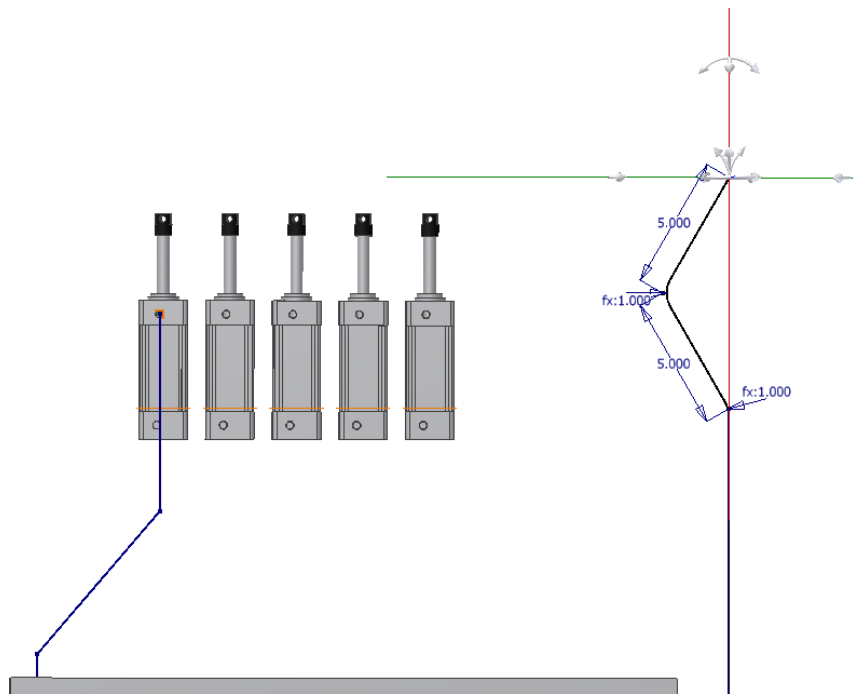
- 12** Pause the cursor over the red axis, and then enter a distance of 5 inches. The 3D Orthogonal Route tool reverts to displaying the pipe routing elements.



- 13** Right-click and select Custom Bend. Add a bend of 60 degrees in the other direction, as shown in the following image.

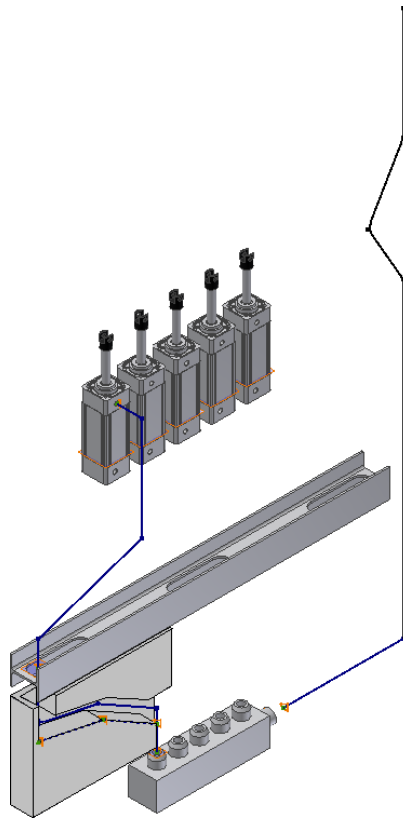


- 14 Pause the cursor over the red axis, and then enter a distance of 5 inches.
- 15 Right-click, select Custom Bend, and then create a bend of 30 degrees as shown in the following image.



- 16** Pause the cursor over the red axis, and enter 8 inches.
- 17** Right-click and select Done.
- 18** Right-click and select Finish Edit.

The linear segments adjacent to the custom bend sequence are joined with the bend to create a single pipe segment with bends, rather than custom elbows.



Create Bent Tube Routes

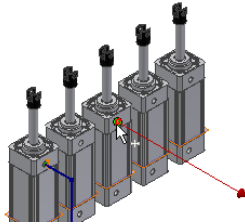
Bent tube type routes can be created at any angle and using any bend radius.

For this exercise you select the Soldered Copper Tube (1/2 ND,.5 Bend) style you created earlier, and then create a bent tube route by directing it along the IBeam and over to the threaded steel route to the right of the model.

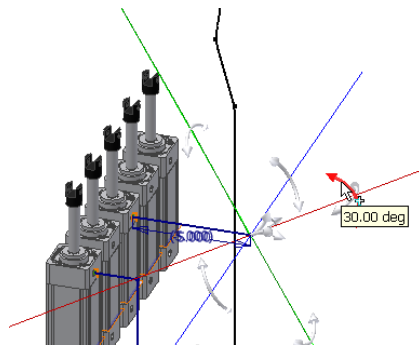
Create a bent tube route

- 1 In the *AirSystemAssy.iam* assembly, activate the AirSystem1:1 run.
- 2 On the Pipe Run tab, Manage panel, Active Style list, select Soldered Copper Tubing (1/2 ND,.5 Bend).

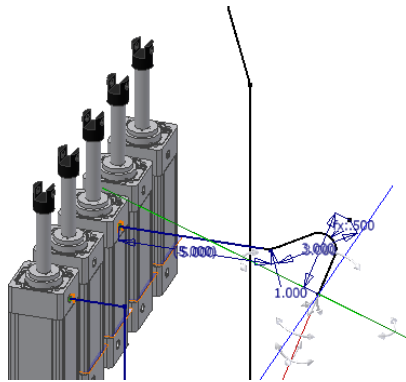
- 3 On the ribbon, click Pipe Run tab ► Route panel ► New Route, and then accept the default route file name and location to continue.
- 4 On the ribbon, click Route tab ► Create panel ► Route to start defining the start point on valid geometry.
- 5 Select the start point of the route.



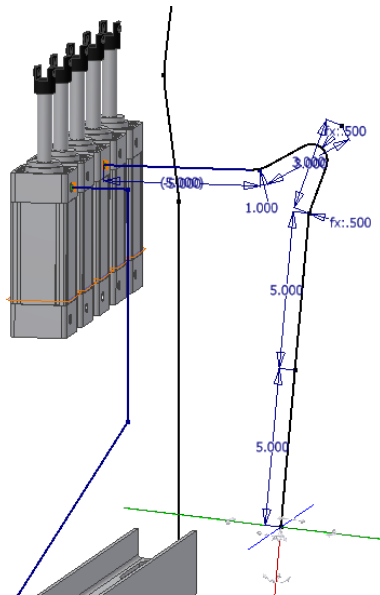
- 6 Enter a distance of 5 inches for the first segment.
The 3D Orthogonal Route tool is displayed with the tube routing elements.
- 7 Click the Radius handle and enter a bend radius of 1 inch.
- 8 Rotate to get a better view of the tool, select the Rotation handle, and then drag up by 30 degrees.



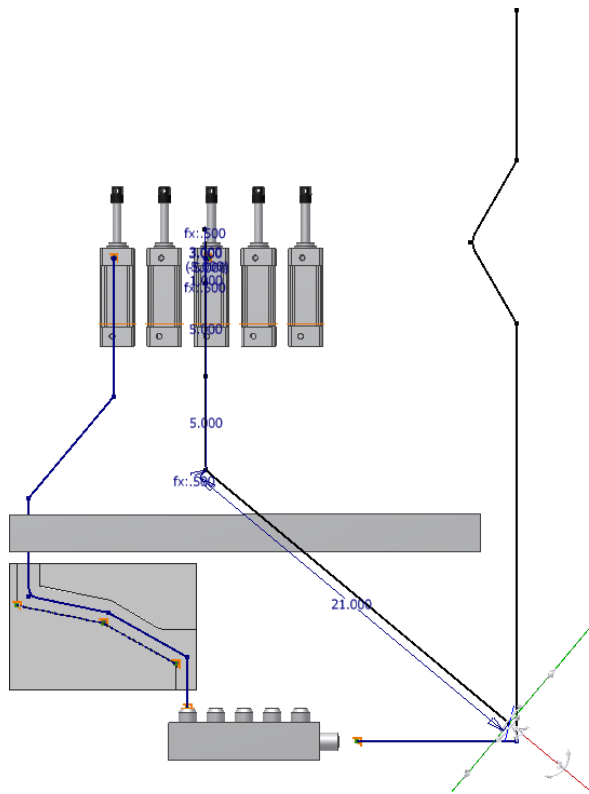
- 9 Pause the cursor over the red axis, and enter a distance of 3 inches.
- 10 Select the Rotation handle, and drag down 140 degrees.
- 11 Pause the cursor over the red axis, and enter a distance of 3 inches.



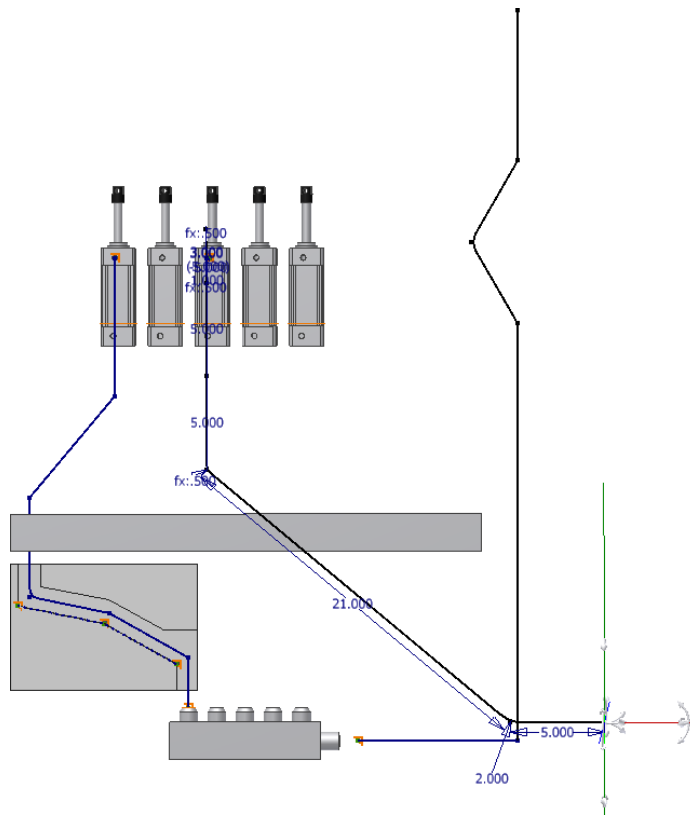
- 12 Click and drag the rotation handle 15 degrees to the right, and then create two segments of 5 inches along the red axis.



- 13 Rotate the view to look at the front of the model. Move the cursor over the Rotation handle, and then enter 50 degrees to drag to the right.
- 14 Pause the cursor over the red axis, and then enter a distance of 21 inches for the next route point.



- 15 Enter a bend radius of 2 inches, click and drag the rotation handles up by 40 degrees, and then enter a distance of 5 inches along the red axis.



- 16 Right-click and select Done.
- 17 Right-click and select Finish Edit.

Realign 3D Orthogonal Route Tool

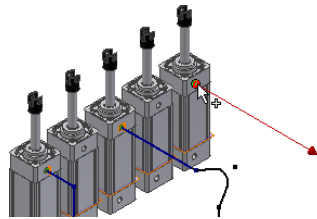
The Rotation Snap tool assists you to reorient a specified direction axis of the 3D Orthogonal Route tool by snapping to available linear edges and planar faces. A custom bend is automatically created at the current route point when the next segment is defined. It is useful when the direction axes of the 3D Orthogonal Route tool does not align with the needed geometry.

In this exercise, you create a new route with the Welded Steel Pipe - ASTM A53/A53M - ASME B16.11 (1/4 ND) style, and then review the skills you have learned about the Point Snap tool to create route points. Finally, use the

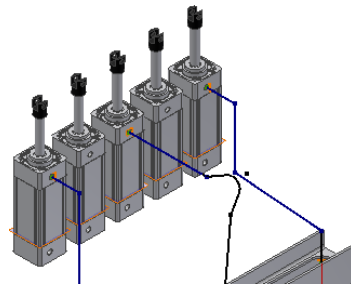
Rotation Snap tool on the right-click context menu to reorient the direction axes of the 3D Orthogonal Route tool by snapping to the valve opening.

Snap the axis rotation to existing geometry

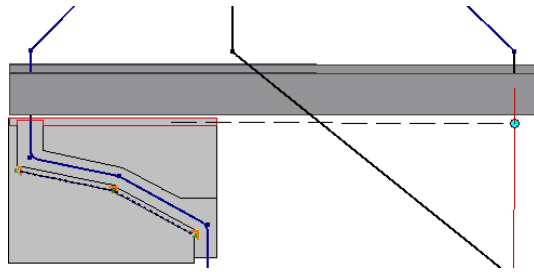
- 1 In the *AirSystemAssy.iam* assembly, activate the AirSystem1:1 run.
- 2 On the Pipe Run tab, Manage panel, Active Style list, select Welded Steel Pipe - ASTM A53/A53M - ASME B16.11 (1/4 ND).
- 3 On the ribbon, click Pipe Run tab ► Route panel ► New Route, and then accept the default route file name and location.
AirSystemAssy.Route04 is added to the Model browser.
- 4 On the ribbon, click Route tab ► Create panel ► Route to start defining the start point on valid geometry.
- 5 Select the start point as shown in the following image.



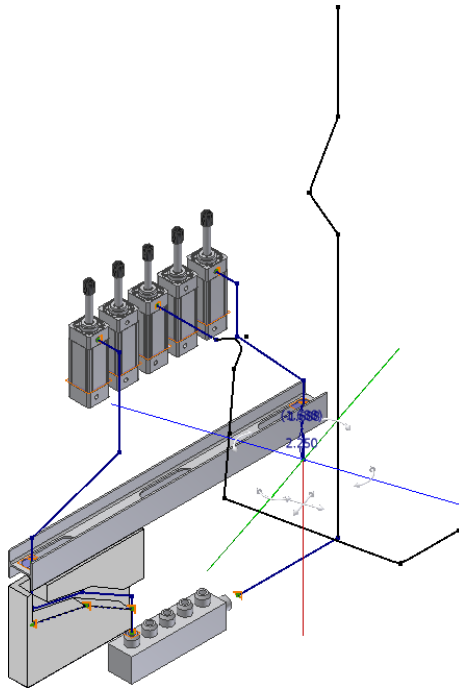
- 6 Click the right circular edge on the IBeam to allow the system to generate an auto region. Use the Select Other tool to cycle through all solutions as needed, and then select the first solution.



- 7 Right-click in the graphics window and select Point Snap.
- 8 Move the cursor over the upper face on the path part as shown in the following image, and then click to snap the new route point.
Rotate the view to get a better view as needed.

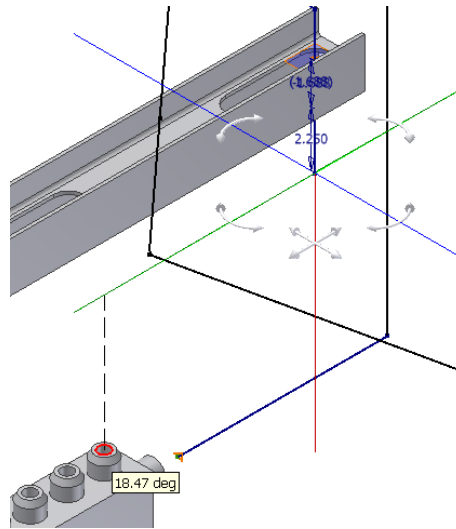


- 9 Pause the cursor over the line extender, and then enter a distance of 2.25 inches for the next point.
- 10 Rotate to the isometric view. Notice that the green axis does not align parallel to the valve part.

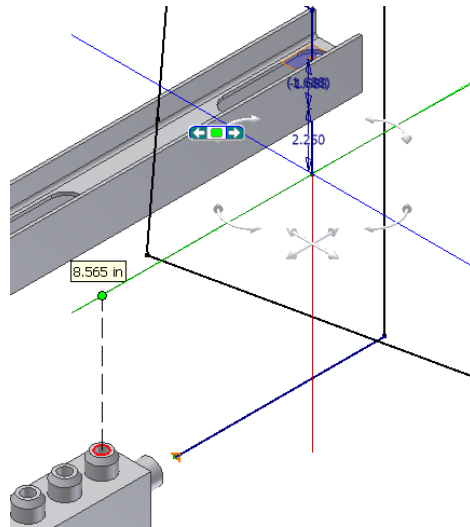


- 11 In this exercise, the green axis should be parallel to the valve part for the rest of the route. Right-click in the graphics window, and select Rotation Snap.

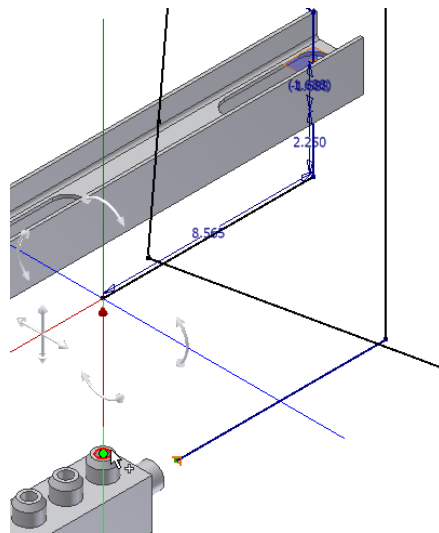
- 12** Click the Rotation handle on the green axis, and drag the cursor over the valve opening. Release the cursor.
The green axis changes to the new orientation. It is parallel to the valve part.



- 13** Right-click in the graphics window to ensure that Point Snap is enabled on the context menu. Highlight the circular edge in the valve opening as shown in the following image. Right-click and choose Select Other Snap.
A preview point displays at the intersection of the line and the axis. A dashed line representing the snap point displays from the cursor to the preview point.
- 14** When the lines appear, click the green button to set the snapped route point.

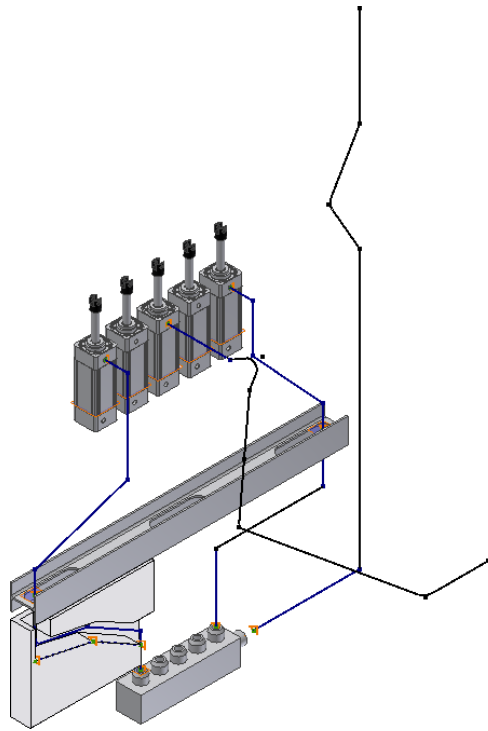


15 For the last point, click the circular geometry.



16 Right-click and select Done.

17 Right-click again and select Finish Edit.



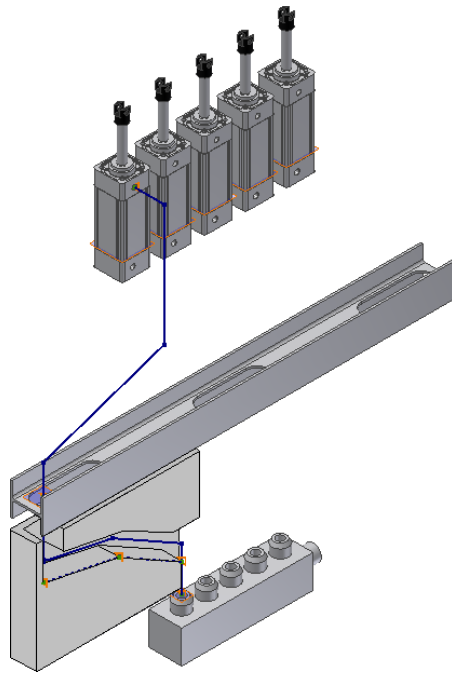
Control Dimension Visibility

In the route environment, dimensions in parametric regions remain visible by default. Auto regions do not involve dimensions so no associated dimensions are displayed.

Too many dimensions may block the view of the design window. In the exercise that follows, you practice controlling the dimension visibility on the route sketch using the Dimension Visibility tool.

Hide or display the route dimensions

- 1 Activate Route01.
- 2 Right-click the route in the Model browser, and then clear the Dimension Visibility check mark.
All dimensions become invisible.



- 3 To display the route dimensions, select Dimension Visibility on the right-click menu.
- 4 Right-click in the graphics window and select Finish Edit.
The run environment is activated.

NOTE In the run environment, the route dimensions are always invisible regardless of the Dimension Visibility setting by default. To display route dimensions in the run environment, right-click the route and select the Dimension Visibility check mark.

Populated Routes

A populated route is a solid, three-dimensional representation of the pipes and fittings. A route starts with a defined path, which you can populate with library components to complete the route. You can place fittings to the populated routes from the Content Center tool using AutoDrop.

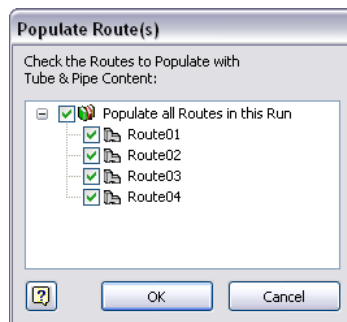
Tube and pipe information is treated like other parts and subassemblies and can be detailed using drawing manager methods and tools. You can document both populated and unpopulated routes. However, routes must be populated before being saved to the ISOGEN or bending machine formats.

Populate one or multiple routes

- 1 With the *AirSystemAssy.iam* assembly open, activate the AirSystem1:1 run.

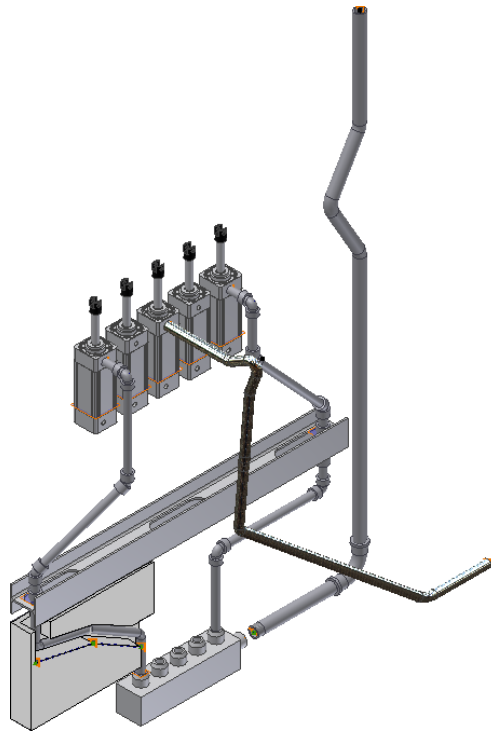


- 2 On the ribbon, click Pipe Run tab ► Route panel ► Populate Route.
- 3 In the Populate Route(s) dialog box, select the check boxes for the routes you need to populate. In this exercise, you populate all routes in the run.



Routes are populated using the styles you set. Each time a route is populated all segments and fittings created are added to the associated run folder.

- 4 Click OK. It will take a moment to populate the routes.



5 Right-click and select Finish Edit.

NOTE To populate a single route in a pipe run, you must activate the run and then click the Populate Route tool on the Pipe Run tab or right-click the route in the Model browser.

Create and Edit Flexible Hose Routes

5

Flexible hose routes in a tube and pipe assembly are commonly used in machine construction to transmit dynamic power, such as hydraulic and pneumatic power. They are defined by placing fittings and hose nodes that determine the shape and appearance of the route in the assembly. The flexible hose style controls which fittings are used and the structure of the flexible hose in the assembly.

In this chapter, you learn how to create a flexible hose style, and then use the styles to create flexible hose routes both in a flat structure and in a subassembly structure. You also learn to populate, edit, and delete the routes.

Workflow for Flexible Hose Routes

The workflow for creating a flexible hose route is as follows:

- 1 Create a tube and pipe assembly.
- 2 Optionally, drop fittings on appropriate pipe segments or assembly geometry to use as the start or end connection points for hose routes.

NOTE You can also start and end the hose route from any existing compatible geometry or component.

- 3 Define and select a flexible hose style, indicating the route structure and fittings to use.
- 4 Create a flexible hose route to connect the geometry or fittings in the tube and pipe assembly.

- 5 The workflow differs depending on the hose fitting definition that is specified in the flexible hose style. If the route contains:
 - Both start and end fittings, connect the start fitting and end fitting, and then add optional intermediate hose nodes.
 - A start fitting, connect the start fitting first, add optional intermediate hose nodes, and then select the end geometry or component.
 - No fittings (both are suppressed), select the start geometry or component, add optional intermediate hose nodes, and then select the end geometry or component.
- 6 Optionally, edit hose nodes and hose length.
- 7 Finish the edit and populate the route.

Create Flexible Hose Routes

With a flexible hose style active, click Pipe Run tab ► Route panel ► New Route to define a new hose assembly.

Many tools and guides used for creating hose routes are the same as those used for piping and tubing routes, such as the 3D Orthogonal Route tool and direction axes. For more information, see [3D Orthogonal Route Tool](#) on page 23 in Chapter 2.

You can insert intermediate nodes in the 3D hose spline. Valid points for intermediate flexible hose nodes are the same as those for rigid routes. In addition, you can select points offset from a face. For more information, see [Route Points](#) on page 22 in Chapter 2. Route points created by selecting arbitrary points offset from a face are not associative and do not update to changes in the model geometry.

NOTE Once a 3D hose spline is finished by right-clicking and selecting Done, you can only insert intermediate nodes onto the spline.

Although flexible route fittings are typically connected to other fittings in the active route or an adjacent route, several types of points are available for selection. Valid fitting connection points include:

- A connection on other tube and pipe fittings

- A connection on a standard Autodesk Inventor® part that has been authored using the Tube & Pipe Authoring tool and published to the Content Center
- The end of a tube, pipe, or hose segment
- A circular edge on any component

Create Hose Routes with Both Fittings

In this exercise you create a new run, and then use a flexible hose style you created earlier to create a hose route. The style specifies a subassembly structure that contains both a start and end fitting.

Create a hose route subassembly with both start and end fittings

- 1 Activate the master runs assembly, click Tube and Pipe tab ► Run panel ► Create Pipe Run. Enter *AirSystem2* for the run file name and accept the default file location.

The new run is activated.

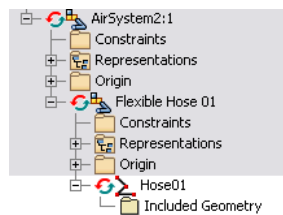
- 2 On the Pipe Run tab, Manage panel, select the Hydraulic Hose- Female Thread - Swivel (1/2 ND 2) style from the Active Style list.

- 3 On the ribbon, click Pipe Run tab ► Route panel ► New Route.

The Create Hose dialog box appears with the default file name and file location.

- 4 Accept the default hose assembly file name and location.

The Flexible Hose 01 subassembly is added under the AirSystem2:1run. The Hose1 route is activated by default.

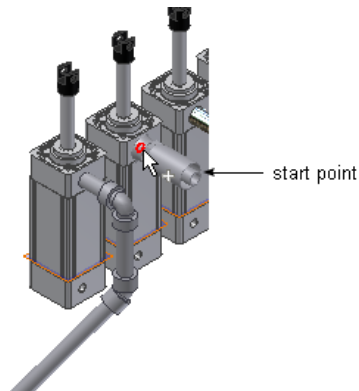


- 5 On the ribbon, click Route tab ► Create panel ► Route.

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



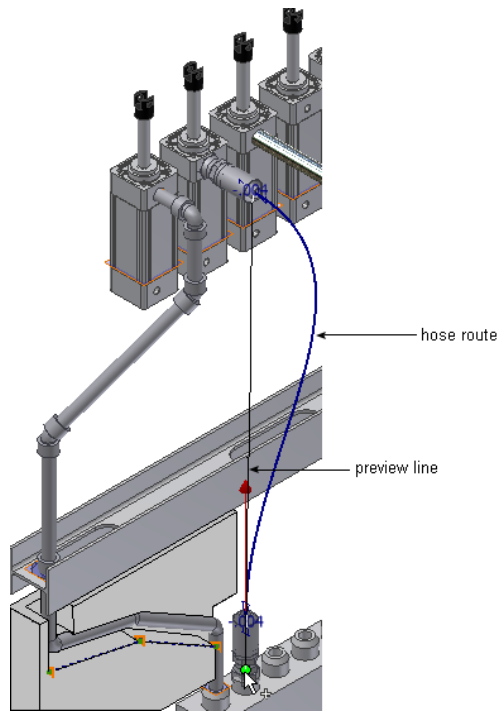
- 6 Move the cursor to the start route point on the geometry as shown in the following image, and then click to set the start fitting. The direction axis on the part should point in the direction of the route.
- 7 Change the direction for the fitting connection. Right-click and select Next Connection, or press the spacebar.



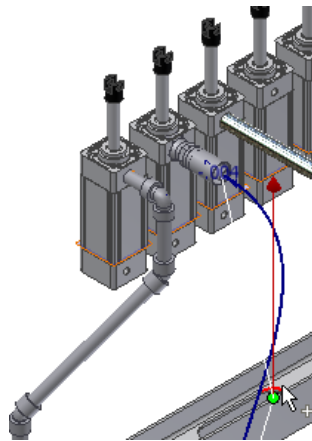
The end fitting appears and is attached to the cursor.

Notice that the start fitting and end fitting reference the same part file in the Content Center in this style.

- 8 Move the cursor to the circular opening on the valve part, and click to set the end fitting.



- 9 To add an intermediate node tangent to a circular edge on IBeam, move the cursor close to the circular edge, and then click to set the hose node. When the associated center node is selected, the circular edge is highlighted.

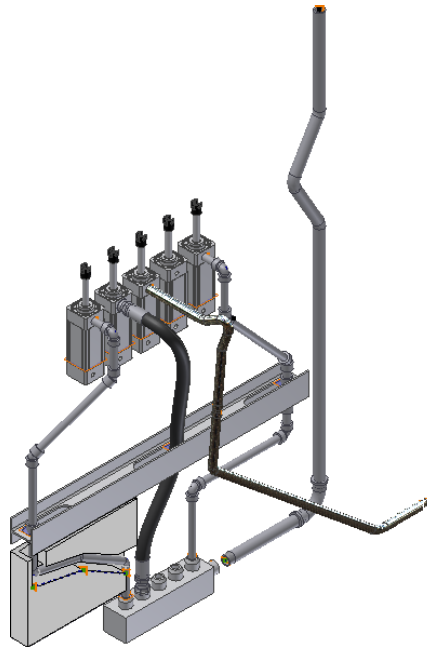


- 10 Right-click and select Done.
- 11 Right-click and select Finish Edit.
The run environment is activated.



- 12 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 at the bottom of the flexible hose assembly.



Change Nominal Diameters

The diameters of all components in a flexible hose route are controlled by the flexible hose style. You cannot modify those components directly using the Model browser.

When modifying the nominal diameter in the style:

- If the part family (hose, start fitting, and end fitting) contains the member with the desired nominal diameter, Tube & Pipe automatically locates the member listed under Components in the Tube & Pipe Styles dialog box.
- If the part family does not contain the member with the desired nominal diameter, you must search for and locate them. Under Components, right-click the appropriate row and select Browse to start the Library Browser tool, and then select one from the compatible parts list.

Change nominal diameters of hose route components

- 1 Activate the Hose 01 route in the AirSystem2:1 run.
- 2 On the ribbon, click Route tab ► Manage panel ► Tube and Pipe Styles. Verify that the Hydraulic Hose- Female Thread - Swivel (1/2 ND 2) style is active.
- 3 On the General tab, change the Diameter from 1/2 in to 5/16 inches.
- 4 Verify the list in the Components table. If Pipe, Start Fitting, or End Fitting is empty, click Browse, and then search for and locate a part from the compatible parts list.
- 5 Click Save.
- 6 Click Close.
- 7 Right-click and select Finish Edit.

The new diameter applies to the start fitting, hose segment, and end fitting in the hose route. Autodesk Inventor Tube & Pipe recomputes the hose route.

Create Hose Routes With One Fitting

In this exercise, you place a threaded tee onto the threaded steel pipe you previously created, and create a hose route using the Hydraulic Hose- Female Thread - Swivel (1/2 ND 1) style. This flexible hose style suppresses the end fitting so the hose route ends at the tee. The fitting is set as suppressed in the style definition.

Create a hose route

- 1 Activate the AirSystem1:1 run.



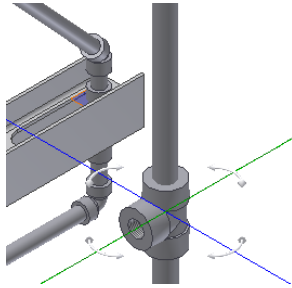
2 On the ribbon, click Pipe Run tab ► Content panel ► Place.

3 In the Place from Content Center dialog box, navigate to and double-click Tube & Pipe ► Fittings ► Tees ► ASME B16.11 Tee Threaded - Class 3000 to open the part family. Select a nominal diameter of 1/2 inch, and then place the tee on the downward pipe segment in the threaded steel route you previously created.

For detailed instructions on placing fittings from the Content Center using AutoDrop, see [Insert Library Parts Using AutoDrop](#) on page 132 in Chapter 6.

4 Use the 3D Orthogonal Route tool to rotate the tee to the orientation as shown in the following image. If the 3D Orthogonal tool is not displayed, right-click the tee, and select Edit Fitting Orientation.

5 Right-click and select Done.



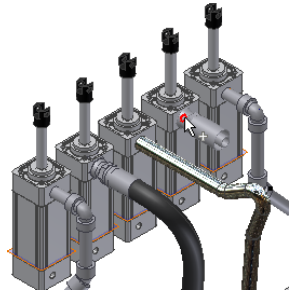
6 On the Pipe Run tab, Manage panel, select the Hydraulic Hose- Female Thread - Swivel (1/2 ND 1) style from the Active Style list.

7 On the ribbon, click Pipe Run tab ► Route panel ► New Route. Accept the default hose assembly file name and location in the Create Hose dialog box.

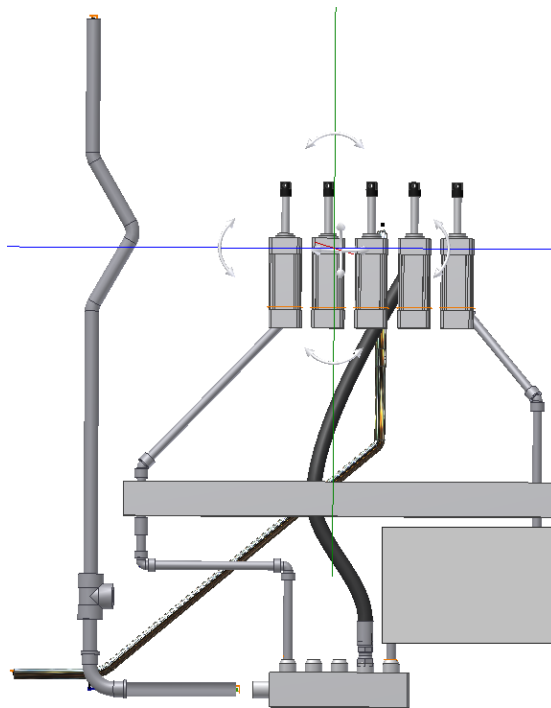
The Flexible Hose 02 subassembly is added to the AirSystem1:1 run. The Hose02 route environment is activated.

8 Click the Route tool.

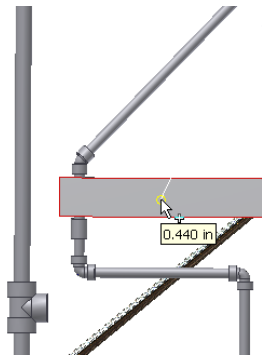
9 Select the start route point on the same geometry as shown in the following image.



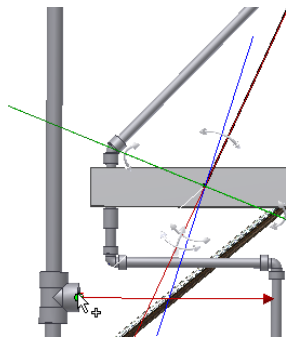
10 Rotate the assembly as shown in the following image.



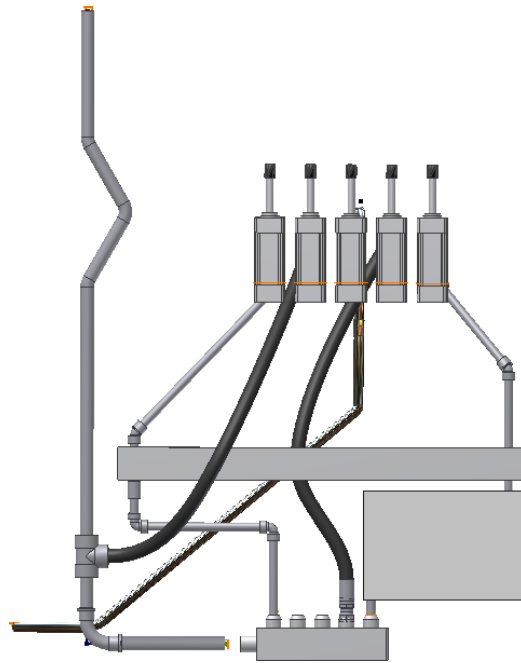
11 Pause the cursor over the planar face of the IBeam, right-click and select Edit Offset. The default offset distance is half the size of the outside diameter (OD) of the hose part, 0.440 inch for this style.



- 12 In the Edit Offset dialog box, enter 3 inches, and click OK.
- 13 Click the IBeam.
- 14 Select the tee to set the final node.



- 15 Right-click and select Done.
- 16 Right-click and select Finish Edit.
Look for the corresponding flexible hose subassembly under the AirSystem1:1 run.
- 17 Populate the hose route.
Your assembly should look like the following image.



Edit Flexible Hose Routes

You can edit both populated and unpopulated hose routes in several ways.

In this exercise, you use the hose subassembly you just created to:

- Edit hose nodes and hose length.
- Edit the active style to change the start fitting and end fitting.
- Change the diameters of all components in the hose route. For detailed instructions, see [Change Nominal Diameters](#) on page 96 in this chapter.
- Check the bend radius and clear violations.

Hose Nodes

After you finish editing a hose route and before it is populated, you can use the Insert Node tool to insert new hose nodes. The route recomputes with each new hose node.

You can also adjust the hose node position and orientation using the 3D Move/Rotate tool, redefine the hose nodes, and delete nodes.

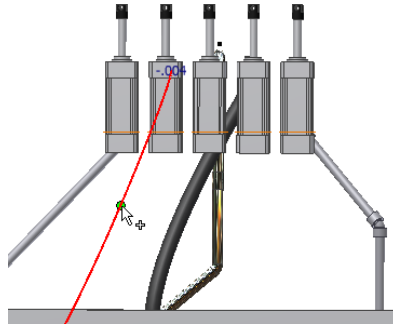
Add a hose node

- 1 In the AirSystem1:1 run, activate Hose02 in the Flexible Hose 02 subassembly.

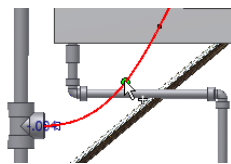


- 2 On the ribbon, click Route tab ► Create panel ► Insert Node, and then move the cursor over the hose spline.

The spline segment is highlighted with a green point indicating the intermediate node position is valid.



- 3 Click to insert the first node.
- 4 Right-click the spline below the IBeam, and select Insert Node to add another hose node.



- 5 Click to insert the second node.

Move a hose node

- 1 Right-click the new route point, and select 3D Move/Rotate.
- 2 Drag the triad in any direction. You can also enter a precise value along the X, Y, or Z axes.
- 3 Click Apply or OK.

NOTE If you want to move the hose node that is tangent to the circular edge of the IBeam, you must right-click the node and clear the Associative check mark.

Redefine a hose node

- 1 To change the position of the new node, right-click the route point and select Redefine.

When you move the cursor over planar surfaces or existing work geometry, the Edit Offset tool appears.

- 2 Move the cursor to the planar face of the I-Beam.
The offset guide indicates the offset value. By default it is the last selected offset value or the system default value of 0.440 in for this style.
- 3 Do either of the following:
 - To use the default offset value, click to set the node.
 - To change the offset value, pause the cursor at an appropriate point, right-click and select Edit Offset, and then enter a precise value.

The route is recomputed.

- 4 Right-click and select Done.

Delete a hose node

- 1 To delete the new route point that you just redefined, right-click the node and select Delete.
- 2 Repeat to delete the other inserted route point.
- 3 After deleting both route points, save the top-level assembly file.

NOTE You can delete the start or end hose node only when the flexible hose style suppresses both fittings or the end fitting respectively. Deleting such hose nodes enables you to redefine hose nodes using the Route tool.

Hose Length

When the hose is too slack or too tight, you can use the 3D Move/Rotate tool to adjust specific nodes and change the hose length closer to the desired length.

You can modify the length of a hose route precisely using the Hose Length tool. When adjusting the hose length, there are two types of length values displayed: Actual Length and Rounded Length. You can choose which one to display when documenting hose routes.

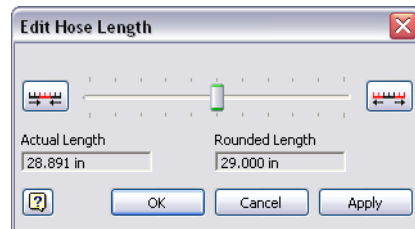
Adjust the length of a hose route

- 1 In the AirSystem1:1 run, activate Hose02 in the Flexible Hose 02 subassembly.



- 2 On the ribbon, click Route tab ► Manage panel ► Hose Length.

The Edit Hose Length dialog box displays the actual length and the rounded length. The rounded length is calculated from the hose round up value specified in the style. The values specific to your own exercise may be a bit different since positions of hose nodes may be different.



- 3 Drag the slider to the left and right to see the effects of the movement on the spline.
A thinner green preview spline represents the hose that results.

- 4 Click the Decrease the Scale tool or the Increase the Scale tool to change the length range.
- 5 Once you are satisfied with the hose length, click OK.

Start Fitting and End Fitting

Use the style to modify fittings in flexible hose routes. You can change existing fittings, remove a fitting, or restore a suppressed fitting.

In this exercise, the hose style in use has specified the start fitting only and suppressed the end fitting. You must do the following:

- Change the start fitting.
- Add a new end fitting.
- Suppress the newly added end fitting.

NOTE If you want to remove both start fitting and end fitting, you can choose to suppress the start fitting and the end fitting is suppressed automatically.

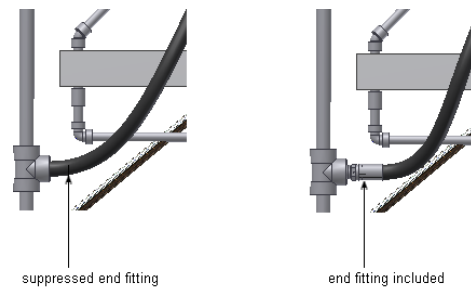
Replace the start fitting

- 1 In the AirSystem1:1 run, activate Hose02 in the Flexible Hose 02 subassembly.
- 2 On the ribbon, click Route tab ► Manage panel ► Tube and Pipe Styles.
- 3 Under Components, look at the Start Fitting used in the style.
- 4 Click Edit.
- 5 Right-click the Start Fitting row and select Browse to access the Content Center. The Library Browser filters out all the hose family and fittings in compatible standards.
- 6 Select Parker Male Taper Thread - Swivel from the list.
- 7 Click OK.
- 8 In the Tube & Pipe Styles dialog box, click Save, and then click Close.
- 9 Right-click and select Finish Edit.
The start fitting in the graphics window and in the Model browser is changed.

Add and suppress an end fitting

- 1 Open the Tube & Pipe Styles dialog box and ensure the hose route style to receive the new end fitting, Hydraulic Hose- Female Thread - Swivel (1/2 ND 1) is the active style.
- 2 Click Edit.
- 3 Under Components, right-click the End Fitting row and clear the Suppress Fitting check mark.
- 4 Click Save, and then click Close.

A new end fitting for the hose route connects the hose segment and the tee. You can verify this in the graphics window and Model browser.



Bend Radius Check

The Flexible Hose style determines the minimum bend radius. Before populating, use the Bend Radius Check tool to verify a single hose route or all hose routes under the pipe run. If any bend radius in a hose route is smaller than the minimum value or the route is self-intersecting, a yellow error icon is displayed after bend radius check.

When violations are corrected, check the bend radius again or click Return on the Quick Access toolbar to update the hose route. In this case, the error icon automatically disappears.

NOTE You can use the Clear Violation tool to remove the error icon from the Model browser at any time, but violations still exist in the hose route until you manually correct them.

Check the bend radius

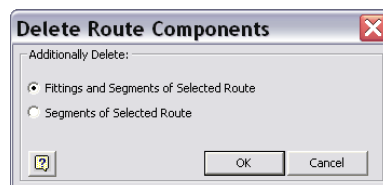
- 1 Activate the hose route.
- 2 Right-click the hose route in the browser, and select Bend Radius ► Check.
- 3 If the Error dialog box displays the violations against the minimum bend radius, click OK.
Optionally, right-click the hose route or the pipe run in the browser, and select Bend Radius ► Clear Violation.
- 4 Edit the hose route.
- 5 Repeat Steps 2 through 4 until no violations occur.

Delete Flexible Hose Routes

You can delete both populated and unpopulated hose routes. When deleting, you are prompted to indicate whether to delete all segments and fittings in the route or only the segments. If only segments are deleted, the fittings are moved to the top of the browser and are available for future use. For this exercise, it is recommended that you delete both the hose segments and the fittings.

Delete a hose route

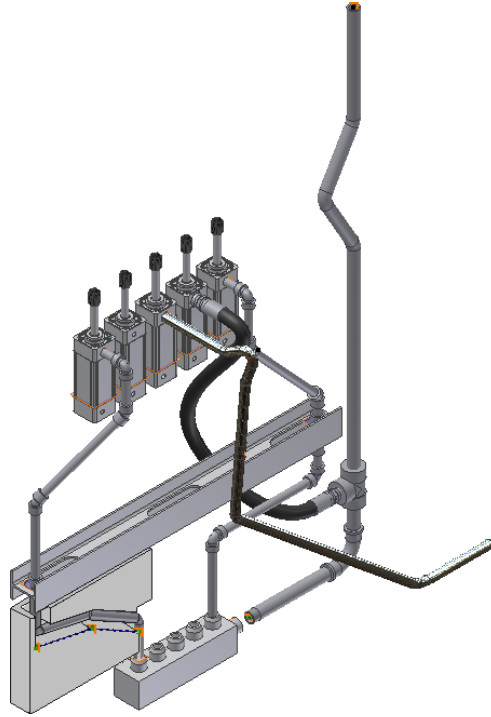
- 1 Activate the AirSystem2:1 run.
- 2 If the Flexible Hose 01 subassembly is collapsed, click to expand it.
- 3 Right-click Hose1 and select Delete Route.
- 4 In the Delete Route Components dialog box, make sure Fittings and Segments of Selected Route is selected, and then click OK.



The Flexible Hose subassembly is deleted completely.

- 5 Save the top-level assembly. This run becomes empty.

Your assembly should look like the following image.

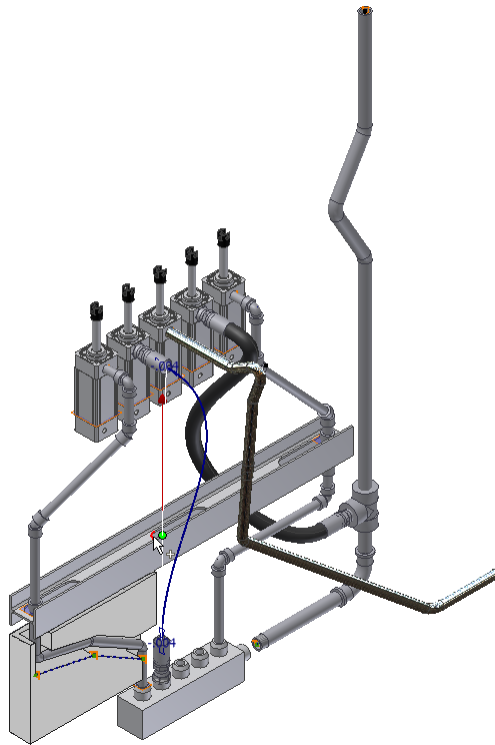


Practice Your Skills

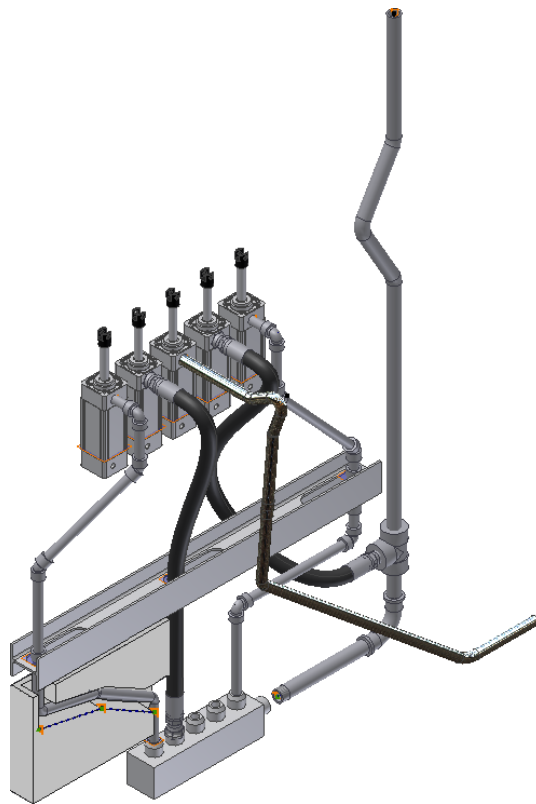
Using the skills you have learned from the previous exercises, try the following in the AirSystem2:1 run:

- 1 Activate the AirSystem2:1 run.
- 2 Create a new hose style with a flat structure that is based on the Hydraulic Hose - Female Thread - Swivel style with the following settings:
New style name: Flat Hose with Female Swivel (1/2 ND 2)
Diameter: 1/2 inch
Clear the Use subassembly check mark
- 3 Create a route using the new style. Note the flexible hose and hose components in the browser.

- 4 On the ribbon, click Pipe Run tab ► Route panel ► New Route.
The Create Route dialog box displays. Note that it is not the Create Hose dialog as displayed in the preceding exercises.
- 5 Accept the default hose route file name and location and click OK.
A new route node is added to the run in the Model browser. Note that there is no flexible hose assembly node.
- 6 Create a new hose route using the new style. To define the route, select the start, end, and then intermediate geometry as shown in the following image.



- 7 Right-click in the graphics window and select Finish Edit.
- 8 Populate the hose route and save the top assembly.



Edit Rigid Routes and Runs

6

Autodesk Inventor® Routed Systems provides varied methods for updating and modifying each component of a tube and pipe system to accommodate changes in design criteria and in the assembly model.

In this chapter, you learn how to use various tab and context menu options to edit routes and runs.

About Editing Rigid Routes and Runs

After finishing the initial definition of a route or run, you can continue to make changes. Available edit tools may vary depending on the edit context, such as auto regions and parametric regions.

Using Undo reverses the last action taken during the current editing session, and exits the current command.

NOTE Consider the affect of changes to assembly model geometry on which the routes and runs depend. For example, deleting parts or editing geometry in the model that affects associative route points may require you to delete the run and redefine the route.

Modifying or changing the style is another way of modifying routes and runs. For detailed instructions, see [Change Styles for Existing Routes](#) on page 49 in Chapter 3.

Auto Regions

The following edit options are available for auto regions:

- Add interior route points.

- Insert, replace, delete, and restore placed fittings.
- Change the active tube and pipe style.
- Change fitting diameters.
- Use the Edit Position or Move Segment tool to reposition route segments.
- Use the Edit Position or Move Node tool to reposition the intermediate route points that are associated with a default coupling.
- Switch to an alternative solution if multiple solutions exist.
- Convert an auto region to a series of continuous sketched route segments.
- Delete auto regions.
- Remove client constraints to convert a segment or point in an autoroute region to a parametric region.

Parametric Regions

The following edit options are available for parametric regions:

- Insert, reposition, or delete segments and route points.
- Insert, replace, delete, and restore placed fittings.
- Change the active route style.
- Change fitting diameters.
- Reposition a placed fitting by editing segment dimensions or using the 3D Move/Rotate tool.
- Use the 3D Move/Rotate tool to move route points in three dimensions.
- Create bends using the Bend tool.
- Edit the dimensions (linear, angular, and radial) using the General Dimension tool.
- Switch between driven dimension and normal sketched dimension.
- Show, edit, and delete geometric constraints.
- Delete route segments. It is also the way of deleting the free terminal route point.

Options for Editing

You can edit both populated and unpopulated routes. If the route is populated, activate the route to edit it in place. Library components populating the route are temporarily set as not visible and the underlying 3D sketch of the route is displayed.

Route Tab

When you are in the route environment, the Route tab is displayed. Appropriate tools are available in the specific edit context. Pause the cursor over the images to view the tooltip.



Pipe Run Tab

When you are in the run environment, the Tube & Pipe tab is displayed. You can place fittings from the Content Center or your project workspace, and connect fittings and components.

In the exercises that follow, you learn the basic Connect fitting tools:

- | | |
|-------------------------|--|
| Connect Fittings | Connects two components relative to one another in a tube and pipe assembly. The components need not be collinear. You can connect a fitting, a conduit part, or a normal Autodesk Inventor® part that already exists in the assembly to another base component. You can also connect fittings when placing or dropping them in the graphics window. |
| Insert Fittings | Inserts and connects a new fitting between two connected fittings in a tube and pipe assembly. The new fitting must be collinear to make the connection. Using this feature you can build a series of connected fittings, or insert fittings between fitting groups such as flanges created during population of a route. |

Context Menus

Many edit actions start by selecting from the context menu. The context menu varies depending on the edit context and the selected item.

In the exercises that follow, you learn the following basic editing tools:

3D Move/Rotate Interactively positions route points in the parametric regions by dragging the triad in a planar mode, axial move or rotate, free movement, or entering values along X, Y, or Z axes. You can also reposition route points by editing the segment dimensions.

By default, the triad axes relate logically to the world coordinate system but may vary from the route direction. To accurately define the X, Y, Z distances relative to the route direction, you can redefine the axis alignment to the route segment.

Change Fitting Diameter Changes the diameter for one or more placed fittings and routed fittings that are used by a style to populate routes and runs. The system finds the diameters that are common to all selected fittings and displays them in a list.

NOTE Change Size performs the same action.

Delete	<ul style="list-style-type: none"> ■ For route segments in parametric regions, deletes the selected route segment. ■ For pipe or tube parts, deletes the selected conduit part and the underlying route segment. ■ For route points, deletes the selected node from the route and re-evaluates automatic routing against the style settings, and then updates the route. You can delete manually placed route points, but not start and end points and route points that are inserted during automatic routing. ■ For client constraints on the auto region, deletes the client constraint so that you can then edit individual route points and segments. ■ For groups of fittings, when removing the base fitting that owns the route point, you must indicate how to handle the remaining fittings in the group.
Delete Route	Removes the route, plus fittings and segments, or segments only in the selected route.
Delete Run	Removes the selected pipe run from the browser and the graphics window. All routes and components in the pipe run are deleted.
Display/Update Settings	Prevents or allows automatic populating of tube and pipe components in an individual route, pipe run, or master runs assembly by selecting Route Objects Only or All Tube & Pipe Objects.
Edit Fitting Connections	Deletes or edits the engagement for fitting connections between two fittings or between a tube, pipe, or hose part or a standard Inventor part and a fitting that was manually inserted from the Content Center Library using the Insert tool or from the active project workspace using the Place Fitting tool.
Edit Fitting Orientation	Activates the selected fitting and provides the 3D Orthogonal Route tool for rotational changes. It is available only for placed fittings.
Edit Position	Activates the selected auto route segment or route point that is associated with a default coupling in an auto region, and provides the 3D Orthogonal Route Tool for translational edits.

Fitting	Turns the appearance of a fitting (coupling) on and off for route points associated to geometry. When cleared, the fitting is not created, the pipe remains a single segment, and the route points remain associative to the selected geometry.
Hose Length	Edits the hose and adjusts the length by changing the weight of the tangency or tension of the hose segment.
Insert Node	Inserts a route point in the selected route. By default, this new node adds a coupling, weld gap, or flange at the selected route location. Available on the Route tab or in the context menu of the graphics window when a route is activated.
Move Node	<p>Moves a route point that is associated with a default coupling in an auto region to a new position. The system recalculates the automatic routing based on the new position of the selected route point.</p> <p>Note that such route points are generated when the linear distance between two points is larger than the maximum segment length that you have specified in the rigid piping or bent tubing styles.</p> <p>To reposition route points in parametric regions, you can edit the segment dimensions or apply the 3D Move Rotate tool</p>
Move Segment	Moves a route segment in an auto region to a new position. The system recalculates the automatic routing based on the new position of the selected segment.
Restore Fitting	Replaces a placed fitting with the default fitting (elbow or coupling) for the set style or replaces a placed fitting with a coupling no matter how many connection points the original placed fitting has.
Route	Continues adding route points to the specified route. Also available on the context menu for free terminal route points that are not used in any other routes or fittings.

For more information about commands on the context menu, see the Help index.

Control Individual Settings

When changes are made to a standard Autodesk Inventor assembly, routes and runs, and positional representations, the tube and pipe assembly defaults to automatically update. To save time for updating the whole tube and pipe assembly, you can defer automatic updates of tube, pipe, and hose components for individual runs and routes. You can see the effect a change has on an individual route before repopulating.

You can also disable automatic updates for the entire tube and pipe runs subassembly. The following image shows the Display/Update Settings list you can access on the Pipe Run tab when a pipe run is activated.



- | | |
|------------------------------------|--|
| All Tube & Pipe Objects | Allows the entire pipe run or specified routes to fully update. When the All Tube & Pipe Objects option is selected, the associated routes and runs are displayed as populated and automatically respond to changes. This is the default setting when new routes and runs are added to a master runs assembly. |
| Route Objects Only | Defers automatic updates for the entire pipe run or specified routes. When the Route Objects Only option is selected, the associated routes and runs are displayed as centerlines, underlying components are hidden and do not respond to changes. |

NOTE When a new route is created and populated, it always displays as populated and not as centerlines. This is true even when the pipe run is set to Route Object Only.

Control the display and update setting for individual runs and routes

- 1 In the *AirSystemAssy.iam* assembly, activate the top assembly or master runs assembly.

- 2 Right-click Tube & Pipe Runs, and then select Tube & Pipe Settings. Ensure the Defer All Tube & Pipe Updates check box is cleared in the Tube & Pipe Settings dialog box. These settings are only available when the check box is cleared.
- 3 Activate the pipe run.
- 4 Right-click the pipe run or a specific route, and then select Display/Update Settings, or click the pipe run or a specific route, and then select the Display/Update Settings tool on the tab.
- 5 Select the display and update setting you need.
- 6 Use the default setting, All Tube & Pipe Objects, so that you can view automatic responses to edits on routes and runs.

Route Points

A route is considered finished when you select Finish Edit. You can also use the Route tool to repair disjointed routes.

A disjoint route can occur when segments are deleted from a route. While deleting segments, you can use the Route tool to close the gap between disjoint segments.

Add to Finished Routes

New route points can be added to a previously finished route by activating the route to edit, and then using the Route tool on the end route point.

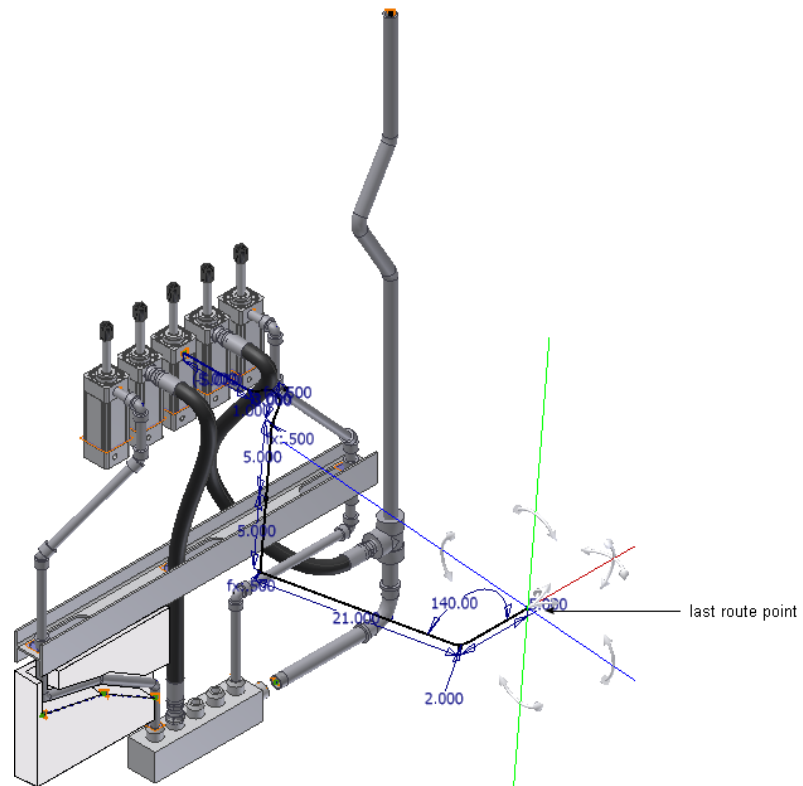
Add a point to an existing route

- 1 Activate Route03 in the AirSystem1:1 run.



- 2 In the graphics window or Model browser, do either of the following:
 - Click the last route point, and select the Route tool on the Route tab.
 - Right-click the last route point, and select the Route tool.

The route resumes at the last selected route point.



3 Add one more point to the route in a location and orientation of your choice, for instance, 8 inches along the red axis.

4 Right-click and select Done.

5 Right-click and select Finish Edit.

The new route point becomes the current last route point. A coupling is added to the route point from which you continued the routing.

Insert Intermediate Route Points

Intermediate route points can be added to any route or run as long as they comply with styles.

Insert intermediate route points

- 1 Activate a tube or pipe route.



- 2 Click Route tab ► Create panel ► Insert Node.
- 3 Click the segment to insert a route point.
- 4 Right-click and select Finish Edit.
A coupling is added to the route in that location.

Delete Route Points

You can delete the manually created route points when the resultant route complies with the style criteria. Tube and Pipe automatically closes Gaps left by deleted route points between collinear segments are automatically closed. Segments adjoining the deleted route points reposition or resize to adjust to the change.

To delete the free terminal route point, you must delete the last segment.

Delete manually created route points

- 1 Activate Route03 in the AirSystem1:1 run.
- 2 In the graphics window or Model browser, right-click the intermediate route point you just inserted manually, and then select Delete.
The last two segments automatically heal to one single segment.
- 3 Right-click and select Finish Edit.

NOTE After the manually inserted route point populates as a coupling fitting, delete the fitting in the run environment and the underlying route point.

Auto Regions

Intermediate segments in an auto region can be repositioned dynamically by using the Move Segment tool and clicking and dragging or by using the precise distances. The movement allowed follows the conditions established by the

current style and connection data. It also depends on the geometry adjacent to the point you select for the operation.

As you drag the selected geometry, the system re-evaluates and updates adjacent route components. Route points that are automatically generated between points on selected geometry update when changes are made to a route.

Move Auto Route Segments Approximately

Using the Move Segment tool on the Route tab, you can move segments in an auto region approximately. The direction arrows appear on the geometry indicating the directions allowed for the move. Click anywhere on the segment near the direction arrow you need, or click and drag the arrow directly. The arrow changes to red indicating that it is the direction being dragged.

If the minimum length setting for the style is violated during a drag, the segment or segments in violation turn red. A tooltip also displays the segment length and the message <Min Pipe Length> in red text.

If alternate solutions are available, the Select Other tool appears for you to cycle through and select the solution you need.

Move segments in an auto region approximately

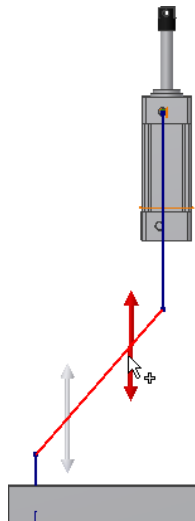
- 1 Activate Route01 in the AirSystem1:1 run.



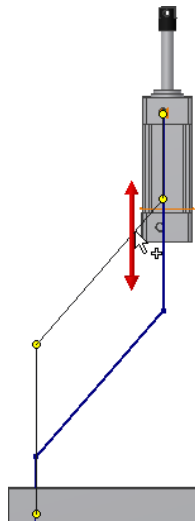
- 2 On the ribbon, click Route tab ► Create panel ► Move Segment.

- 3 Pause the cursor over the right end of the segment as shown in the following image, until the arrow changes to red.

The movement allowed depends on the geometry closest to the point you select for the drag operation. If the point is a coupling or an associated route point, the movement is restricted.



- 4 Click and drag the segment upward as shown in the following image. To end the drag operation, release the cursor.



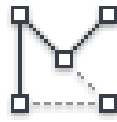
- 5 Right-click and select Done.
- 6 Right-click and select Finish Edit.

Move Auto Route Segments Accurately

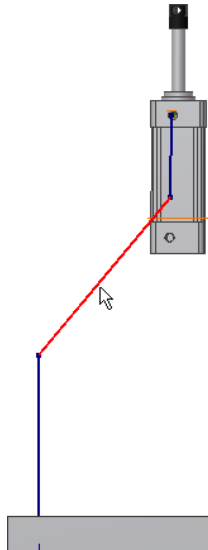
By entering precise values, the Edit Position tool not only applies to the route points that are associated with a default coupling fitting in an auto region, but can also move the intermediate segments in an auto region.

Move segments in an auto region accurately

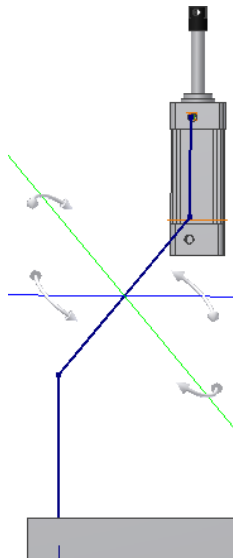
- 1 Activate Route01 in the AirSystem1:1 run.



- 2 In the graphics window, right-click the segment as shown in the following image and select the Edit Position tool.



The 3D Orthogonal Route tool is displayed with available direction to reposition the segment.

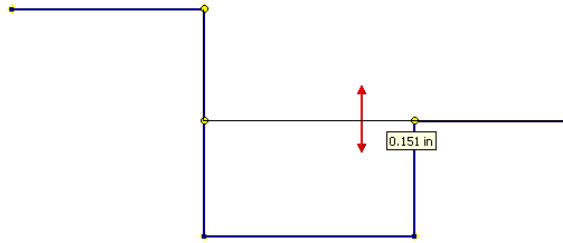


- 3 Pause the cursor over the direction axis, and then enter a specific value.
- 4 If alternative solutions are available, the Select Other tool appears for you to cycle through and select the solution you need.
- 5 Use Undo to reverse the last action and exit from the current command.
- 6 Right-click and select Done.
- 7 Right-click and select Finish Edit.

Remove Unwanted Segments or Route Points

In an auto region, when adjacent segments are coplanar and perpendicular to each other, you can use the Move Segment tool to remove the unwanted segment and the route point involved.

To remove them, drag the geometry until the route points are coincident with adjacent route points.



Release the cursor to stop the drag. The unwanted segment is removed.



You can then delete the extra route points using Delete from the context menu, or by dragging the unwanted route points until they are coincident with an adjacent one. When they are coincident, release the cursor to end the drag and remove the route point.

While you are dragging the segment, use the ESC key to cancel the drag and return to regular editing.

NOTE Ensure that the segment being moved does not attempt to overlap any other existing segment. Otherwise, the system will automatically calculate a route shape.

Convert Auto Region to Parametric Sketch

If you need to have more control over the segments in an auto region, use the Convert to Sketch tool to convert it to a series of continuous sketched segments. The Model browser reflects the deletion of the auto region and new sketched route points.

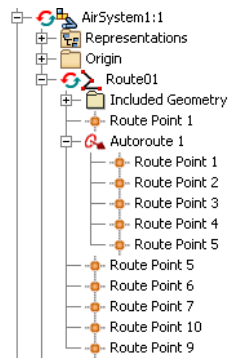
All client constraints in the auto region are then deleted so more edit options are available to adjust the route. It is equivalent to deleting all client constraints within the auto region manually.

TIP If you need to convert only one segment from within an auto region, right-click the segment, select Show All Constraints, select the client constraint for the segment, and right-click and delete the client constraint symbol. If you want to break an auto region at one route point, you can choose to delete the client constraint symbol for the route point.

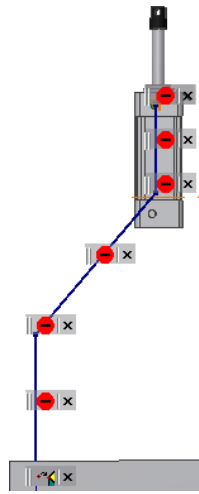
In this exercise, before you convert the auto region in the AirSystemAssy.Route1:1 you should route to the parametric region.

Convert an auto region to parametric sketch

- 1 Activate Route01 in the AirSystem1:1 run.
- 2 In the Model browser, expand the Autoroute 1 node and review the nodes in the Model browser. You can then compare the browser behavior with the new parametric region.



- 3 Right-click in the graphics window and select Show All Constraints. Review the client constraints on the segments in the auto region.

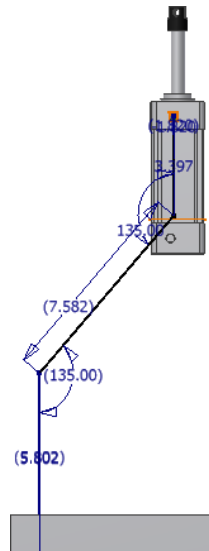


- 4 In the Model browser, right-click the Autoroute 1 node and select Convert to Sketch.

The auto region changes to a series of continuous sketched segments. Dimensions are added to the route sketch where appropriate.

NOTE If Auto-Dimension is not enabled during route creation, dimensions will not be added here.

- 5 Press F8 to verify that all client constraints are deleted so edit options for parametric regions are available to adjust the route points and segments. To hide all constraints, press F9.



- 6 In the Model browser, verify that the Autoroute1 node disappears.
- 7 Right-click and select Finish Edit.

Dimensions

Auto regions always dynamically update to assembly changes and the shape of the route may vary from the original auto route solution. They do not involve dimensions until you convert them to parametric regions.

For parametric regions, there are three typical types of dimensions pertaining to the route sketch:

- Linear dimension, such as the segment length.
- Radial dimension, such as the bend radius.
- Angular dimension, such as the included angle at the direction turn.

Similar to Autodesk Inventor, dimensions on the route sketch can be categorized into two types:

Normal sketched dimension (driving dimension) It is used to drive the route geometry. For instance, sketched route segments are manually created using a specified normal sketched dimension.

Driven dimension It is enclosed in parentheses as displayed in the graphics window and allows route geometry to dynamically respond to associated changes. It typically occurs on the route sketch that relates to the assembly geometry or when an auto region is converted to a parametric region.

You can switch between the driven dimension and the normal sketched dimension using the Drive Dimension tool. Once a driven dimension is switched to a normal sketched dimension, you can edit the value. If converting the driven dimension to a driving dimension will over-constrain the geometry, the conversion is not allowed.



Driven Dimension tool

In the route environment, normal sketched dimensions can be manually set using the General Dimension or edits in place. If you want to place dimensions based on geometry outside of the active route, you must first include it in the route sketch using the Include Geometry tool.



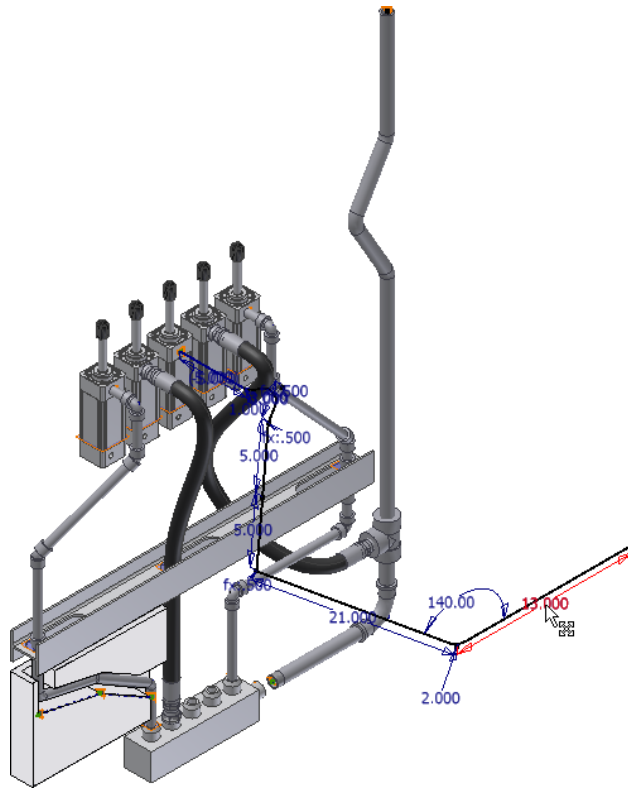
General Dimension tool

NOTE Placing dimensions may over-constrain the route sketch. You may also fail to switch the dimension type. You can identify the geometric constraints as needed.

In this exercise, you edit the segment length in place. For more information about how to place and edit dimensions on the route sketch (3D sketch), refer to the Help.

Edit the route dimension

- 1 Activate Route03 in the AirSystem1:1 run.
- 2 Double-click the dimension on the last segment as shown in the following image.



- 3 On the Edit Dimension dialog box, enter 5 inches.
- 4 Right-click and select Finish Edit.

Fittings

You can place fittings from either the Content Center or the active project work space. They can be placed onto tube and pipe route segments, free terminal route points, adjacent to existing fittings, between connected fittings, or in the background of the graphics window.

Segment length must comply with the minimum segment length style once the fitting is placed. If the fitting is not compatible with the route in some other way, such as size and material differences, the insertion is allowed, but you are alerted to the incompatible conditions. The fitting being dropped

must also be appropriate for the drop point selected. For example, elbows may not be dropped on straight segments.

For a placed fitting or a routed fitting for which you have changed the fitting diameter, you can:

- Change its orientation, as well as connection point.
- Change the fitting route point position on straight rigid segments.
- Replace existing fittings.
- Restore original fittings, typically elbows and couplings.
- Delete placed fittings and default couplings. You cannot delete the default directional fittings.

Place Fittings in the Active Project Workspace

Fittings from the active project workspace are added to the design using the Place Fitting tool on the Pipe Run tab.

NOTE You cannot place conduit parts using this tool.

Place a fitting from your project workspace

- 1 Activate the pipe run to receive the fitting.



- 2 On the ribbon, click Pipe Run tab ► Route panel ► Place Fitting.
- 3 In the Open dialog box, browse to and select the component to place, and then click Open.
The selected component is placed in the graphics window, attached to the cursor.
- 4 Click a point of your choice for insertion.
 - To insert an elbow, select a direction change route point.
 - To insert other types of components, select any point along an existing run segment.

- 5 To place additional occurrences of the same part, move the cursor to a different location and click. Continue until all occurrences are placed.
- 6 Right-click and select Done.
- 7 Right-click and select Finish Edit.

TIP For consumed fittings in a tube and pipe run assembly, click to highlight the fitting in the Model browser or the graphics window, and then click the Place Fitting tool to place more occurrences of the highlighted fitting quickly. Using the Model browser, you can insert any compatible fitting from another pipe run into the target pipe run. Using the graphics window, you can only select fittings within the target pipe run.

Insert Library Parts Using AutoDrop

Both library fittings and conduit parts can be placed into a tube and pipe assembly from the Content Center using AutoDrop.

Conduit parts can be placed only in the background of the graphics window. You can connect the placed conduit parts to other components using the Connect Fittings tool. Hose routes do not accommodate placed fittings.

To locate tube and pipe library parts, start the Place from Content Center or Replace from Content Center tool.

In this exercise, you activate a run, locate a standard fitting, select the nominal diameter, and then use AutoDrop to insert it into the run.

For more information about the Content Center libraries, see [Use Content Center Libraries](#) on page 149.

NOTE If you have changes to the default parameters for the library part and it is the first time for AutoDrop, you must save it as a custom part before completing the placement. This typically occurs when placing conduit parts.

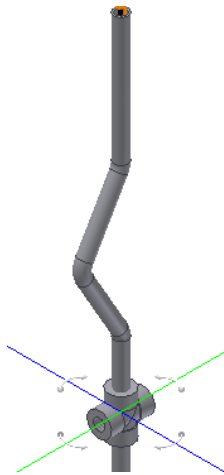
Insert a library fitting using AutoDrop

- 1 Activate the AirSystem1:1 run.



- 2 On the ribbon, click Pipe Run tab ► Content panel ► Place. You can also right-click in the graphics window to access the tool.

- 3 In the Place from Content Center dialog box, go to Tube & Pipe ► Fittings ► Crosses.
- 4 In the Crosses pane, click ASME B16.11 Cross Threaded - Class 3000. All family members are listed in the pane below.
- 5 Click OK to select the part family and close the Place from Content Center dialog box.
The ASME B16.11 Cross Threaded - Class 3000 Part Family dialog box is displayed with a list of all available nominal diameters.
- 6 On the Table tab, select:
ND: 1/2
As Standard
To verify the detailed information of the selected cross family, click and go through the Select, Table View, and Family Information tabs.
- 7 Click OK to complete the part member selection.
The cross is attached to the cursor in the graphics window.
- 8 Move the cursor over the pipe segment, and then click to set the cross fitting.



- 9 Optionally, use the 3D Orthogonal Route tool to edit the orientation of the placed cross.
- 10 Right-click and select Done.

- 11 When quitting the AutoDrop, you can right-click the placed fitting and select Edit Fitting Orientation to display the 3D Orthogonal Route tool again.
- 12 Right-click and select Finish Edit.

Practice Your Skills

In this exercise, you use the skills you just learned to insert a pipe part from the Content Center Library into the background of the graphics window. Note that you cannot drop the conduit part onto route segments.

In addition to ND (Nominal Diameter), you must specify the SN (Schedule Number) and PL (Pipe Length) parameters.

Insert a library pipe part

- 1 Activate the AirSystem1:1 run.
- 2 Right-click in the graphics window and select the Place from Content Center tool.
- 3 In the Place from Content Center dialog box, navigate to and select Tube & Pipe ► Conduits ► Pipes ► ASTM A 53/A 53M Pipe.
- 4 In the Content Center Part Family dialog box, specify:
ND: 1/4
SN: 40
PL: 10
- 5 Click OK.
- 6 In the Save As dialog box, use the default location and file name.
- 7 Place the pipe part anywhere in the background of the graphics window.
- 8 Right-click and select Done.

TIP You can use the Connect Fittings tool to connect the placed tube or pipe part to another component in the run.

Adjust Fitting Position and Orientation

Fittings are associated with the underlying route points on the route sketch, with the exception groups of connected fittings. In this case, the first fitting placed is associated with an underlying route point and all adjacent fittings are associated to that first fitting. Repositioning the route point simultaneously changes the position of the fitting or fitting group. You can activate the route environment, and then reposition the route point using the 3D Move/Rotate or General Dimension tool and reposition the default coupling fitting using the Edit Position or Move Node tool.

You can also use the Edit Fitting Orientation tool to redefine the fitting orientation and change the connection point.

In this exercise, you change the orientation and connection points on the fitting you placed in the preceding exercise.

Change fitting orientation and connection point

- 1 Activate the AirSystem1:1 run.
- 2 In the graphics window, right-click the cross, and then select Edit Fitting Orientation.
The 3D Orthogonal Route tool is displayed at the fitting.
- 3 Use the rotation axes to reorient the fitting.
 - For exact rotation, right-click the rotation arrow, and then select Enter Angle.
 - For exact position, right-click on the direction axis parallel to the segment, and then select Enter Distance.
- 4 Optionally, to change the connection point, right-click in the graphics window, and then use Select Orientation to set the new connection point.
- 5 Right-click and select Done.
- 6 Right-click and select Finish Edit.

Restore Default Fittings

Using the Restore Fitting tool, you can restore the placed fitting to a coupling or an elbow no matter how many connection points the placed fitting has.

The placed fitting to restore must be within the tube or pipe segment or it cannot be placed at the free terminal of the run.

Restore a default fitting

- 1 Activate the AirSystem1:1 run.
- 2 Right-click the placed cross and select Restore Fitting.
The placed fitting changes to a coupling fitting at the same position.



Replace Existing Fittings

You can replace one occurrence of a selected fitting instance with a new fitting instance from the Content Center or all occurrences at once. The replacement fittings can be inserted at previously defined route points.

Replaced fittings use the route information of the previous part. Orientation and position can be adjusted if the part type and connection information allows.

In this exercise, you place a tee from the Content Center to replace the coupling you just restored from the placed cross. In addition, you can also place a non-library replacement fitting using the Place Fitting tool.

Replace existing fittings with library fittings

- 1 Activate the AirSystem1:1 run.



- 2 In the Model browser or graphics window, click the coupling, and then click Pipe Run tab ► Content panel ► Place.
The Replace from Content Center dialog box is displayed.
- 3 Navigate to Tube & Pipe ► Fittings ► Tee ► ASME B16.11 Tee Threaded - Class 3000 and double-click to open the part family.
The ASME B16.11 Tee Threaded - Class 3000 Part Family dialog box is displayed.
- 4 Optionally, to replace all occurrences in the highlighted fitting type, select the Replace All check mark.
In this exercise, you only have one occurrence of the cross fitting so you see no difference.
- 5 In the dialog box, select:
ND: 1/2
As Standard
Click OK.
The cross fitting is replaced with the tee fitting.
- 6 Optionally, to edit the orientation of the placed tee, right-click the tee and select Edit Fitting Orientation. To end the edit, right-click and select Done.



- 7 Right-click and select Finish Edit.

Connections

Fitting connections are used to maintain the fixed relationship between components in a tube and pipe assembly. You can connect fittings or normal Autodesk Inventor components that already exist in the tube and pipe assembly to other components including connected fittings. You can also connect fittings being dropped from the Content Center or being placed using the Place Fitting tool to another component. After the connection is created, use the Edit Fitting Orientation tool to rotate the free fitting or component to a new orientation. Select multiple fittings to rotate them as a group.

You can break connections to edit the fitting connections independently and accommodate more changes to the tube and pipe run assembly. You cannot break connections between fittings placed during populate route.

In this exercise, you delete the connection between the tee fitting and the hose route, and then adjust the hose route and connect to another tee fitting you placed on the upper segment of Route03 in the AirSystem1:1 run.

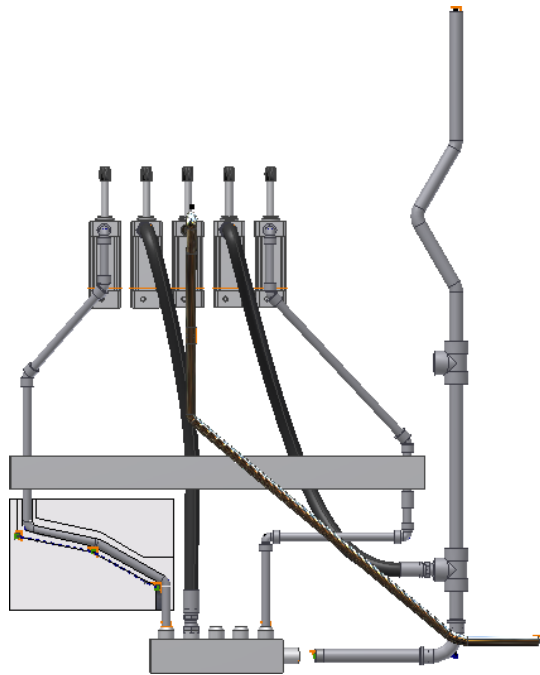
Delete Fitting Connections

You can delete fitting connections and edit the engagement when they are formed:

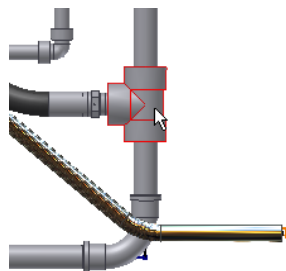
- By using the Connect Fittings tool.
- By dropping a fitting on the end of a pipe segment or adjacent to other fittings.

Delete fitting connections

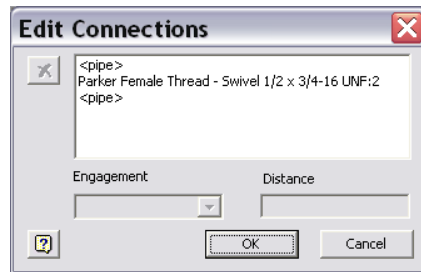
- 1 Activate the AirSystem1:1 run.
- 2 Rotate the graphics window to get a better view.




- 3 Right-click the tee fitting as shown in the following image, and select Edit Fitting Connections.



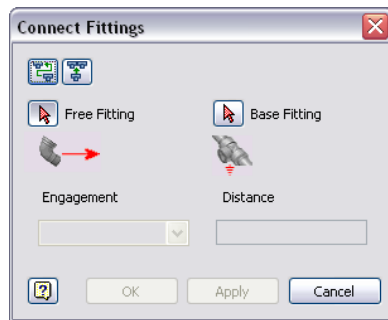
The Edit Connections dialog box displays with all existing connections on the tee fitting, including two pipe segments and one hose fitting.



- 4  In the connections list, select Parker Female Thread - Swivel 1/2 x 3/4-16 UNF:2, and click the Delete button.
The connection with the hose fitting changes to no connection.
- 5 Click OK.

Connect Fittings and Components

Use the Connect Fittings tool to connect two components relative to one another in a tube and pipe assembly. You can connect a fitting, a conduit part, or a normal Autodesk Inventor part that already exists in the assembly to another base component. You can also connect fittings before placing them in the graphics window.



You can also insert and connect a collinear fitting between existing fittings. The dialog box changes depending on the operation you have selected.



To complete the connection, you must have one free fitting and a base fitting or fittings as specified in the following images. You also specify the engagement type, as well as the engagement distance when User Defined is selected from the Engagement list.

- | | |
|------------------------|--|
| Free fitting | Can be a fitting or a normal Autodesk Inventor part in the active pipe run assembly that is not driven by a node in a route or by any other fitting. When inserting a fitting between an existing fitting connection, the fitting must be collinear. Once the free fitting is connected to the base component, it is always positioned relative to the base fitting during updates and other operations. |
| Base Fitting(s) | Can be a fitting, a conduit part, or a normal Inventor part that is located anywhere within the assembly. When inserting a fitting, the base fittings are restricted to connected collinear fittings. In both cases the base fitting specifies the connection that constrains the position and routing of the free fitting. |
| Engagement | Available only when you select a connection between two fittings or between a component and a fitting that was inserted using the Connect Fitting tool. It includes three engagement types: <ul style="list-style-type: none"> ■ Free Fitting: Uses the free fitting engagement type for the selected connection. ■ Base Fitting: Uses the base fitting engagement type for the selected connection. |

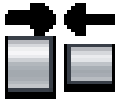
- **User Defined:** A positive number moves the free fitting or component away from the connected base fitting or component. A negative number moves the free fitting or component toward the connected base fitting or component.


Distance Indicates the engagement distance used for the connection between fittings and components. Read-write only when the User Defined engagement type is used.

In this exercise, you connect the hose route to the upper tee you manually placed. The hose route is flexible and acts as the free fitting.

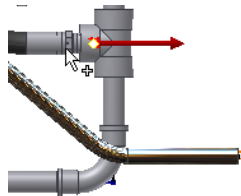
Connect fittings and components

- 1 Activate the AirSystem1:1 run.
- 2 Expand the Flexible Hose 02 assembly, and then activate the hose route.
- 3 In the graphics window, right-click the intermediate Route Point 2, and select Delete.
The hose route recalculates the hose length.
- 4 Right-click and select Finish Edit.
The AirSystem1:1 run is activated.

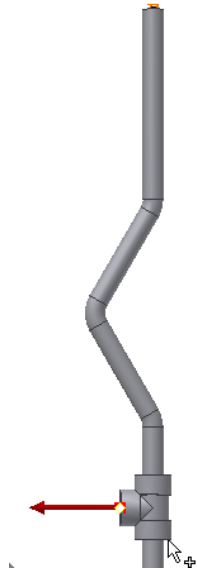
- 5  On the ribbon, click Pipe Run tab ► Route panel ► Connect Fittings.

- 6  In the Connect Fittings dialog box, ensure the Connect Fittings tool is enabled.

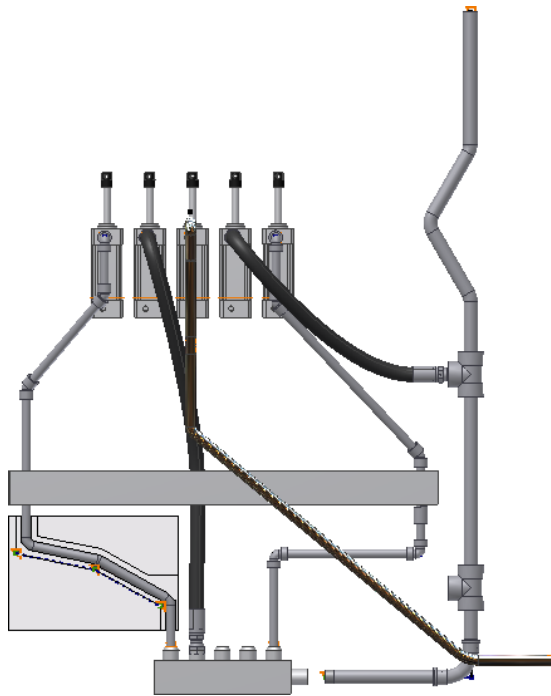
- 7 Ensure the Free Fitting tool is selected and click the hose fitting.



- 8 Click the Base Fitting tool, and then click the tee fitting on the upper segment.



- 9 Select Female Fitting as the engagement type.
- 10 Click Apply or OK.
The hose route is recomputed. Right-click in the graphics window and select Isometric View. Your assembly should look like the following image.



- 11 In the graphics window or Model browser, right-click the unconnected tee on the downward segment, and select Delete.
The tee fitting is no longer needed so you delete it. The pipe segment automatically heals.

Edit Bent Tube Routes

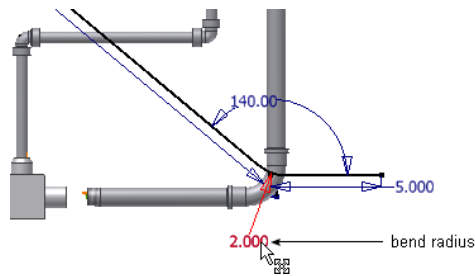
All editing options work for bent tube runs. Edit position, however, behaves differently for tube runs. When moving a tube segment, both adjacent points are moved by the same distance and direction. The adjoining bends change their included angle to make the new route possible. No new segments are introduced. In addition, you can also edit the bend radius and move coupling nodes.

Change Bend Radius

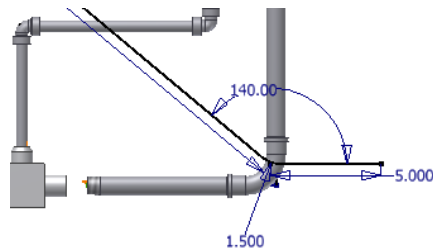
The radius dimension is displayed for each bend in a tube run by default. You can change the bend radius for selected nodes in the active route. To edit the bend radius on individual nodes, you edit dimensions.

Change bend radius on individual nodes

- 1 Activate Route03 in the AirSystem1:1 run.
- 2 Double-click the bend radius dimension as shown in the following image.



- 3 Enter 1.5 inch as the new value, and then click the green check mark. The radius changes for that node only.



- 4 Right-click and select Finish Edit.

NOTE Alternatively, you can choose Select Sketch Features from the Select tool on the Quick Access toolbar while the top-level assembly is activated, and then double-click the bend radius dimension. Enter the new bend radius, and click Update.

Move Coupling Nodes

In tube routes you can move coupling nodes in three dimensions when the route points are not fully constrained. The system then dynamically updates the route sketch and the collinear constraints on the adjacent segments remain. You can also reposition coupling route points by editing the segment dimensions.

NOTE To remove a coupling, you must delete the associated route point in the route environment.

In this exercise, you use the 3D Move/Rotate tool.

Move the coupling node

- 1 Activate Route03 in the AirSystem1:1 run.
- 2 Right-click the intermediate route point on the segment, and select 3D Move/Rotate.
- 3 Drag the triad axes or enter the X, Y, or Z values to move the point as needed.
- 4 Click OK.

Delete Routes and Runs

You can delete a selected run or any individual routes that it contains. You can also delete one or more selected routes.

When you delete a populated route, you must also indicate whether to delete the segments and fittings that it contains or to save the fittings for future routing. Fittings that are not deleted are placed above the first route in the active run.

Delete a run

- 1 Activate Tube & Pipe Runs in a tube and pipe assembly.
- 2 Right-click the pipe run to delete, and select Delete Run.

Delete a route and its components

- 1 Activate the run containing the route to delete.

- 2 In the Model browser, right-click the route and select Delete.
If the routes to delete are populated, the Delete Route Components dialog box displays.
- 3 Do either of the following:
 - To delete both segments and fittings, click the button beside Fittings and Segments of Selected Route.
 - To delete segments and keep the fittings, click the button beside Segments of Selected Route.
- 4 Right-click and select Finish Edit.

Use Content Center Libraries

7

The Autodesk Inventor® Content Center provides standard Autodesk Inventor parts, as well as conduit parts and tube and pipe fittings. The components comply with industry standards.

In this chapter, you learn the basics about the Content Center.

About Content Center

The Content Center is an Autodesk Inventor tool used for accessing and maintaining the libraries. You can create as many libraries as you need on the Autodesk® Vault server, and attach them to the Content Center using the Configure Content Center Libraries tool.

Autodesk Inventor® Routed Systems provides three tools to manage libraries, configure libraries, and edit library content with varied functions:

Autodesk Vault server To use libraries in the Content Center, the server must be installed. You can create new libraries on the server and then use the Configure Content Center Libraries tool to attach it to the Content Center. You can permanently delete a library from the server after you detach it from the Content Center.

Configure Content Center Libraries tool You can attach the libraries on the server to the Content Center.

Content Center Editor tool You can edit library parts from within the Content Center.

In this chapter, you learn the basics about the Content Center. For more information about managing, configuring, and using the Content Center, refer to the Help.

To manage and configure libraries, you must have read/write permissions to the Content Center and libraries. Otherwise, go to the system administrator to request permissions.

NOTE In the exercises that follow, you must have read/write permissions to the Content Center.

Content Center Libraries

The Content Center Library contains libraries of standard and custom Autodesk Inventor parts (fasteners, steel shapes, shaft parts) and features that can be placed into assemblies using AutoDrop. It also includes conduit parts (hoses, pipes, tubes) and tube and pipe fittings (couplings, elbows, flanges, tees, and so on) that are standard for creating tube and pipe systems.

Libraries from several different industry standards are provided. You access the library parts using the Place from Content Center and Replace from Content Center tools. Once you locate the part you need in a library, you set parameters for the part, and then use AutoDrop to insert the part in the assembly file or directly onto a run. When a library part is dropped into an assembly, an *.ipt* file is created for the inserted part and is added to the Model browser.

For Tube & Pipe specific libraries, you can manage them the same as you do with Inventor-specific libraries. For example, to reuse custom library parts that you published in earlier versions of Autodesk Inventor Routed Systems, you can import the appropriate library database files. Using the Publish Part tool, you can add your own tube and pipe parts that have been authored to the Content Center libraries for standard use.

NOTE When a tube and pipe style uses the conduit part or fittings in a library that is currently not available, you must configure the library before you can continue modifying this style and populating routes that use this style.

Manage Libraries

To manage libraries on the server, click Start ► Programs ► Autodesk ► Autodesk Vault server ► Console. Log in to the server, and click OK. Once the console is open, you can perform several operations.

Create a new library on the Autodesk Vault server

- 1 Start the Vault server Console.
- 2 In the console navigation pane, expand the appropriate server.
- 3 Right-click Libraries and select Create Library.

If the library has not been attached to the Autodesk Vault server, you must attach it first.

Attach a library to the Content Center

- 1 In the Vault server Console navigation pane, expand the appropriate server.
- 2 Right-click Libraries and select Attach Library.
- 3 Select the library to attach from the list, and click OK. The *.mdf* and *.ldf* file of the library must be placed under the data folder of your SQL server path.

Edit read/write permission to a library

- 1 In the Vault server Console navigation pane, expand the appropriate server.
- 2 Right-click a library in the list and select or clear the Read Only check mark.

NOTE For default libraries you cannot clear the Read Only check mark.

Detach a library

- 1 In the Vault server Console navigation pane, expand the appropriate server.
- 2 Expand the Libraries folder.
- 3 Right-click a library in the list and select Detach.

This removes the library from the server, but does not permanently delete it.

Export a library

- 1 In the Vault server Console navigation pane, expand the appropriate server.
- 2 Expand the Libraries folder.
- 3 Right-click a library in the list and select Export.

Delete a library permanently

- 1 In the Vault server Console navigation pane, expand the appropriate server.
- 2 Expand the Libraries folder.
- 3 Right-click a library in the list and select Delete.
- 4 Click OK.

Configure Libraries

To configure Content Center Libraries for use in a project, open Autodesk

Inventor Routed Systems and click  ► Manage ► Projects.

Using the Configure Content Center Libraries tool you can:

- Add new libraries.
- Import Inventor libraries and the Tube & Pipe Library from previous versions.
- Edit or remove existing custom libraries in the Libraries list.
- Update the libraries in the list.


Typically the prerequisite for configuring libraries in the Content Center is that the libraries must exist on the server. Importing libraries to the Content Center is an exception:

- If the target library is a pair of *.mdf* and *.ldf* files, start the Autodesk Vault Manager. Right-click Libraries in the Admin Tools browser and select Import Library.
- If the target library is *.mdb* file, start the Configure Content Center Libraries tool. Click the Import Library tool.

Workflow: Configure libraries in Content Center


- 1 Open Autodesk Inventor Routed Systems.

- 2 Click  ► Manage ► Projects.

- 3  In the Projects dialog box, set the desired active project and click the Configure Content Center Libraries tool on the bottom right corner.

Workflow: Edit library contents in the Content Center


- 1 Open Autodesk Inventor Routed Systems.

- 2  On the ribbon, click Manage tab ► Content Center panel ► Editor.

Retrieve the most current library data

- 1 Open Autodesk Inventor Routed Systems.

- 2 Click  ► Manage ► Projects.

- 3  In the Projects dialog box, click the Configure Content Center Libraries tool on the bottom right corner.

- 4 Select one or multiple libraries and click the Update Library tool.

Edit category properties and part families



- 1 On the ribbon, click Manage tab ► Content Center panel ► Editor.
- 2 Use appropriate tools to make edits.

Place Tube and Pipe Parts

Both conduit parts and fittings can be placed into a tube and pipe assembly from the Content Center using AutoDrop.

The following tools are available to locate tube and pipe library parts:

- Use the Place from Content Center tool to directly place library parts. For detailed instructions, see [Insert Library Parts Using AutoDrop](#) on page 132 in Chapter 6.
- To replace an existing fitting or all instances in the same fitting type, right-click the fitting to replace and start the Replace from Content Center tool. For detailed instructions, see [Replace Existing Fittings](#) on page 136 in Chapter 6.

Author and Publish

8

To take advantage of the Content Center functionality, such as queries and dropping conduit parts and fittings, you must publish custom tube and pipe iParts and normal parts to the Content Center. For example, custom parts in the Content Center can be shared in multiple projects so you do not need to save the same custom parts for each project workspace.

In this chapter, you learn how to use the Tube & Pipe Authoring tool to author tube and pipe iParts, and use the Publish Part tool to publish authored parts to the Content Center for future use in tube and pipe assemblies. You also learn how to publish normal parts as fittings.

About Authoring and Publishing

You must publish parts to a library to which you have read/write permissions. Otherwise, go to your system administrator to request permissions.

Publishing parts automatically points to the default category in which they must be published in the active library. If you click into another category, click the default category in the library tree again to continue the publishing.

The task of authoring and publishing is performed using two tools in sequence:

- Autodesk Inventor® Tube & Pipe Authoring tool
- Autodesk Inventor Publish Part tool

Author Tube and Pipe Parts

After you create a custom fitting or component and transform it to an iPart factory, you can use the Tube & Pipe Authoring tool to prepare the part for publishing to the Content Center. You can also author and publish a normal part (not an iPart) as a fitting.

You can author and publish tube and pipe iParts as either pipes or fittings, but can only author and publish normal parts as fittings.

When authoring tube and pipe iParts, you specify:

- Part type
- Connection number
- End treatment of each connection
- Parameter and iPart table mapping
- Connection point and connection axis
- Gender (female, male, or neutral)
- Engagement of each connection
- ISOGEN properties for fittings
- Mating point and pipe axis (for branch fitting only)
- Cut options (for branch fitting only)

When authoring normal parts, you specify all of the previous items, with the exception of the Parameter and iPart table mapping. In its place, you specify a fixed nominal size.

Access Tube & Pipe Authoring Tool

The Tube & Pipe Authoring tool is available as soon as you have an iPart or a normal part file open.

Access the Tube & Pipe Authoring tool

- 1 Open a normal part or an iPart file.

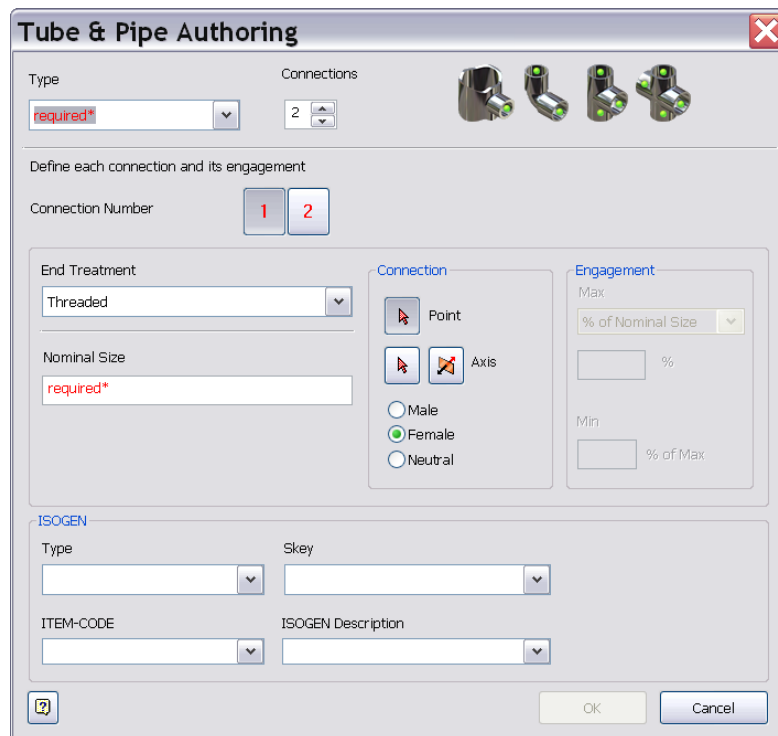


- 2 On the ribbon, click Manage tab ► Author panel ► Tube and Pipe.

TIP You may need to expand the drop-down menu to select the Tube and Pipe author as shown in the following image.



The Tube & Pipe Authoring dialog box displays. The following image is an example of authoring a pipe iPart.



For information on preparing tube and pipe iParts, see the Help.

Author Parameters

Use the Tube & Pipe Authoring dialog box to specify the part type, number of connections, end treatment, required part parameters, engagement, and optional ISOGEN properties for tube and pipe iParts or normal parts being authored.

Once authored, you can publish them to the Content Center. If you do not want to make the part available from the library, you can save the part to a different location.

Type

Specifies the part type to author from the available list. The selected type determines the root category to which you can publish the part in the Content Center.

- To author a standard, two-connection segment, select Tube, Pipe, or Hose. These three part types are not available when you are authoring a normal part.
- To author a standard fitting, select any of the available fitting types.
- If it does not fit into any available fitting types, select Other.

All standard part types such as tubes, pipes, hoses, adapters, crosses, elbows, flanges, branched fittings, gaskets, and so on are available for authoring and publishing.

Connections

Specifies the number of connections for the part being authored. Each part type defaults to a connection number. If you do not use the default number of connections, set a new value in the Connections list. If you are authoring a cap, elbow, tee, or cross, click the appropriate fitting image.

Connection Number

Specifies the connection to define. The number of connections changes dynamically depending on the quantity specified for the part being authored.

Click the connection number button representing the connection to set. The button changes from red to black when required connection criteria is satisfied. Continue to click buttons and set parameters until all connections are defined.

- Red buttons indicate that the connection criteria have not been satisfied.
- Black buttons indicate the connection has the necessary information for authoring and publishing.

Angle

Read-only display of the angle value between the two primary connections in a self draining elbow or tee. This area of the dialog box does not appear unless the fitting being authored has an angle value greater than 45 and less than 90 degrees.

End Treatment

Specifies the end treatment used for each connection number. The End Engagement list filters out all possible end treatments for the current connection point.

Click the arrow to select an end treatment from the list.

Parameter

Specifies parameters for each connection number. All listed parameters must be mapped. This setting is available only when you are authoring a tube and pipe iPart.

Click the arrow to select a parameter from the list, and then map each one to the appropriate iPart table column. Once set, parameter mappings are automatically populated for all other connections. You can reset them if the connection is a different size.

Typically fitting iParts must contain the nominal diameter (ND). For butt welded branched fittings, the schedule number (SN) is also required.

Conduit iParts must contain:

- Nominal diameter (ND)
- Schedule number (SN)

- Outside diameter (OD)
- Inside diameter (ID)
- Pipe length (PL)

NOTE Pipe Length is only required on the first connection number. You do not need to map the iPart column to Pipe Length for all other connection numbers.

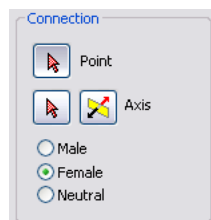
Nominal Size

Specifies the fixed nominal size of the normal part being authored. Normal parts can only be authored and published as fittings. This setting is available only when you are authoring a normal part.

Connection Point, Connection Axis, Gender

Connection settings determine the connection geometry, type, and direction of the axis. In this case, set and map the parameter before you continue.

This section typically has the following options with an exception of the last connection number on a branched fitting:



Connection Point tool

Sets the point at which the fitting connects with existing geometry. Click the icon, and then select either a circular edge or existing work point for the connection in the graphics window. For gasket parts and butt weld components, select a point on the connection face. A work point is displayed at the center of the circular edge.

Connection Axis tool

Sets the direction in which to connect the part being published. Click the icon, and then select either a circular edge or existing work axis for the connection. An arrow in the graphics window shows the set axis direc-

tion. The axis direction should point outside of the part, not inside. To flip the arrow in the opposite direction, click the Flip Direction button.

Flip direction Changes the axis to the opposite direction.

Male, Female, Neutral Select the appropriate gender for the connections. For example, gaskets and butt weld components are typically neutral gender.

NOTE For more information about the connection information for authoring branched fittings, refer to the Help.

Engagement

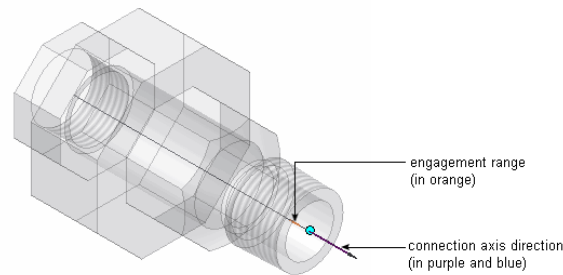
Before a custom fitting can be placed in a tube and pipe assembly, engagement information must be set. Library parts from the Content Center already include this information.

Pipe engagement determines the range (minimum and maximum engagement position values) for how a pipe is inserted into a fitting. The maximum engagement position (Max) is defined by a work point on the connection axis.

The Max work point can be an existing work point or it can be created by the intersection of a selected planar face and the connection axis:

- The actual Max distance is the distance from the connection point to the Max point along the connection axis.
- The minimum engagement position (Min) is the distance from the connection point to the Min.

The set engagement range displays in the graphics window, and updates dynamically as changes are made to the Max and Min settings.



NOTE The Engagement section of the dialog box is not enabled until the required settings in the previous sections are set.

There are three types of engagement that you can set:

% of Nominal Size The Max is a percentage of the nominal size value (NS). This engagement type automatically adjusts to changes in the nominal size. Gaskets and butt weld components have 0% engagement.

To Plane/Point (Associated) Establishes an association between the connection point and a work point representing the maximum engagement position. The point can be an existing work point or work point that is created by the intersection of a selected planar face and the work axis.

Distance The actual maximum engagement position distance. You can enter the distance in the dialog box.

The minimum engagement position is always specified as a percentage of the Max distance.

When a pipe is inserted into a fitting, the pipe fitting procedure applies the defined minimum and maximum to ensure the engagement position is within the specified range. Depending on the minimum increment set in the style, the end of the pipe falls between the Min and the Max.

NOTE For more information about setting the different types of engagement, refer to the Help.

ISOGEN Properties

To support the optional ISOGEN output format, you must add certain properties to the published fittings. The ISOGEN properties you must add are:

Type	Specifies the piping component type property required by ISOGEN <i>.pcf</i> files.
SKEY	Four-character symbol keys that consist of the two-character pairs for part type and their valid end types. The SKEY is used as part of ISOGEN end point input. Wild card values of **, @, or + may also be included in the SKEY. The wild card values are replaced by additional information such as a range of integer values denoting bend radius or number of segments.
Item-Code	Specifies the parameter to use for the manufacturer's item-code for the selected part. For iparts, it selects the ITEMCODE column in the family table and maps it to the item-code property. For non-iparts, enter the needed code based on the manufacturer's data. You can also click the arrow to map the item code to other parameters.
ISOGEN Description	Provides a manufacturing description of the selected part. Click the arrow to select from the list, or enter the needed description.

Prepare iParts

When creating the iPart, make sure the necessary values are included in the iPart table. It is recommended that you learn what parameters are required to author tube and pipe parts. You can then create appropriate parameters and features when transforming the part to an iPart factory.

For more information about preparing an iPart, see:

- Autodesk Inventor Skill Builders: Authoring, Publishing, and Styles - Part 1. It is available at: <http://www.autodesk.com/inventorpro-skillbuilder>.
- Help system.

In this exercise, you use sample iParts to learn how to author and publish to the Content Center.

When a sample tube and pipe iPart is open:

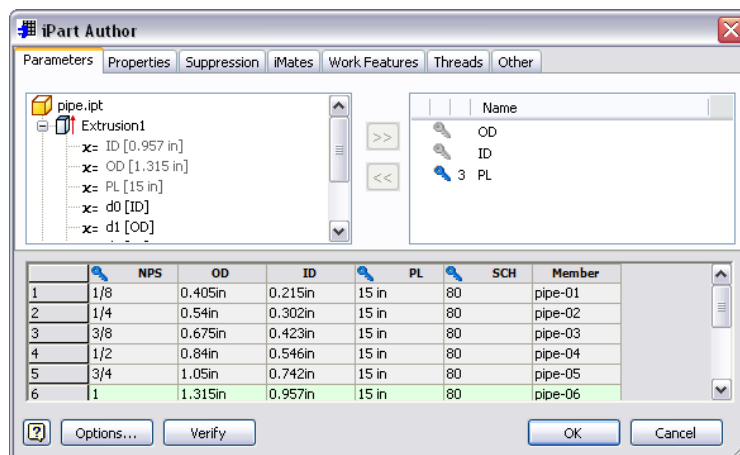
- Use the Parameters tool to verify the model parameters and user parameters that are used in the iPart. It is recommended that you use model equations to define the part parameters so that multiple part occurrences specified in the iPart Authoring table can dynamically update in proportion.

- Double-click Table in the Model browser to open the iPart Author table. Verify all part occurrences that are defined for the iPart. Each row in the iPart Author table represents a part occurrence for the part family to publish to the Content Center.
- If you must customize the pipe length of the part when placed from the Content Center at a later time, specify it as Custom Parameter Column in the iPart Author table before authoring. Click the column head to select the entire column, right-click, and then select Custom Parameter Column.
- Each row in the iPart Author table can have its own material style. When needed materials for an iPart are not available in the Styles Library, routes and runs using this material cannot be properly populated. To add new materials to the Styles Library, you must enable the Styles Library for your project. For detailed instructions on how to enable the Styles Library, see [Set Up Projects For Exercises](#) on page 11. For detailed instructions on how to add and modify material styles, see the Index entry “material styles” in the Help.

Set Default Rows in iPart Author Tables

Each iPart Author table has a default row. The default row determines the library part that displays when you open a part family in the Content Center.

For instance, open the sample pipe iPart *Autodesk\Inventor Professional <version>\Tutorial Files\Tube & Pipe\Example_iparts\pipe.ipt*. In the Model browser, double-click Table to open the iPart Author dialog box:



The row highlighted in green is the default row. To change the default row, right-click the appropriate row number, such as 1/2, and select Set As Default Row. Click OK and the pipe part in the graphics window automatically updates.

Once this pipe iPart is published to the Content Center, when opening the part family, the preview image shows the part with parameters in the default row.

You can open 90-degree elbow and 45-degree elbow to verify their default rows on the iPart Author dialog box.

NOTE Do not change the default row for sample iParts.

Set iPart Family Key Columns

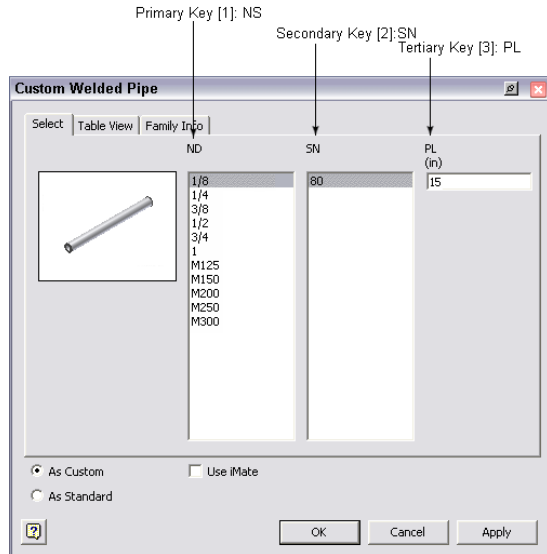
An iPart can have up to 9 keys used to define criteria for AutoDrop and filtering out the iPart for tube and pipe styles.

Pipe iParts typically have 3 family key columns: nominal size, schedule number, and pipe length. For the sample pipe iPart:

- On the Other tab, NPS is primary key {1} and SCH is secondary key {2}.
- On the Parameters tab, PL is tertiary key {3}.

Fitting iParts typically have 1 family key column: nominal size. For the sample elbow iParts, on the Other tab, NPS is primary key {1}.

The following image illustrates the relationship between iPart family key columns and part family thumbnail page in the Content Center.



Family key columns play an important role in the file naming schema for part families in the Content Center. After authoring and publishing, the sequential combination of hierarchical keys form the value of the pre-defined property {DESIGNATION} in the Content Center. In this case, {DESIGNATION} is pipe {NS} {SN} {PL}, in which pipe indicates the conduit part type.

TIP When you publish authored parts to the Content Center, you can also set and modify family key columns on the Publish Guide. For more information, see [Publish Authored Parts](#) on page 175.

Author iParts

The Tube & Pipe Authoring tool predefines a list of available part types to author, such as Tubes, Pipes, Hoses, Couplings, Elbows, and Tees. When no category is appropriate for the fitting part being authored, select Other.

In this exercise, you use the Tube & Pipe Authoring tool to standardize three sample iParts that are stored in *Autodesk\Inventor Professional <version>\Tube & Pipe\Tutorial Files\Tube & Pipe\Example_iparts*: pipe iPart, 45-degree and 90-degree elbow iParts.

Keep in mind that you cannot select an edge adjacent to a torus or spline face to define connection points and connection axes.

- In the exercises of authoring the pipe iPart, you learn to define connection points and connection axes using valid circular edges.
- In the exercises of authoring the 45-degree and 90-degree elbow iParts, you learn to define connection points and connection axes using existing work points and work axes that were prepared for the exercises.

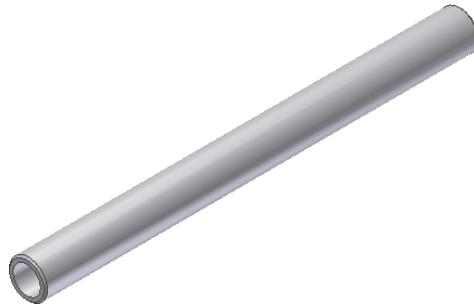
In addition, you can use the skills learned from authoring tube and pipe iParts to author normal parts.

Pipe iPart

In this exercise, you define connection points and connection axes through valid edges on the sample pipe iPart.

Author a pipe iPart

- 1 Open the *pipe.ipt* iPart file.



- 2 In the Model browser, right-click Table and select Edit via Spread Sheet or Edit Table to verify the iPart authoring parameters. Close the file.
- 3 On the ribbon, click Manage tab ► Author panel ► Tube and Pipe.
- 4 In the Type list, select Pipe. The default value in the Connections list is 2 and cannot be modified.
- 5 Verify that connection number button 1 is selected to indicate that you are setting the information for Connection 1.
- 6 In the End Treatment list, select Welded for Connection 1.

- 7 Click the red required field and complete the mapping as shown in the following image.

The Parameter list represents the attributes for the published part and it determines the controlling parameters for styles. The Table Mapping list represents the attributes for the pipe iPart. Parameter and table mapping are always selected as a pair.

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

- 8 Click the Connection Point tool, and then select the connection point in the graphics window.
- 9 Pause the cursor over the geometry until the circular edge is highlighted as shown.



- 10 Click to select the highlighted edge.
A connection point is displayed at the center of the circular edge.



- 11 Click the Connection Axis tool, and then define the connection axis in the graphics window.
- 12 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 the Flip Direction tool. The connection direction must point toward the engaging fitting, so it often points outward.

- 13 Ensure the Neutral check box is enabled. Neutral is automatically selected when the Welded end treatment is set.
- 14 Specify the Engagement setting. For pipe parts, when connecting to the adjacent fittings, the connection only applies to the engagement definition of the adjacent fittings by default. Because 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.

Max: Distance: 0 inch

Min % of Max: 0

NOTE If the engagement section is dimmed, a required setting, such as the connection axis, was not yet set. You must complete all previous settings before you can set engagement values.

The connection number button 1 should now be black, indicating that all connection criteria are satisfied for Connection 1.

- 15 Click the connection number button 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, you do not need to set it again.

- 16 Use the same settings as Connection 1 and be sure to set End Treatment to Welded.

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 need to specify the Pipe Length parameter for connections other than Connection 1. The Pipe Length parameter is unique for a conduit part.

- 17 Verify that each connection number button changes from red to black so all required connection criteria are set.
- 18 Click OK.
- 19 On the Authoring Result dialog box, click OK.
- 20 Save the authored pipe iPart and close the part file.
All authoring changes are saved for the next time you want to author the iPart again. You can publish the authored pipe to the Content Center at a later time.

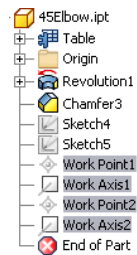
Elbow iPart

In this exercise, you define connection points and connection axes using existing work points and work axes that were created before authoring.

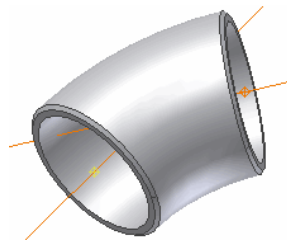
Author the 45-degree elbow iPart

- 1 Open the *45elbow.ipt* iPart file.

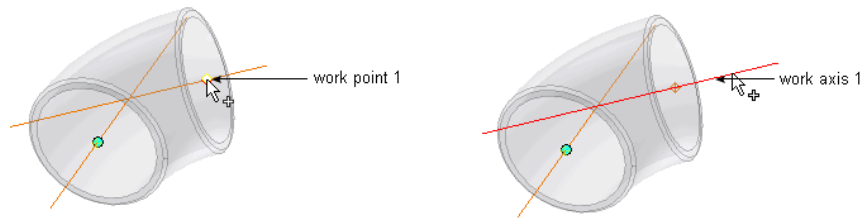
In the following image, work points and work axes that were created to define connection points and axes are displayed in the Model browser.



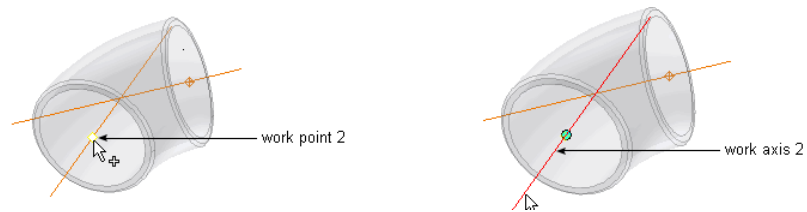
- 2 Press CTRL or SHIFT to select all predefined work points and work axes in the browser, right-click and select Visibility.
Click the browser icon to highlight the corresponding work point or work axis in the graphics window.



- 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 button 1.
- 6 Set the following:
End Treatment: Welded
Parameter and Table Mapping: Nominal Size: NPS
Connection Point and Connection Axis: See the following images
Connection: Female
Engagement
Max: Distance: 0 inch
Min % of Max: 0



- 7** Click connection number button 2, and then repeat Step 6 using the following image to set the Connection Point and Connection Axis.



- 8** 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

- 9** Click OK.
- 10** On the Authoring Result dialog box, click OK.
- 11** Save the authored 45-degree elbow iPart, and close the part file.

Author the 90-degree elbow iPart

- 1** Open the *90LongElbow.ipt* part file.
- 2** Repeat the previous steps using the same settings and connection points as for the 45-degree elbow iPart.
- 3** Save the authored 90-degree elbow iPart, and close the part file.

Practice Your Skills

Use the skills you have learned from the previous exercises and try to author a normal part. Keep in mind that you can only:

- Author a normal part as a fitting.
 - Enter a fixed nominal size.
- 1 Prepare a normal part using Autodesk Inventor.
 - 2 Click the Tube and Pipe Authoring tool.
 - 3 On the Tube & Pipe Authoring dialog box:
 - Select one fitting type.
 - Use the default number of connections or set the connections you need.
 - Select an appropriate End Treatment option depending on the part feature.
 - Enter a value for the nominal size.
 - Define the connection point and connection axis using geometry on the part or predefined work points and work axes.
 - Specify the gender as female or male for each connection.
 - Specify the Max and Min Engagement settings using the Fixed option or the interactive method.
 - Optionally set ISOGEN properties, such as Type, SKey, and ITEM-CODE.
 - 4 Click OK.
 - 5 Save the authored normal part.

Publish to Content Center

The Content Center setting defines the library where you want the part to belong. Parts can only be published to libraries that have Read/Write permission. Otherwise, go to your system administrator to request permissions.

When publishing parts, iParts, or features, the Content Center automatically points to the default root category where you must publish the part. Create appropriate subcategories as needed, and then specify the family properties and category parameters so the published parts or features can be queried at a later time.

NOTE Do not change the default root category Autodesk Inventor matches in the Content Center; otherwise, the publishing functionality is disabled for the category that was changed to. To continue the publishing process, click to restore the default category in the library tree.

Set Up the Library and Subcategories

You cannot add a new library to the Content Center while you are in the process of publishing. Use the Autodesk Vault server and Configure Content Center Libraries tool to create and attach the library to the Content Center beforehand.

In addition, you cannot create new categories in the Content Center while in the process of publishing. Use the Content Center Editor tool to set up the categories beforehand.

To publish authored parts in the exercise that follows, you first set up the library. You use the default category to publish at a later time.

Set up a new library

- 1 Log in to the Vault server Console.
- 2 Expand the appropriate server.
- 3 Right-click Libraries and select Create Library. Create a new library named *CustomLibrary*. Click OK.
By default, the new library is attached to the server and you have the read/write permission.
- 4 Exit the console.
- 5 Ensure that the Internet Information Server (IIS) is started. Otherwise, libraries on the server cannot be accessed. This is not required for standalone installations.
- 6 Start Autodesk Inventor Tube & Pipe.

- 7 Click  ► Manage ► Projects.

The Projects dialog box displays the project configuration information.

- 8 On the Project pane, ensure that the Tube & Pipe Tutorial Files project is active and use the Configure Content Center Libraries tool ► Add Library tool to attach CustomLibrary to the Content Center.
For detailed instructions, see [Set Up Projects For Exercises](#) on page 11 in Chapter 1.
- 9 Save the changes for the project and close the Projects dialog box.
- 10 On the ribbon, click Tools tab ► Content Center panel ► Editor and update the libraries in the Content Center as needed.
- 11 Optionally, you can create subcategories to publish parts.
For detailed instructions, refer to the Help.

In the exercises that follow, you publish the authored parts to the default category that the system automatically matches. You do not need to create subcategories in this step.

Publish Authored Parts

When publishing authored parts, you must map the part parameters to the category parameters. The information you provide to define the part sets the structure for its location in Content Center, provides a base for the part file name, and makes the part attributes available for future queries.

In the exercises that follow, you use the Publish Part tool to add authored pipe, 45-degree elbow, and 90-degree elbow iParts to CustomLibrary that you created in the previous exercise.

NOTE If you need to use published parts to define tube and pipe styles, you must specify the Standards Organization and Standard family properties during publishing. They are the key style criteria to filter out library parts on the Tube & Pipe Styles dialog box.

Pipe iPart

In this exercise, you publish the authored pipe iPart to *CustomLibrary*.

Publish an iPart

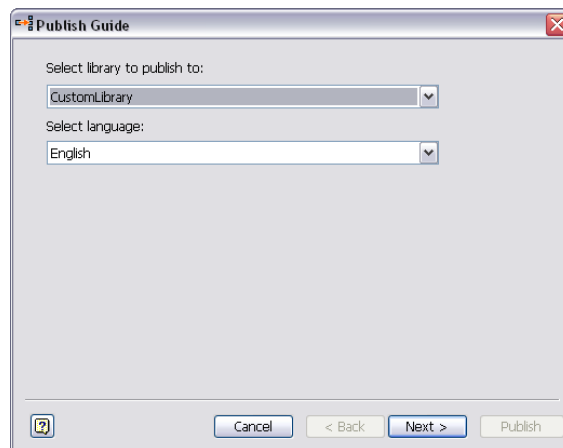
- 1 Open the *pipe.ipt* part you previously authored.



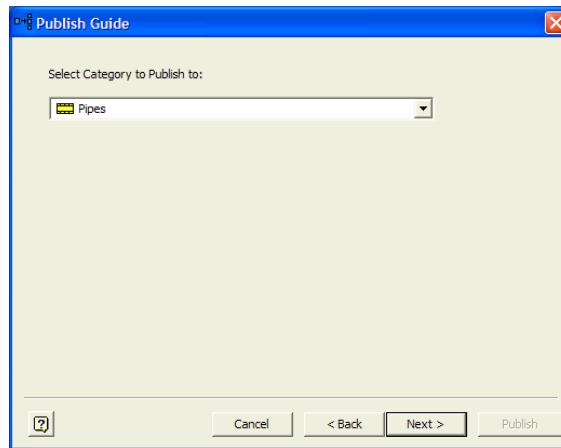
- 2 On the ribbon, click Manage tab ► Content Center panel ► Publish Part.

The Publish Guide dialog box displays.

- 3 Select CustomLibrary from the list and accept the default language, and then click Next.



- 4 Select Pipes from the list and click Next.

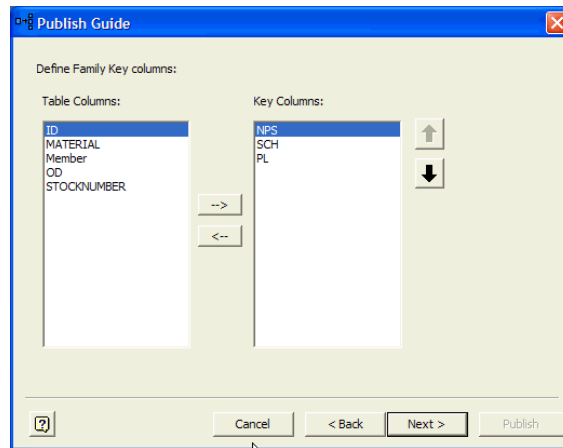


- 5 Accept the default mapping between the iPart family columns and category parameters and click Next.

The mapping has been done during authoring. You do not need to do anything for tube and pipe parts publishing unless you need to change the default mapping.

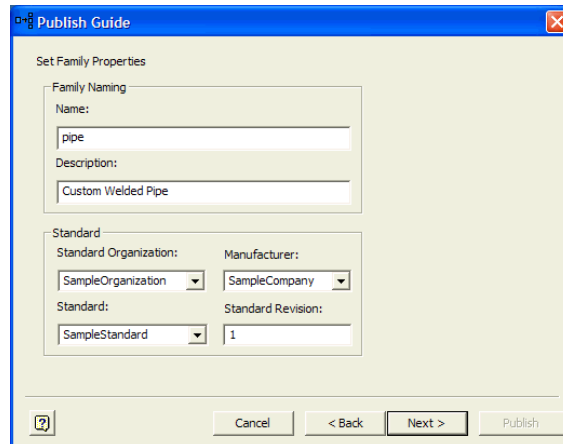
NOTE In the mapping table, required family columns and category parameters represent a different background color from optional properties that follow.

- 6 Optionally, define the family key columns if you have not set the family key columns during iPart preparation or that do not satisfy the design. In this exercise, ensure that you have the three family key columns for the pipe part in the order of NPS, SCH, and PL. Click Next.



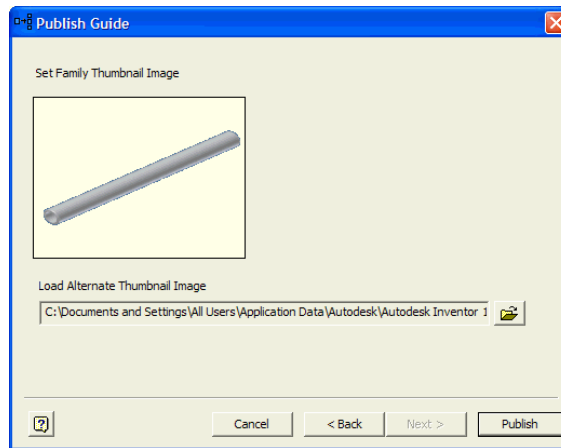
NOTE For detailed instructions on how to set the family key columns when preparing a tube and pipe iPart, see [Prepare iParts](#) on page 163 in this chapter.

- 7 Specify the pipe part family properties as shown in the following image:



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

- 8 Click Next.
The preview image of the published part is displayed. You can change the thumbnail image for future use in the Content Center.



- 9 Click Finish.
- 10 On the Publish Guide dialog box, click Publish.
- 11 When the Publish Result dialog box prompts about the successful publish, click OK.
When processing is complete, the part is added to Custom Library ► Tube & Pipe ► Conduits ► Pipes category. You can use the Content Center Editor tool to verify the published pipe part family.

Once published, the Standard information is available in the lists on the Tube & Pipe Styles dialog box. The part materials are available when the appropriate standard specification is selected.

You can also navigate to the published union part family using the Content Center tool and practice using the AutoDrop functionality to place it in a sample model or model of your choice.

Elbow iPart

Use the skills you just learned to publish the 45-degree and 90-degree elbow iParts to Custom Library ► Tube & Pipe ► Fittings ► Elbows category.

Publish the 45-degree elbow iPart

- 1 Repeat the previous steps to publish the 45-degree elbow iPart to *CustomLibrary*.

Use Tube & Pipe ► Fittings ► Elbows as the category path.

Click Next to go to the next wizard.

- 2 Accept the default mapping between the iPart family columns and category parameters and click Next.

- 3 Accept the default family key columns and click Next.

- 4 Specify the following family properties and click Next.

Family Naming:

Name: 45Elbow

Description: Butt Weld 45 Elbow

Standard:

Standard Organization: SampleOrganization

Manufacturer: SampleCompany

Standard: SampleStandard

Standard and Revision: 1

- 5 Review the published elbow information and go back to the preceding wizards to make appropriate changes as needed. Click Next until you reach the final screen of the wizard, and then click Finish.

- 6 Click Publish, and then OK.

Publish the 90-degree elbow iPart

- 1 Repeat the steps above to publish the 90-degree elbow iPart to *CustomLibrary*.

Use Tube & Pipe ► Fittings ► Elbows as the category path.

Click Next to go to the next wizard.

- 2 Accept the default mapping between the iPart family columns and category parameters and click Next.

- 3 Accept the default family key columns and click Next.

- 4 Specify the following family properties and click Next.

Family Naming:

Name: 90LongElbow

Description: Butt Weld 90 Long Elbow

Standard:

Standard Organization: SampleOrganization

Manufacturer: SampleCompany

Standard: SampleStandard

Standard and Revision: 1

- 5 Review the published elbow information and go back to the preceding wizards to make appropriate changes as needed. Click next until you reach the final screen of the wizard, and then click Finish.
- 6 Click Publish, and then OK.

Create Styles Using Published Parts

Once you successfully publish custom conduit parts and fittings to the Content Center, you can create new styles based on those published parts.

Most styles require a pipe, elbows, and a coupling. Welded styles, such as the style in this exercise, do not require a coupling.

In this exercise, you create a welded Rigid Pipe with Fittings style using the published pipe and 90-degree elbow. Only two parts are required by this style, since it is a welded style with 90-degree elbows only.

Set a new style with published parts

- 1 With a tube and pipe runs assembly open, open the Tube & Pipe Styles dialog box.
- 2 On the Tube & Pipe Styles dialog box, select ASTM A53/A53M-ASME B16.11 - Welded Steel Pipe from the Style list as the base.
- 3 Click New to create a new style named *Custom Welded Pipe (1/4, 90)* and clear the values.
- 4 In the Components table, right-click the Pipe row and select Browse.
- 5 In the Library Browser, click the Standard check box and select SampleStandard.
- 6 Click the Filter tool, so the custom pipe you created is the only item listed.
- 7 Select the pipe, and then click OK.
- 8 On the General tab under Diameter, set the following options:

Nominal Diameter: 1/4 inch

Schedule: 80

Keep in mind:

- SampleStandard is the Standard parameter (not Standard Organization) you manually set for the part family standard properties during publishing.
- When SampleStandard is selected in the Standard list, enable the materials filter to display all available materials in the list. If the desired material is not displayed, use the Content Center tool to verify the part family does own this material. Use the Styles Editor tool to ensure this material is added to the Styles Library.

9 Accept the other general settings.

10 On the Tube & Pipe Styles dialog box, Rules tab, specify:

Min: 1.000 inch

Max: 200.000 inch

Inc.: 0.1 inch

11 Repeat steps 4 through 9 to select the 90-degree elbow part.

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.

12 Click Save.

13 Click Close.

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

Document Routes and Runs

9

Drawing views, parts lists, and the Bill of Materials table specific to the Tube & Pipe Add-in are used to describe individual pipe runs and components.

In this chapter, you learn how to create design views for drawings, create varied drawing views, recover route centerlines, dimension routes and runs, and create and export the bill of materials, create parts lists, and annotate drawings with piping style data.

About Documenting Routes and Runs

In drawings, tube and pipe information is treated like other parts and subassemblies. You can describe the individual pipe runs and components and detail them using normal drawing manager methods and tools unless noted otherwise.

The two drawing properties are typically used to roll up conduit parts in the parts lists correctly. You can add them to either specific drawing documents or drawing templates:

- Conduit segment length. With an internal name of Base QTY. It specifies the length of conduit parts.
- Raw material description. With an internal name of Stock Number. It stores the raw material descriptions for conduit parts.

To create tube and pipe drawings correctly, you may need to do the following:

- If you are migrating R9 or earlier tube and pipe drawing documents to the current version of Autodesk Inventor®, determine the BOM to use. In this exercise, you use the new BOM.

- To document individual routes and runs, create design view representations in which you turn off the visibility of unnecessary components, and then apply them to appropriate drawing views.
- Use broken views to fit long nondescript sections of pipe on a drawing.
- Use detail views to show selected fittings.
- To dimension routes and runs correctly, recover route centerlines. To dimension unpopulated routes, you can recover route centerlines.
- Add the Pipe Length and Stock Number properties to parts lists and a Bill of Materials table.
- To create BOMs for specific routes and runs, set their BOM Structure property to Reference.

Workflow for Documenting Pipe Runs

To document a tube and pipe assembly, you may need to use the following workflow.

Document a tube and pipe assembly

- 1 Create a tube and pipe assembly containing populated or unpopulated routes.
- 2 Edit design views for the drawing file if you want to output specific views, especially in some complicated assemblies.
- 3 Optionally, edit drawing templates.
- 4 Create one or more base views.
- 5 Create other required views from the base view or any available parent views.
- 6 Recover route centerlines and dimension routes and runs.
- 7 Create and export the BOMs.
- 8 Create parts lists.
- 9 Add balloons to the pipe run components and annotate drawings with the piping style data.

NOTE The results in your exercise may differ from figures illustrated in this chapter, depending on specific working environment, original assembly, and workflow used.

Use Drawing Templates

When documenting a tube and pipe assembly, each new drawing file uses a drawing template. You can update existing templates or create and add new ones into the Templates folder. For **Windows® XP** this is located under *Program Files\Autodesk\Inventor<version>\Design Data\Tube & Pipe* by default. For **Windows Vista®** this is located under *Users\Public\Documents\Autodesk\Inventor<version>\Design Data\Tube & Pipe* by default.

For more information about customizing drawing templates, refer to the Help.

Prepare Design View Representations


A Design View is created in the assembly environment and preserves a designated representation view of assembly components. It can be mapped to drawing views of the assembly file.

In cases where only specific pipe runs in a complicated assembly need to be documented, you can define a specific design view in which components that are contained in the top-level assembly but not within the pipe runs are hidden. Thus, components with Visibility off are not displayed in the drawing view when the relevant design view is selected for the drawing file.

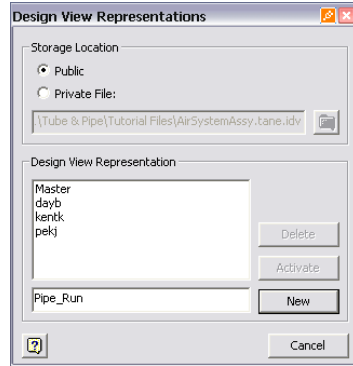
Before creating drawing views for pipe runs, you can customize specific design views for the assembly environment using the Design View Representations tool.

In this exercise, you create a design view in which you turn off the visibility of IBeam in the tube and pipe assembly, *AirSystemAssy.iam*.

Create a new design view

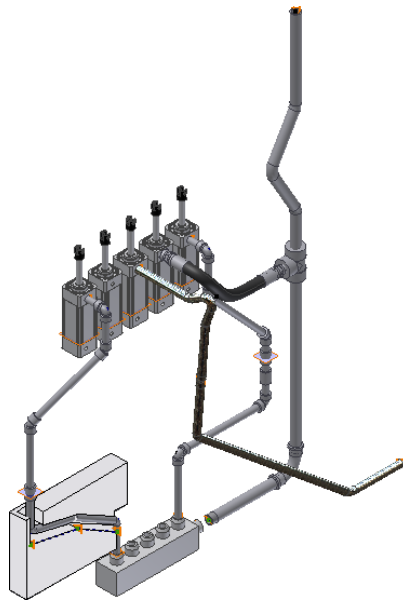
- 1 Open the *AirSystemAssy.iam* assembly.
- 2  At the top of the Model browser, click the Design View Representations tool.

- 3 In the Design View Representations dialog box, accept the default public storage location, or select Private and specify your own storage location.
- 4 Enter a Design View name, *Pipe_Run*, and click New.



The new *Pipe_Run* design view is active by default. In this case, *Pipe_Run* is used to create drawing views for the tube and pipe assembly in later exercises.

- 5 Close the dialog box.
- 6 In the Model browser, right-click the *IBeam:1*, and then clear the Visibility check mark. You can also define any other assembly viewing characteristics.
- 7 Turn off the visibility of the *AirSystem2:1* run.
- 8 Save the top-level assembly.



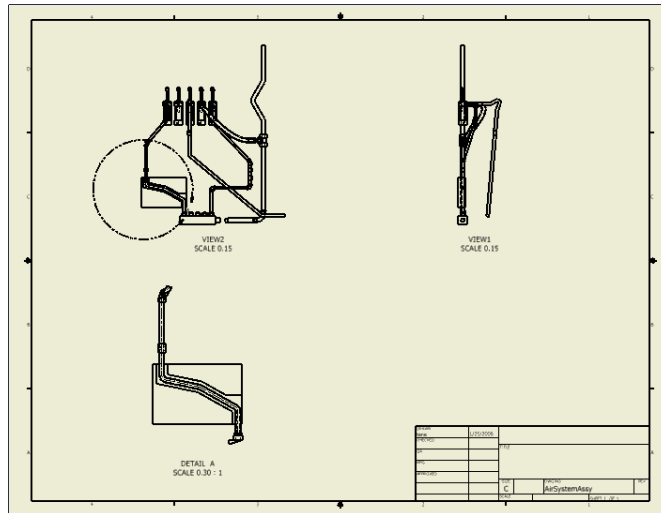
NOTE You can set the association with the design view when creating a drawing view. The association with the design view can be changed after the drawing view is created. When associations are set, it is possible to update the drawing view automatically when changes are made to the assembly in the selected design view.

Create Drawing Views

In this exercise, use the Place Views tab to create a base view, projected view, and detail view for a tube and pipe assembly you previously created. Pause the cursor over the images to view the tooltip.



The following figure shows the completed drawing views:



Create Base Views

Before creating other drawing views, such as projected views and detail views, you must first create at least one base view.

Create a base view for a tube and pipe assembly

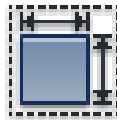
- 1 Open the *AirSystemAssy.iam* assembly. Make sure you have the Pipe_Run design view active for the tube and pipe assembly in which the IBeam and AirSystem2 are not visible.



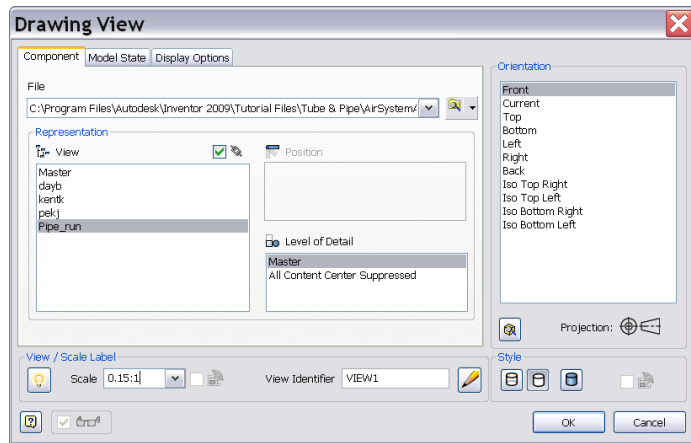
- 2 Click  ► New.

- 3 In the New File dialog box select *Standard_AIP.idw* and then click OK. The Drawing environment is activated. Examine the Model browser. The IBeam and Pipe Run 2 have visibility turned off.

- 4 Save the drawing document as *AirSystemAssy.idw*.



- 5  On the ribbon, click Place Views tab ► Create panel ► Base. The Drawing View dialog box displays.



- 6 In the Drawing View dialog box, Component tab, specify:
- File **Windows XP:** *C:\Program Files\Autodesk\Inventor<version>\Design Data\Tutorial Files\Tube & Pipe\AirSystemAssy.iam*. File **Windows Vista:** *C:\Users\Public\Documents\Autodesk\Inventor<version>\Design Data\Tutorial Files\Tube & Pipe\AirSystemAssy.iam*

If an assembly is already opened, it is selected by default. You can click Browse to locate the assembly that you need.

Representation: Pipe_Run, Associative

The active design view for the assembly is selected by default. The drawing view automatically updates when the component visibility in the associated design view changes.

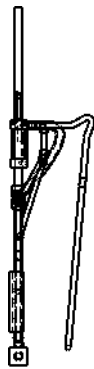
View / Scale Label: 0.15:1, Visible

View Identifier: VIEW1

Orientation: Front

style: Hidden Line

- 7 In the Drawing View dialog box, Display Options tab, use the defaults.
- 8 Move the preview to the upper-right quadrant of the drawing sheet, and then click to place the view.



VIEW1
SCALE 0.15

NOTE If you click OK on the dialog box instead, the view may be placed in a random location. You can move the view by clicking inside the blank area and dragging the rectangular border of the view.

- 9 Save the drawing document.

Create Projected Views

Projected views can be created on any drawing views. When a projected view is placed in the drawing sheet, you can edit the projected view properties using the Edit View tool on the context menu.

Now, project a right view from the base view you just created.

Create a projected view



- 1 On the ribbon, click Place Views tab ► Create panel ► Projected.
- 2 Click inside VIEW1 that is used as the parent view.
- 3 Move the preview to the upper-left quadrant of the drawing sheet, and then click to set the view location.
A black rectangular border appears.
- 4 Right-click and select Create.

The back projected view is in place now.

- 5 To edit the projected view properties, right-click inside the projected view, and select Edit View.

NOTE In the Drawing View dialog box that is displayed, some drawing view settings, such as File and Style, are disabled because the projected view respects the parent view. You can change its default inherited relationship with the parent view to activate more settings.

- 6 In the Drawing View dialog box, specify:

Representation: Pipe_Run, Associative

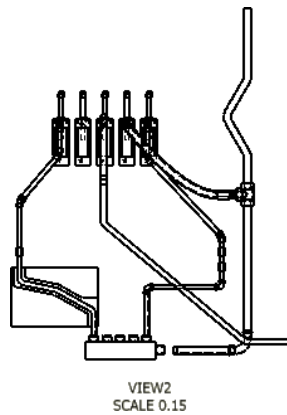
Scale: Scale from Base, Visible

Label: VIEW2, Visible

Style: Style from Base

- 7 Click OK.

The projected view updates to new settings.



- 8 Save the drawing document.

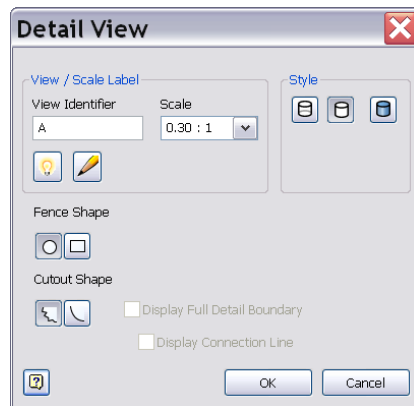
Create Detail Views

Detail views do not respect the drawing view settings from the parent view. They can be placed anywhere in the drawing sheet.

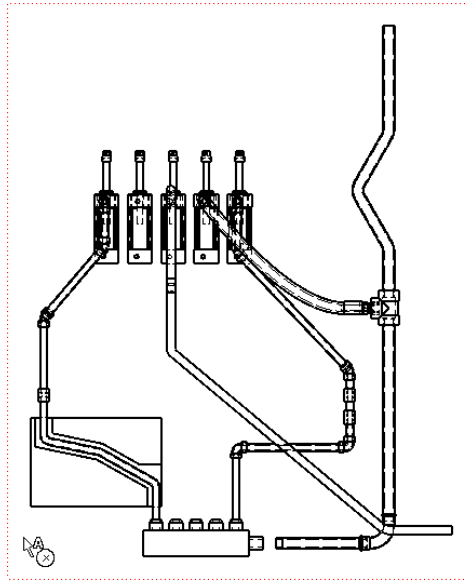
Add a detail view



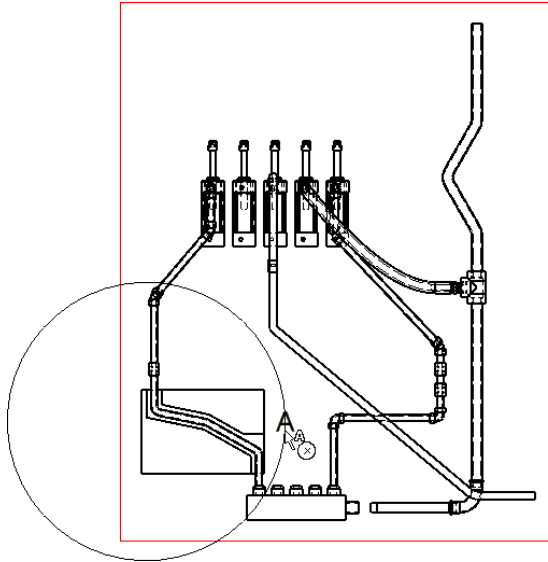
- 1 On the ribbon, click Place Views tab ► Create panel ► Detail.
- 2 Click inside VIEW2 that is used as the parent view.
The Detail View dialog box displays.
- 3 In the Detail View dialog box, specify the settings as shown in the following image:



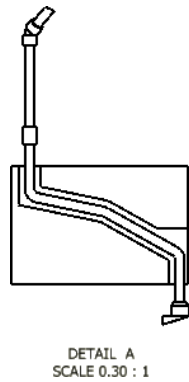
- 4 Click OK.
The circular selection symbol is attached to the cursor.



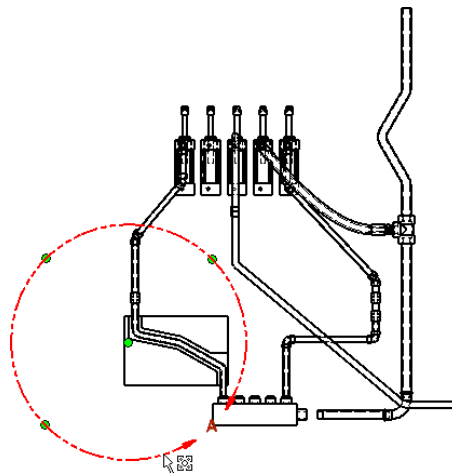
- 5 Place the cursor inside the projected view as shown in the preceding image. Click and drag the cursor to include the part of AirSystemAssy.Route1:1 as shown in the following image.



- 6 Click to set the portion of the projected view.
The cursor changes to a plus symbol. The detail view preview is displayed.
- 7 Move the cursor to the appropriate position, and then click to set the detail view.



- 8 If the portion in the detail view is not as desired, you can use the circular selection tool to adjust the portion. Move the cursor over the circular selection tool in VIEW2 as shown in the following image, click to activate it. To move the circular selection tool, click and drag the green center point. To change the portion, click and drag the circular edge outward or inward.



- 9 Save the drawing document.

Practice Your Skills

In this exercise, you learn to recognize the three drawing views you just created. This will make it easier to recover route centerlines in later exercises.

Move the cursor over the Model browser to highlight each drawing view. Examine the drawing view structure in the Model browser:

- The base view (VIEW1) is the parent view of the projected view (VIEW2).
- The projected view (VIEW2) is the parent view of the detail view (A).

Recover Route Centerlines

By default, the drawing manager hides the centerlines of tube, pipe, and hose routes in drawing views. Centerline recovery is used to control the availability of the route centerlines in active tube and pipe drawing views for dimensioning.

You can control the centerline recovery at the tube and pipe runs assembly, individual run, or individual route levels in the active drawing view, depending on the amount of dimensioning you need. For example, when dimensioning all or most of the routes and runs in a tube and pipe assembly, include route centerlines for the entire tube and pipe runs assembly, and then hide the centerlines for the individual runs and routes you do not need.

You must dimension routes to the centerlines in drawing views. Otherwise, the dimensions may be incorrect. When the specified route centerlines are recovered, you can dimension routes to the centerlines. If you change the centerline recovery back, all recovered centerlines in the active drawing views are deleted so associated dimensions may disappear or become incorrect.

NOTE When a new route or run is created, the route centerline in drawing views has the same centerline recovery setting as the parent. To add or remove centerlines for other Autodesk Inventor components in a standard assembly, change the Automated Centerline Settings on the Tools tab ► Options panel ► Document Settings ► Drawing tab.

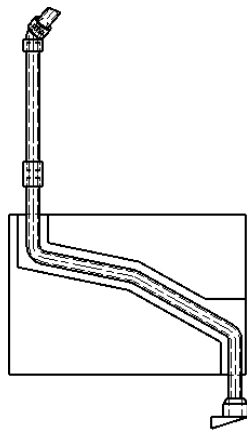
In drawing views that are created from the base view, the route centerline visibility respects the setting in the base view. For example, if you have recovered route centerlines in the base view, the relevant route centerlines are automatically recovered in all associated drawing views that are subsequently created. When route centerlines are not recovered in the base view, you can manually recover them for drawing views you need.

Recover route centerlines in the detailed view (DETAIL: A)

- 1 In the Model browser, under Sheet 1, navigate to VIEW1:AirSystemAssy.iam > VIEW1:AirSystemAssy.iam > AirSystemAssy.iam > Tube & Pipe Runs > AirSystem1:1.

- 2 Expand AirSystem1:1, right-click Route01, and select Include Route Centerlines.

The route centerlines are recovered in both the base view and associated drawing views. Verify the detail view as highlighted in the following image.



DETAIL A
SCALE 0.30 : 1

Dimension Drawing Views

There are two types of dimensions for documenting the drawing views of the tube and pipe assemblies:

Model dimension Controls the features that are applied during sketching or creation of the feature. Autodesk Inventor Tube & Pipe can access the model dimensions of components and place them in a drawing view. You can then retrieve them to display the overall dimensions of pipe runs.

Drawing dimension Added in the drawing view for further documentation using the General Dimension tool on the Annotate

tab. It does not affect features or parts. You can add general dimensions, baseline dimensions, ordinate dimensions, and so on.

The following image shows the Annotate tab on which you can use tools to add dimensions. Pause the cursor over the images to view the tooltip.




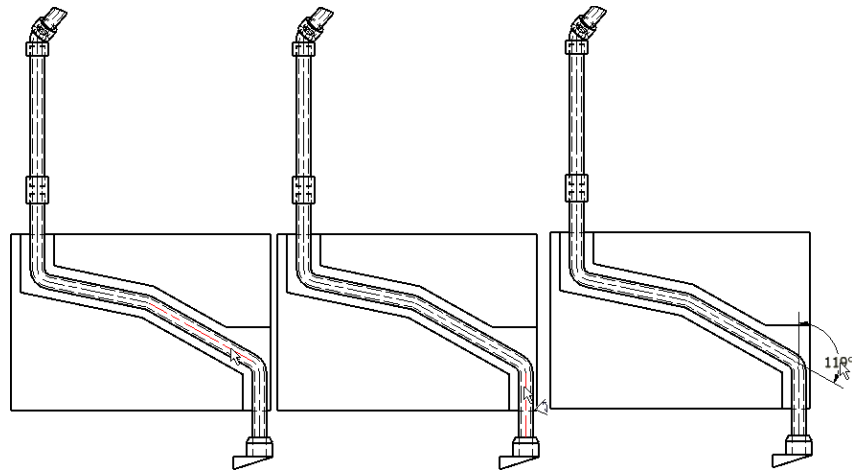
Dimension tools include:

- General Dimension
- Baseline Dimension Set
- Baseline Dimension
- Ordinate Dimension Set
- Ordinate Dimension

In the following exercises, you learn to add general dimensions in the detail view DETAIL A. For more information on dimensioning, refer to the Help.

Add general drawing dimensions

- 1 In the *AirSystemAssy.idw* drawing document, click the Annotate tab.
- 2  On the ribbon, click Annotate tab ► Dimension panel ► Dimension.
- 3 Click the intersection point of two route centerlines and the point on the valve as shown in the following images, and then drag to set the dimension.



- 4 Press ESC to exit the command.
- 5 Save the drawing document.

Create and Export Bills of Material

In a tube and pipe bill of materials (BOM), conduit parts and fittings from the Content Center that make up routes and runs are set to Purchased. Master runs assembly, rigid route sketches, hose splines, and hose assemblies are not distinct line items in the bill of materials so they are set to Phantom.

You can override the default BOM structure to meet your design needs. For instance, if you want to create and export the BOM information for specific routes and runs, set all other routes or runs to the Reference BOM structure. In addition, using the BOM Editor, you can:

- Edit materials.
- Add custom iProperties.
- Enable structured and parts-only BOMs and set view properties accordingly.
- Enable the part number merging and set the merging criteria.
- Set quantity overrides for components.
- Control whether different components with the same part number can be merged into one row. If the merging is enabled, you can create your own list of values to exclude from being merged into one row. The blank part


number is permanently excluded so different components with the blank part number do not merge.

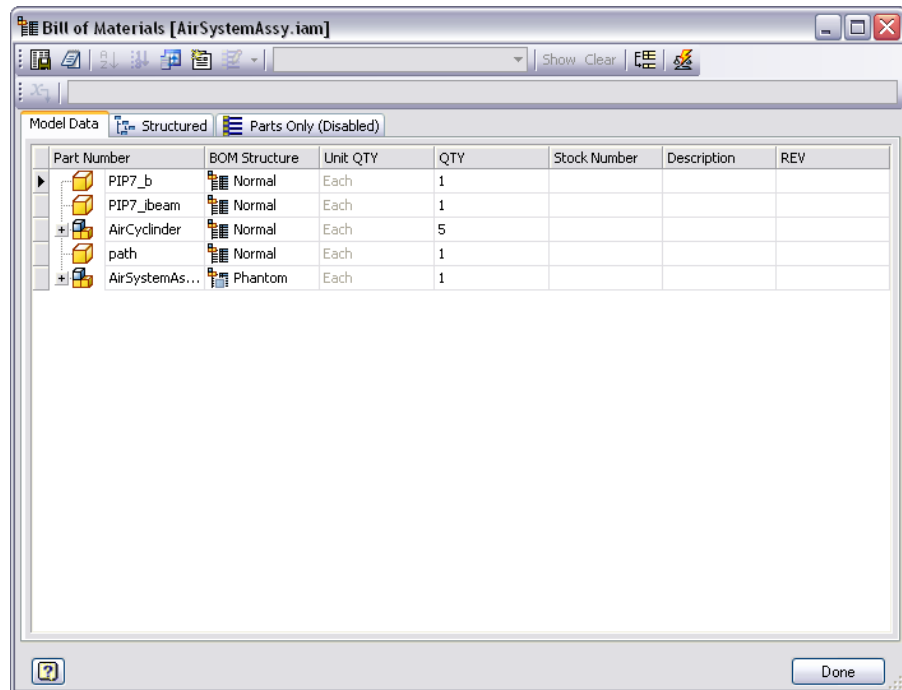
NOTE If you are migrating tube and pipe drawings from R9 or earlier versions, ensure the stock number (Stock Number) and conduit segment length (Base QTY) properties are selected into the BOM table.

Enable the Parts Only View


In this exercise, you create a bill of materials table for the tube and pipe assembly and enable the Parts Only view for the exercises that follow.

Create and export bill of materials for specific runs

- 1 In the *AirSystemAssy.idw* drawing window, expand VIEW1:AirSystemAssy.iam in the Model browser.
- 2  Right-click AirSystemAssy.iam and select the Bill of Materials tool. 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 the View Options tool, and then select Enable BOM View.

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

Add Base QTY and Stock Number

In this exercise, you add the Base QTY and Stock Number properties to the Bill of Materials table so that conduit parts are rolled up correctly.

Add Tube & Pipe-specific properties to the Bill of Materials table

1 In the Bill of Materials dialog box, verify on the Model Data tab whether the Base QTY and Stock Number columns are included.



If they are not included, click the Choose Columns tool on the toolbar. In the Customization dialog box, click and drag the needed columns to the table.

NOTE You can also remove unwanted columns using the Customization dialog box.

- 2 Close the Customization dialog box.
- 3 Click Done.
- 4 Save the drawing document.

Create and Export Bill of Materials


As with bill of materials (BOM) in drawing documents, the BOM Structure property defines the status of each component in the BOM table: Normal, Phantom, Reference, Purchased, and Inseparable. Tube and pipe components typically follow the default BOM structure. Conduit parts and fittings from the Content Center that make up routes and runs are set to Purchased. The master runs assembly, rigid route sketches, hose splines, and hose assemblies do not have distinct line items in the bill of materials so they are set to Phantom.

However, you can override the default BOM structure to meet specific design needs. For instance, you can modify the BOM structures so that you can create and export the BOM information for specific routes and runs. When a component has a BOM Structure of Reference, the BOM treats the component and all its direct and indirect children as if they do not exist. All child components, that are a part of a Reference component, are excluded from quantity, mass, or volume calculations, regardless of their own BOM Structure value.

The *AirSystemAssy.iam* assembly contains two runs. In this exercise, you export the bill of materials information for the AirSystem1:1 run.

NOTE The visibility setting that is specified in the design views does not impact the bill of materials. The components with the visibility off are calculated in the bill of materials table.

Create and export bill of materials for specific runs

- 1 In the Bill of Materials dialog box, Model Data tab, expand AirSystemAssy.Tube and Pipe Runs in the Part Number list.
Verify that both runs are displayed on the Structured tab and all tube and pipe components are displayed on the Parts Only tab. (BOM view must be enabled to view this tab.)
- 2 Click the AirSystem2:1 row, and then select Reference from the BOM Structure list.
- 3 Click the Structured and Parts Only tab to verify the result.
Only the AirSystem1:1 run is displayed on the Structured tab. On the Parts Only tab, the hose part and associated parker fittings in the AirSystem2:1 run disappear.
- 4  To export the bill of materials, click the Export Bill of Materials tool on the toolbar.
- 5 Select either the Structured or Parts Only view to export, and then save the file to your local disk.
- 6 In the Bill of Materials dialog box, click Done.
- 7 Save the drawing document.


Create Parts Lists

If Base QTY and Stock Number properties are not selected into the parts lists, parts lists do not include those properties the first time you create them.

Keep in mind that the drawing manager always groups parts using the Part Number property even if you remove it from the parts list. Each conduit part in the current version of Autodesk Inventor has a unique part number:

In this exercise, you create a parts list for the base view (VIEW1), use the Column Chooser tool to add the Base QTY property to the parts lists. The Stock Number property is already included in the default parts list style.

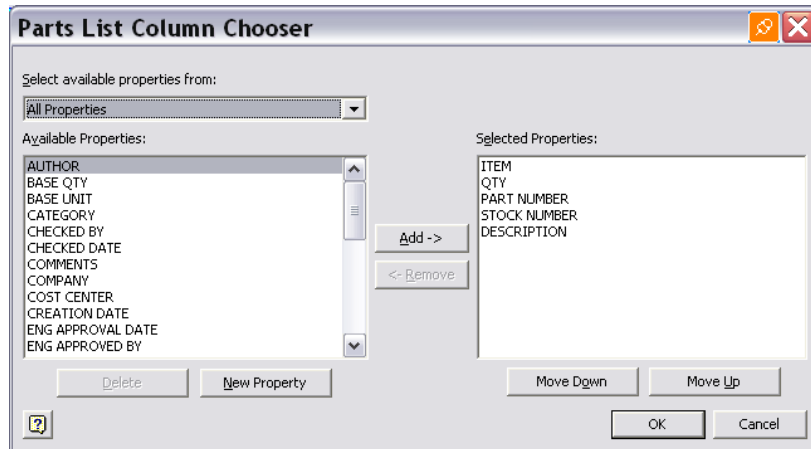
Add Base QTY and Stock Number to a parts list

- 1  On the ribbon, click Annotate tab ► Table panel ► Parts List tool to create a Parts Only parts list for the base view (VIEW1).

- 2 In the graphics window or Model browser, right-click the parts list and select Edit Parts List.



- 3 In the Edit Parts List dialog box, click the Column Chooser tool.
The Parts List Column Chooser dialog box displays. By default, the STOCK NUMBER property is in the Selected Properties list.



- 4 In the Available Properties list, select BASE QTY, and then click Add.
- 5 Click OK.
Both BASE QTY and STOCK NUMBER columns are displayed in the Edit Parts List dialog box.



- 6 Optionally, use the Group Settings tool to group parts list rows based on the BASE QTY and STOCK NUMBER.
- 7 In the Edit Parts List dialog box, click OK
The parts list automatically updates.
- 8 Save the drawing document.

NOTE Using the Style Editor tool, you can add the BASE QTY and STOCK NUMBER properties to the parts list style. For detailed instructions on how to use the Style Editor tool, refer to the Help.

Annotate Drawings with Piping Styles

You may want to use information that is defined in tube and pipe styles to annotate tube and pipe drawings. You can start to define sketched symbols, and then use the Text tool (highlighted below) on the Sketch tab to insert property references in text. When property values change, text that contains the property updates with the new values. Pause the cursor over the images to view the tooltip.

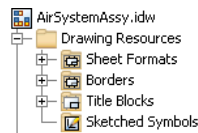


Drawing Sketch panel bar

In this exercise, you create a parts list for the base view (VIEW1), use the Column Chooser tool to add the Base QTY property to the parts lists. The Stock Number property is already included in the default parts list style.

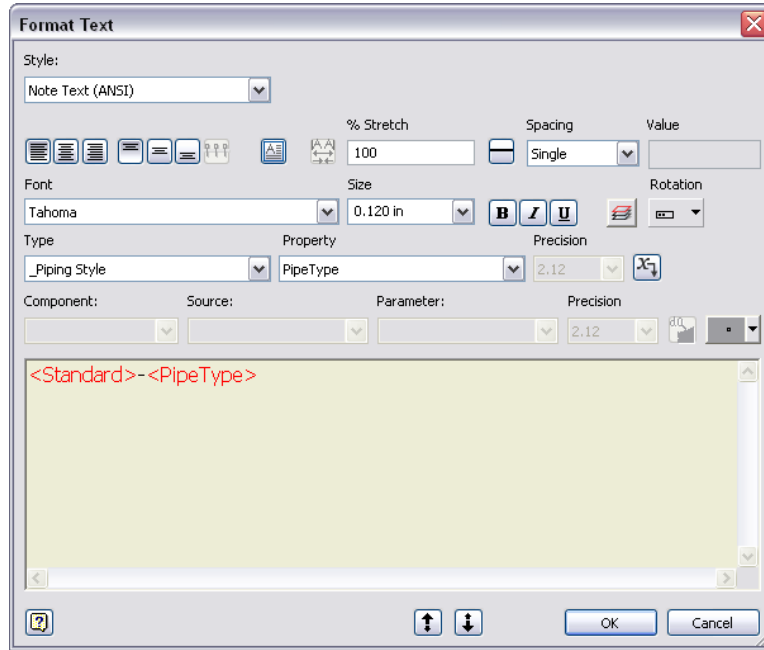
Create sketched symbols with piping style data and annotate drawings

- 1 In the *AirSystemAssy.idw* drawing window, expand Drawing Resources.

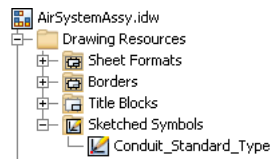



- 2 Right-click Sketched Symbols and select Define New Symbol.
The drawing sketch environment is activated.
- 3 **A** On the ribbon, click Sketch tab ► Draw panel ► Text, and then click a location in the graphics window to place the first property.
The Format Text dialog box is displayed.
- 4 Use the default text style and select _Piping Style from the Type list.
The Property option is activated. All piping style properties available in the Property list are from tube and pipe styles.
- 5 **X** Select a specific piping style property from the Property list and click the Add Text Parameter tool, for instance, Standard.

- 6 Repeat this step to add more piping style properties to define the text symbol. To separate properties, type the needed separator from the keyboard before adding a new property.
In this exercise, you select the Standard and PipeType properties with a separator as shown in the following image.



- 7 Accept the other defaults and click OK.
A string of all selected piping style properties is previewed in the location you clicked in Step 3.
- 8 Right-click in the graphics window and select Done.
- 9 Optionally, to edit the property set, move your cursor over the preview string, right-click and select Edit Text.
- 10 Right-click again and select Save Sketched Symbol. Enter the name in the Sketched Symbol dialog box and click Save. In this exercise, enter the name *Conduit_Standard_Type*.
The new sketched symbol is added to the Model browser, Drawing Resources, Sketched Symbol list. The drawing annotation environment is activated.



- 11  To place the symbol to annotate tube and pipe parts, click Annotate tab ► Symbols panel ► User, pick the desired symbol, and then pick an edge on a tube or pipe part.
- 12 Save the drawing document.

NOTE When placing symbols to annotate drawings, ensure that you pick an edge on the conduit part or fitting while placing a custom piping style sketched symbol. To end, right-click and select Continue. Right-click again and select Done.

Cable and Harness

Part 2 of this manual presents the getting started information for Cable and Harness in Autodesk Inventor® Routed Systems. This add-in provides tools for creating and manipulating three-dimensional wire harnesses in the context of a standard Autodesk Inventor® assembly.

Get Started with Cable and Harness

10

This chapter provides basic information and exercises to help you get started with Cable and Harness.

You also learn how to identify and label connection points on an electrical component and set properties for the pins and the part.

About Cable and Harness

Cable and Harness provides tools for creating and manipulating three-dimensional wire harnesses in a standard Autodesk Inventor® assembly.

Prerequisites

We assume that you have a working knowledge of the Autodesk Inventor interface and tools. If you do not, use the integrated Help system for access to online documentation including advanced topics, tutorials, and skill builders, and complete the exercises in the *Autodesk Inventor Getting Started* manual.

At a minimum, we recommend that you understand how to:

- Use the assembly, part modeling, sketch, and drawing environments and browsers.
- Place and constrain parts in an assembly.
- Edit a component in place.
- Create, constrain, and manipulate work points and work features.

- Create and annotate basic drawings.
- Set color styles and use the Style Editor.
- Use Content Center.

Be more productive with Autodesk® software. Get trained at an Autodesk Authorized Training Center (ATC®) with hands-on, instructor-led classes to help you get the most from your Autodesk products. Enhance your productivity with proven training from over 1,400 ATC sites in more than 75 countries. For more information about training centers, visit the online ATC locator at www.autodesk.com/atc.

We also recommend that you have a working knowledge of Microsoft® Windows NT® 4.0, Windows® 2000, Windows® XP, or Windows Vista® and a working knowledge of concepts for setting up, connecting, and routing wires through mechanical assembly designs.

Back Up Tutorial Data Files

For each exercise in this section, you use files that contain the example geometry or parts for that task. These files are included in the *Tutorial Files* directory for each application. For Cable and Harness, the files are located in the *Tutorial Files\Cable & Harness* folder where you installed Autodesk Inventor. Before you begin the exercises, back up the files so the originals are always available. You can revert to these files if you make any mistakes during the exercises, or if you would like to repeat an exercise.

Back up the tutorial files

- 1 In the folder where Autodesk Inventor is installed, go to the *Cable & Harness* directory and create a folder called *Exercise_Backup*.

The default installation location is:

Microsoft Windows XP operating system:

- *Program Files\Autodesk\Inventor <version>\Tutorial Files*

Microsoft Vista operating system:

- *Users\login-name\AppData\Local\Autodesk\Inventor <version>\Tutorial Files*


- 2 Select the *Cable & Harness* folder and copy the exercise data into your new folder.

Now you can use the tutorial files as you work through the exercises in this book.

Keep any files you create for an exercise in the *Tutorial Files\Cable & Harness* directory to avoid the possibility of file resolution problems.

Set Up Projects For Exercises



For the exercises, click  ► Manage ► Projects and then browse to and select the *EnclosureAssembly.ipj* as the active project. The default installation location for this project is:

Microsoft Windows XP operating system:

- *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness*

Windows Vista operating system:

- *Users\login-name\AppData\Local\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness*

To add connectors from the Content Center, use the Configure Content Center Libraries tool to configure libraries. Ensure that you have logged into the Autodesk Vault server and the needed libraries are ready on the server. The connector content is included in the Routed Systems library.

Help

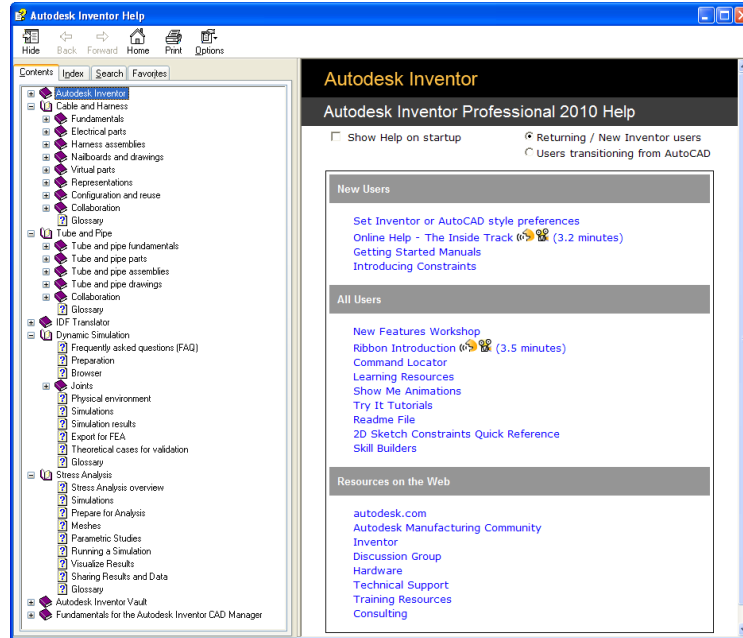
The Help system provides information about every feature in Cable and Harness as well as standard features.



- 1 To access Help, click the Help button in the InfoCenter above the Ribbon. Then, click the Contents tab to display the table of contents if it is not already displayed.

The Help home page provides quick access to various help components such as: Skill Builders, Show Me animations, Advanced Productivity topics, and Index and Search functions.

- 2 Under the Contents tab, expand Cable and Harness to display and examine the related Help topics.



Work in Autodesk Inventor Installations

If Cable & Harness is not installed on the system, viewing the harness data, the harness subassembly, and the data it contains is read-only. The geometry of the harness components is visible through Autodesk Inventor, but the components cannot be edited, and new cable and harness components cannot be added. However, you can edit electrical parts, and move pins, but electrical part properties are not available.

About Electrical Parts

Electrical parts are normal Autodesk Inventor parts or iParts with extended properties and one or more defined connection points, known as pins. Electrical parts are the only harness components that are not created in the context of a harness assembly. Instead, electrical parts are created by editing normal Autodesk Inventor parts. While editing the parts you identify the special work points called pins, provide required properties, and optionally add additional custom properties. You can define and edit the pins individually or as a group. Once defined, the electrical part is placed in an assembly.

The part used can be fully modeled or a simple representation of the part, as long as there is some geometry to designate the connection points, or pins. For example, the part could be a simple plane with work points representing the pins. For a group of pins, the part could be a simple plane with a single point representing the start location for the group. These pins are the attach points for the wires in the harness assembly.

You can also create, author, and publish your own connectors or place generic connectors from the Cable and Harness ► Connectors category of the Content Center.

Workflow for Electrical Parts

The harness workflow begins with electrical parts. The following are the basic steps to create an electrical part and place it in an assembly.

Create an electrical part and place it in an assembly

- 1 Edit an Autodesk Inventor part.
- 2 Add individual or group pin definitions, each with a unique name and additional properties, if appropriate.
- 3 Optionally, author and publish the connector part to Content Center.
- 4 Optionally, provide a placeholder reference designator (RefDes) or generic value for that electrical part type in the part file.
Do it once for each electrical part.
- 5 Place the part in an assembly.
- 6 Assign the reference designator for an individual occurrence or group.
The reference designator for the individual occurrence or group is required

in each harness assembly that uses the electrical part, not once on each part occurrence.

Create Electrical Parts

Use the Harness panel on the Model tab to add pins and the reference designator placeholder. The reference designator and pins are specified while editing the part file or editing the part in place. Pause the cursor over the commands to view the tooltips.

Place Pins and Define Pin-level Properties

A connector can contain one or more pins. Valid geometry for pin selections includes both associative and non associative points. The points you select determine whether the pins are updated if the geometry to which they are associated changes.

Non-associative points are arbitrary points on any face. They do not update if the geometry changes. Associative points, which update as the geometry is changed, include any one of the following:


- Existing work points
- Center points on any circular component such as a face, a hole, and cylindrical cuts or arc edges
- Existing sketch points
- Model vertices

Each pin must have a unique name. By default, each individual pin name is a sequential number starting with 1. For pin groups, you provide a prefix letter and start number, and then select the naming scheme that you need. Each pin name within a part must be unique. When the browser name changes for an individual pin, the pin name updates. The reverse is also true. To change naming for a pin group, delete the pin group and recreate it with the naming you need.

In this exercise, you edit the part in place to add individual pins and a placeholder reference designator to a part. For your convenience, the connectors are already placed in the assembly.

Start by opening an assembly in the default project.

Open the assembly

- 1 Open Autodesk Inventor if it is not already open, and click 
 ➤ Manage ➤ Projects.
- 2 On the Projects dialog box, verify the *Enclosure_Assembly.ipj* project is the active project, and then click Done.


If needed, browse to the project. The default location is:

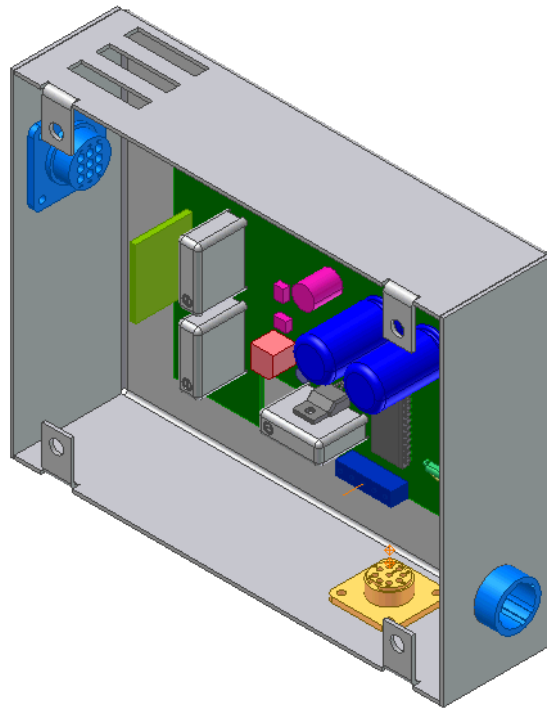
Microsoft Windows XP operating system:

- *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness*

Windows Vista operating system:

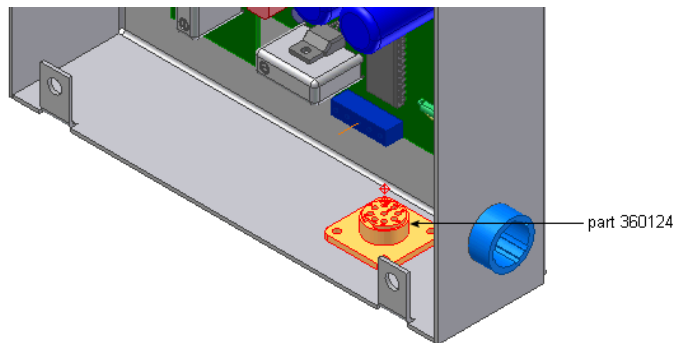
- *Users\login-name\AppData\Local\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness*

- 3 Click  ➤ Open.
- 4 Select the file named *Enclosure_Assembly.iam* and click Open.
The assembly looks like the following image.



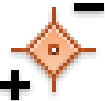
Add pins in the assembly

- 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 select the part for editing.

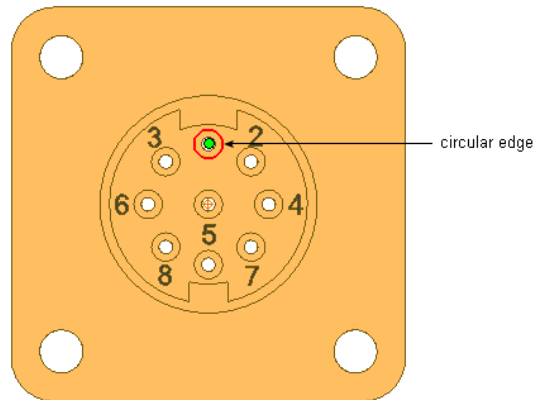
- In the Model browser, delete work points 1 through 9. You re-create these points in the next steps.



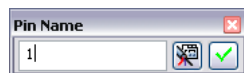
- On the ribbon, click Model tab ► Harness panel ► Pin.

NOTE If the Harness panel is not visible, right-click on the Ribbon, click Panels, and select Harness.

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



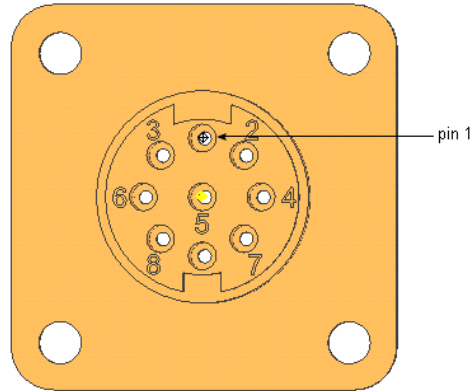
- On the Pin Name dialog box, use the default pin name of 1.
- Click the Harness Properties button to display the entire Pin Properties dialog box.



- Click the Custom tab to see where custom properties are added. For this exercise, do not add any additional properties for the pin.

- 11 Click OK.

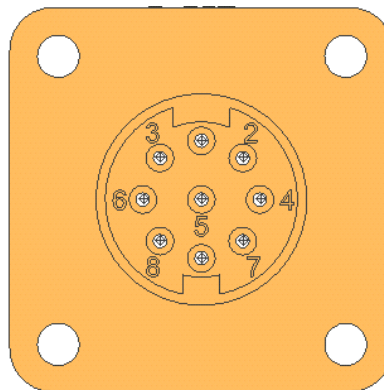
The work point representing the pin appears for pin 1.



- 12 Define a pin for hole 2:

- Highlight the circular edge.
- Accept the Pin Name of 2.
- Click the check mark button.

- 13 Repeat the steps to define a pin at each hole. 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.



pins defined at each hole

- 14 Right-click, and then select Done to exit Place Pin mode.

Note that 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.

Set Part Properties

Specific property data must be added to a part to provide a complete electrical definition. These properties are also visible on the part occurrence in the assembly.

The part name and part number are automatically set based on the part file name and the Autodesk Inventor part number. If appropriate, you can set a value for the placeholder reference designator property.

The reference designator, or RefDes, is a unique identifier that maps the part to the schematic design. Typically a placeholder identifier, such as U?, is added in the part environment, and then a specific identifier is added for each occurrence of the part in the context of the assembly. For example, if a certain RS232 connector occurs in an assembly multiple times, each occurrence must have a unique identifier, such as U1, U2, and U3.

Additional custom properties can also be added to the part. Custom properties provide specific information to downstream processes such as reporting. Custom properties such as the vendor and vendor part number, often come from the data book for the component.

To save time on data entry and reduce entry errors, consider creating a template for electrical parts that contains placeholders for commonly used properties.

Add RefDes Placeholders


In this exercise, you add a placeholder RefDes to the electrical part.

Add a RefDes placeholder

- 1 With nothing selected, select Model tab ► Harness panel ► Properties. Alternatively, right-click the part in the browser, and then select Harness Properties from the context menu.
- 2 On the General tab of the Part properties dialog box, enter a reference designator (RefDes). In this case, enter the placeholder *U?*.
- 3 Click OK on the dialog box, and click OK on the message box if it appears. If the message box does appear, select to not show the message box again.

You also need to click OK on the dialog box again after the message box closes.

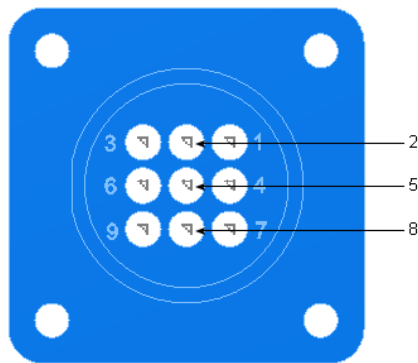


- 4 Click  ► Save and click OK in the dialog box.
- 5 Right-click and select Finish Edit.
In a later exercise, you set the RefDes value for the occurrences.

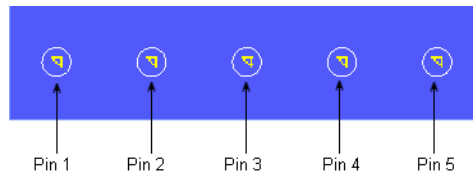
Practice Your Skills

Use the skills you just learned to add pins and a RefDes placeholder to the other electrical parts.

- 1 Edit the part 360575:1, and add pins at each hole. Name the pins to match their number label.



- 2 Add the reference designator *U?*.
- 3 Right-click and select Finish Edit. Alternatively, click Return on the Ribbon.
- 4 In the browser, expand the PCB subassembly to locate and edit part LTP.
- 5 Add the pins as shown. Pin 1 is the pin closest to part 360124.



- 6 Specify the reference designator *J?*.
- 7 Right-click and select Finish Edit. Alternatively, click Return on the Ribbon.
- 8 In the browser, edit part 360575:2 and notice how this occurrence inherits the pins set in 360575:1. You only need to pin the part once and all occurrences inherit the pins.
- 9 Return to the assembly.



- 10 Click  ► Save and click OK.

Modify Pinned Parts

Once an electrical part is defined, you can modify it in several ways.

- Add or modify electrical properties on both the parts and pins.
- Modify pins or a pin group using standard Autodesk Inventor work point operations (Ground, 3D Move/Rotate, Redefine Feature).
- Delete selected pins or a group of pins.

NOTE Any wires attached to a deleted pin dangle. To reattach a dangling wire move the wire to another pin using Edit Wire from the context menu.

- Change the start location, orientation, and spacing of a group of pins.
- Rename individual pins.

NOTE Delete the pin group and make a new one to change naming or pin or row number for a group.

When modifying pins using Redefine Feature, you create the geometry used to define the individual pin or the start location for a group of pins. You can

place or project work points onto part faces, linear edges, or an arc or circle. Work points can also be constrained to the center points of arcs, circles, and ellipses.

To modify library parts ensure that you have edit and publish permission to the Content Center.

Add Custom Properties to Parts

In this exercise, you add a custom property to a part. Each custom property requires a data type. Each data type, except for text, has a default unit associated with it.

Add custom properties to a part

- 1 In-place edit part 360124.
- 2 In the browser, select 360124:1, right-click, and then select Harness Properties from the context menu.
- 3 Click the Custom tab, and enter the values as follows:
Name: Vendor
Type: Text
Value: AMP
- 4 Click the Add button, and then click OK.
The property is added to the list, and the dialog box is closed.
- 5 Save the part file and return to the assembly.
- 6 Repeat the steps to add the same Vendor properties to the electrical parts LTP and 360575:1.

Place Electrical Parts

You place and constrain electrical parts in an assembly just as you place and constrain standard Autodesk Inventor parts. Once placed, with a harness assembly added, you set a specific reference designator for each occurrence of that part in the context of each harness assembly.

Electrical parts can be placed anywhere below a harness assembly or parent assembly. They can be placed outside of the harness assembly or demoted into the harness assembly, depending on how you prefer to document the

harness assembly. Connectors that already reside within subassemblies can remain there.

If the connector is physically part of the harness, like a mating connector, consider adding it directly to the appropriate harness assembly. An object such as a terminal block that is not part of the harness, may not belong in the harness assembly.

If reusing a top-level assembly that contains a harness assembly, we recommend that you demote the electrical parts into the harness assembly. From within Cable and Harness, use *Save Copy As* to copy the top-level assembly and its contents (the harness assembly, other subassemblies, and so on) and reuse it in another assembly.

In this assembly, the four electrical parts you created earlier were placed for you. You set the reference designator for each part occurrence after the harness assembly is added in the next chapter.

When placing and constraining mating connectors for a harness assembly, remember the following information:

- When the connectors reside in the harness assembly, they can be constrained to objects in the top-level assembly only while editing the top-level assembly. The connectors cannot be constrained while in-place editing the harness assembly.
- If you are unsure whether to place mating connectors as children of the top-level assembly or in the harness assembly in your initial design, it is good practice to place the connectors in the top-level assembly without constraints. Using this method, you can demote the connectors into the harness assembly before the harness is created without losing constraints. Any RefDes information added to a connector at the time they are demoted is lost and must be edited to reassign the RefDes in the harness. Any wires or cables attached to connectors at the time they are demoted dangle and must be edited to reassign the wires to the connector pins.
- Constraints placed in the harness assembly provide more flexibility for individual components within the harness to adapt to assembly changes. They do not adapt to changes as a rigid body. For example, with a top-level assembly, if you constrain to a subassembly, all the parts in the subassembly move as one rigid body. With a harness assembly, you can constrain all connectors in the subassembly to different locations and they adapt independently of one another. They do not move as a rigid body.

Work With Harness Assemblies

11

This chapter provides basic information about the tools added by the cable and harness environment. It also tells how to create a harness assembly and set properties.

About Working in Harness Assemblies

The Cable and Harness software includes features for defining electrical parts, inserting wires, cables, and ribbon cables from a library, routing wires through segment paths, inserting splices, and attaching virtual parts. You can generate reports for several common report types, or configure them to your specific needs. The harness assembly information can also be represented in drawings or nailboard drawings. It can also be saved to an *.xml* format.

With the cable and harness tools you can:

- Create or edit harness assembly files.
- Create electrical parts.
- Author and publish connector parts to the Content Center.
- Retrieve generic or published connector parts from the Content Center.
- Edit and retrieve harness object definitions from a library.
- Import electrical connectivity wire lists (including cable wires) using either *.csv* or *.xml* file formats.
- Allocate space by defining possible wire, cable, and ribbon cable paths through the assembly.

- Route wires and cables through harness segments and automatically calculate lengths and bundle diameters.
- Configure and generate reports of the harness assembly.
- Insert and arrange splices, attach virtual parts, and make other modifications to the harness design.
- Create accurate 2D harness documentation with intelligent properties and dimensions that automatically update as the 3D design changes.
- Use the browser to organize and edit electrical parts, pins, wires, cables, ribbon cables, splices, virtual parts, and segments. You can also change visibility on all harness objects through the browser, except for virtual parts.
- Optionally, copy and reuse the completed harness in another location.

WARNING Do not perform operations specific to Autodesk Inventor® on segments, wires, cables, and ribbon cables. It can cause problems with operations in cable and harness. For example, do not use Autodesk Inventor operations such as extrusions, sweeps, and so on, to change harness objects.

Cable and Harness Environment

To access the Cable and Harness environment, you create or edit a harness assembly. Features specific to Cable and Harness are available only in this environment. Other features, such as a Nailboard, are only available once a valid harness assembly is created.

Some of the features specific to the Cable and Harness environment include:

Cable and Harness tab	Panels and commands for the cable and harness environment.
Cable and Harness browser items	Contained in the Model browser, the Cable and Harness browser items include the contents of one or more harness assemblies in a hierarchy. The harness assemblies act as containers for all the objects created or placed in that particular harness.
Segment, wire, ribbon cable display tools	Switches between rendered and centerline display for wires, segments, and ribbon cables.

Harness panel	Activated by editing normal Autodesk Inventor parts, this panel contains the command set for creating electrical parts.
Connector command	Activated by editing normal Autodesk Inventor parts, use this command to author connectors.
Nailboard tab	Activated by selecting the Nailboard command from the Cable and Harness tab. This tab contains the command set for manipulating and annotating a 2D representation of the 3D harness.
Nailboard View command	Use the Nailboard View command, in the standard Drawing environment, to create a nailboard from an open drawing.
Connector View command	Use the Connector View command to add connectors to your nailboard drawing.

Cable and Harness also adds electrical data types and units unique to the electrical domain, and makes connectors available from the Content Center Library.

Create Harness Assemblies

When an assembly file is first opened in Autodesk Inventor, the assembly environment is displayed. To begin designing a harness in your assembly model, create the harness assembly.

A standard Autodesk Inventor assembly can contain multiple harness assemblies.

Workflow for Harness Components

The following are the basic steps to place and create harness objects such as wires, cables, ribbon cables, and segments in an assembly:

Place and create harness objects

- 1 Create electrical (pinned) parts, adding properties as needed.
- 2 Open the Autodesk Inventor assembly file in which the harness will be created or exists.
- 3 Optionally, author and publish electrical parts to the Content Center.
- 4 Place and constrain electrical parts in the assembly.

- 5 Create the harness assembly.
- 6 Insert wires, cables, and ribbon cables into the harness assembly, and then add properties as needed.
- 7 Create segments that define the possible wire and cable paths through the assembly.
- 8 Add properties to the segments as needed.
- 9 Route wires and cables through selected segments.
- 10 Insert splices, add wire and cable points, add ribbon cable points and folds, adjust ribbon cable twist, attach virtual parts, and make other modifications to the harness.
- 11 Create a nailboard of the harness assembly and generate parts list and bill of materials information.

To create the Harness assembly, click Assemble tab ► Begin panel ► Cable and Harness.



Create Harness command

In the Create Harness dialog box, provide a unique name and location for the harness subassembly. By default, the file is named *<top level assembly>.Harness<number>.iam* (where number is a sequential number that increments with each harness assembly) and is saved to the location of the open assembly file, along with the corresponding harness part. You can provide a different name and location if appropriate. You can also change the name to something harness specific or descriptive.

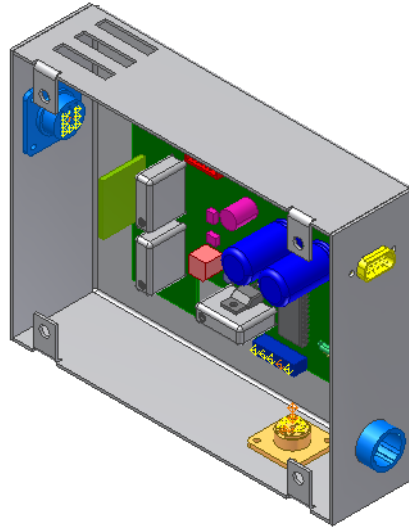
Once you provide the name and location, the system adds a harness assembly to the browser. It is added to the browser along with other placed components and is arranged in the order it is added to the assembly. You can locate the harness anywhere in the assembly hierarchy, except in another harness assembly. For example, you can nest the harness assembly in another standard Autodesk Inventor assembly, but not in another harness assembly. When you edit a harness assembly, you can connect only to harness objects that share the same parent assembly as that harness assembly.

In this exercise, you open an existing assembly and prepare to wire it. You also become familiar with the features included in the cable and harness design

environment, including the tab, display tools, and browser. A ribbon cable harness assembly was already added for you.

Create a harness assembly

- 1 If not already open, navigate to and select the file *Enclosure_Assembly.iam*. The assembly is displayed as shown.



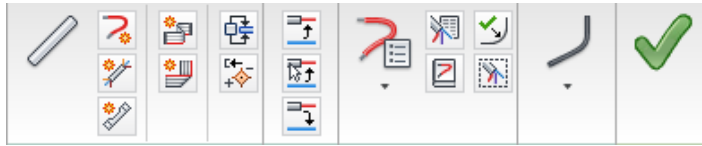
- 2 On the ribbon, click Assemble tab ► Begin panel ► Cable and Harness.
- 3 On the Create Harness dialog box, enter the name for the harness assembly as shown, and accept the default location (example is for Windows® XP)

Harness Assembly File Name: *Harness Assembly1.iam*

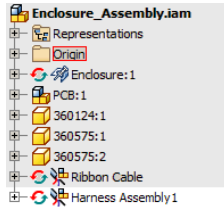
Harness Assembly File Location: *Program Files\Autodesk\Inventor <version>\Cable & Harness\Tutorial Files\Enclosure_Assembly\AIP\Cable and Harness*

- 4 Click OK.

The system adds the harness assembly to the browser and activates the cable and harness environment. The following illustration shows the Cable & Harness tab. Pause the cursor over the images to view the tooltips.



- 5 Locate the new harness assembly in the browser hierarchy.




NOTE Clicking the Create Harness tool again adds a second harness assembly after the existing one at the same level.

- 6 Examine the Visibility panel of the Cable and Harness tab to see the display settings for wires, cables, ribbon cables, and segments. Click the arrow to display the list.

The display for these harness objects can be changed at any time. The display for all objects of a selected type in a selected harness assembly can also be changed.



- 7 Activate the top-level assembly and click  Save.

- 8 Click OK.

The harness assembly is saved using the name and location indicated previously.

Use the Cable and Harness Browser

All harness objects added to an assembly file are automatically contained in the harness assembly. The objects include wires, cables, cable wires, ribbon cables, segments, splices, virtual parts, and optionally electrical parts and pins.

Each harness assembly contains an Origin and a Representations folder and a single harness part. Although harness assemblies respond to representations in the parent assembly, they do not respond as expected to representations created at the harness assembly. For best results, create representations at the parent assembly level only.

The harness part contains one folder for each of the harness objects that make up the harness. Each segment has a work point for every point used to shape the segment. Ribbon cables contain work points and folds used to shape the ribbon cable. The Virtual Parts folder contains a folder for each type of virtual part in the harness.

NOTE Promoting a harness part from its parent assembly corrupts the cable and harness data. In this case, close the assembly without saving. Demoting the harness back to the subassembly does not fix the data.

Set Properties for Harness Components

Every harness component contains a set of properties. These properties are used for the documentation you produce, such as reports and drawings. Properties are also used to check specific aspects of the design, such as bend radius and bundle diameters.

Properties are added at different levels within the assembly. They can be added to the source component in the library as well as to any occurrences of that component within the assembly.

When you create a harness object, some properties are required, such as the Pin Name for an electrical part occurrence, or the Wire ID for a wire. You can decide on which other properties to set.

The documentation you produce for downstream consumers drives the properties you set. Some properties, such as length or adjusted length for a harness segment are derived from the placement or settings of the related component. These properties cannot be changed, but can be viewed and used in the documentation.

You can set and view properties when the component is active in Cable and Harness. Some properties can also be accessed from other objects. For example,

from the Wire Properties dialog box, you can view and modify properties on the parts and pins to which it is connected.

You can also use the property dialog boxes to add and remove virtual parts. Search for “virtual parts” in the Help index for more details.

Customize Properties

The Custom tab of the Properties dialog box may contain predefined properties. These properties are available in the property Name list, and have a preset name and data type. You can add them to the object individually and set the value.

If you need a property for a harness component that is not predefined by Cable and Harness, you can create a custom property. To create a custom property, you name the property and set its data type and value. Once the property is fully defined, it is added to the property Name list on the Custom tab. The property is also available in the Name list within the same part file or harness assembly for use on other harness objects.

For more information about setting data types for custom properties, refer to the Help.

Set Occurrence Properties

Harness objects have general properties that are stored in their source files, and unique occurrence properties that are stored within the assembly in which they are placed. Harness component occurrences get their properties from the source component. For example, when an electrical (pinned) part is placed in the harness assembly, its properties come from the part definition. When specific properties are needed for an occurrence, you can override certain component-level property values with an occurrence property value.

In the property list on the Custom tab, properties added to an occurrence are represented with white backgrounds. Properties listed with a dark background indicate a source-level property. For example, on the Wire Properties dialog box, for an occurrence wire, the properties added at the library level are displayed with a dark background.

Assign Occurrence Reference Designators

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

NOTE Add occurrence properties to electrical parts only after they are in their proper assembly location. Occurrence properties are lost when electrical parts are promoted or demoted, and must be reassigned.

Set a reference designator for an electrical part occurrence


- 1 In the browser, double-click Harness Assembly1, if not already active.
- 2 Right-click the part name 360124, and then select Harness Properties from the context menu.

NOTE You can select the parts, but the browser has a gray background.

- 3 On the Part Properties dialog box, set a RefDes of *U3* for the occurrence, and then click OK.
- 4 Repeat the steps 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 then set a RefDes of *J12*.

The generic placeholder RefDes for each part is overridden at the occurrence level with the unique value. The RefDes on each part occurrence is required.

NOTE Once the harness assembly is added, you can also assign terminals, seals, and plugs to electrical part pins.

- 7 Activate the top-level assembly and click  Save.
- 8 Click OK.

Use the Cable and Harness Library

12

All harness objects in Cable & Harness come from a single library. In this chapter, you learn how to navigate in the library and use the Cable and Harness Library tool.

About the Cable and Harness Library

The cable and harness library contains the list of standard library definitions used to create occurrences in a harness assembly. It contains definitions for the following types:

- Wire
- Cable
- Splice
- Raw ribbon cable
- Virtual parts: Label, Loom, Plug, Seal, Terminal, and custom types

These definitions do not have a physical representation, and exist only in the cable and harness library file. You select occurrences from this library list to insert them in the harness assembly.

You can use any standard definition in the library or add your own definitions. You can also modify and remove existing definitions and add new object types. To add a large number of definitions, import the harness objects using an import file.

With the cable and harness library tools you can:

- Add new definitions and object types.
- Modify, copy, and delete existing definitions.
- Import and export definitions.
- Explore the harness objects in the list.
- View the basic properties for a selected harness object such as a wire or cable.
- View the total number of a certain harness object type in the library.

NOTE If a cable and harness assembly that was created in Version 8 of Cable and Harness is opened in the current version, the data format of the library must be migrated to the current version which includes cables. Outdated wire definitions must also be removed from the existing library. If needed, see “migration, Cable and Harness Library” in the Help index for more details about migration options.

Locate the Cable and Harness Library File

The cable and harness library file contains all the library data. By default, the file is named *Cable&HarnessDefaultLibrary.iwl* and is located in the *Design Data\Cable & Harness* directory as set for the project. You can change this location using the File Locations tab on the Harness Settings dialog box.

Typically, one cable and harness library file is created and stored on a server for all harness assemblies to reference. If the Cable and Harness Library is accessed by multiple users, it must be placed in a shared location. You can set a different cable and harness library file and location for each harness assembly, although only one is recommended.

The first person to open the Cable and Harness Library can add, edit, and remove harness objects. All others attempting to access the Cable and Harness Library, are notified that the library is being edited and it cannot be accessed. In this case, the Cable and Harness library is read-only, but you can still insert any harness object listed in the Cable and Harness Library into the harness assembly. For all harness objects but cables and raw ribbon cables, the default library definition is used when the cable and harness library cannot be found. The default library definitions cannot be modified.

Each time you open the Cable and Harness Library, the system determines if changes were made to the library data that require an update. If updates are required, the changes are immediately available in the active harness assembly.

In this exercise, you access and view the name and location for the current cable and harness library file.

View the location of the Cable&HarnessDefaultLibrary.iwl file

- 1 In the browser, double-click Harness Assembly1, if not already active.
- 2 In the browser, right-click Harness Assembly1, and then select Harness Settings from the context menu.
- 3 On the Harness Settings dialog box, click the File Locations tab.

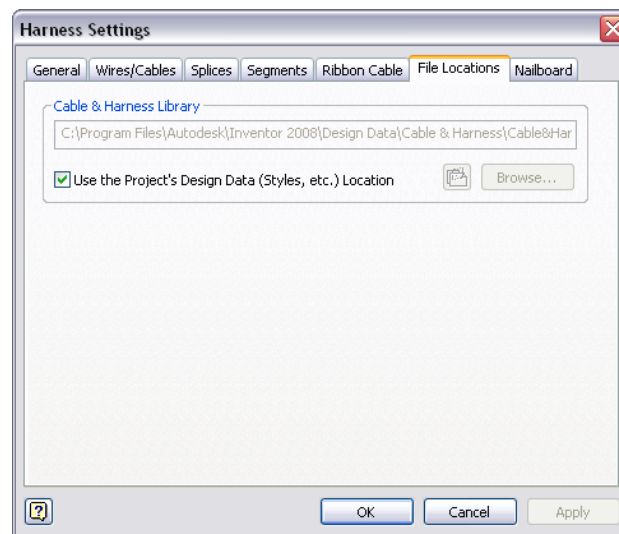
This harness assembly references the default cable and harness library file as set for the Design Data of the project. The default file and location is:

Microsoft® Windows® XP operating system:

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

Windows Vista® operating system:


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



NOTE Set the location for the Design Data of the project by clicking the Tools tab ► Options panel ► Application Options ► File tab, and entering the path under Design Data (Styles, etc.).

- 4 To set the library file to a location other than the default location, clear the Use the Project's Design Data (Styles, etc.) Location and click the Browse button to navigate to and select the file in the needed location.

NOTE To create a new, empty Cable and Harness Library, enter a new file name. If a new name is entered, the cable and harness library file is created in the specified directory.

- 5 If the file location or name is changed, click Update , then click Apply or OK on the dialog box to load the library.
- 6 For this exercise, leave the library in the default location.

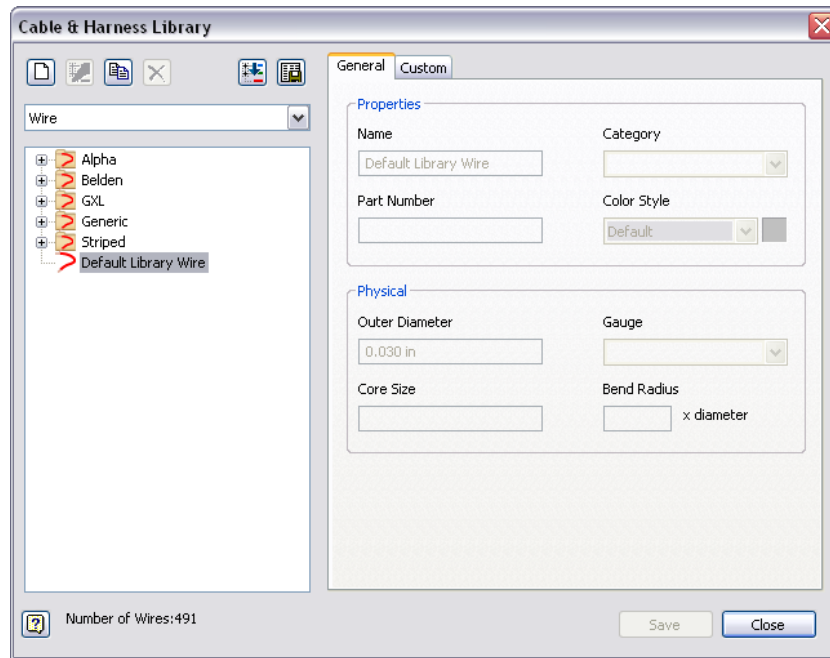
Display the Library Dialog Box

Click Cable and Harness tab ► Manage panel ► Library to display the Cable & Harness Library dialog box.



Cable & Harness Library command

The following illustration shows the Cable & Harness Library dialog box with the Wire object type and Default Library Wire selected.



The features on the Cable & Harness Library dialog box include a drop-down list that includes the harness object types available in the library. The data and format displayed for each object type is similar. You select the object type for the library data to view and use, and then features specific to that object type are displayed.

The Cable & Harness Library includes:

Cable and Harness Library toolbar The tool set for the library that you use to create, edit, copy, and delete individual library wire or library cable definitions. Import and export functions are also provided to bring library data in and out of the library. Pause your cursor over the toolbar to view the tooltips.



List of harness object types Lists available harness object types. Click the arrow to select from the list. The individual objects of that type are displayed in the list of individual harness objects. For User-defined harness objects, select Add New Type and enter a unique type.

Categories and list of harness objects	Lists the contents of the cable and harness library. The library contents include individual harness objects and harness objects in specified categories.
Default library definitions	The definition used by default when the associated library cannot be found. This definition cannot be deleted or modified. There is no default cable or raw ribbon cable.
Number of harness objects	The total number of individual objects in the selected library.
Object Properties tabs	Displays properties for the selected definition and the category in which the selected object is organized. For example, displayed wire properties include color style, part number, outer diameter, gauge, and core size. Displayed cable properties include category, part number, outer diameter, and number of wires.

Adding Library Definitions

You can add library definitions to the library individually or load multiple definitions into the library at one time by importing from a spreadsheet or other list.

Create New Library Definitions

To create new library definitions, specify the required properties. For example, specify the wire name, color, and outer diameter for a new library wire definition. Typically, these required properties are listed on the General tab.

If appropriate, organize new definitions into new or existing categories, assign a part number, and add additional physical properties. Any number of custom properties may also be added.

For example, the library Wire object type has two tabs:

General	Sets required, physical, and optional properties for new definitions such as bend radius.
Custom	Sets new or edits existing custom properties. All custom properties must have a data type.


The library Cable Wire dialog box includes a Wires tab in addition to the General and Custom tabs listed previously:

Wires Lists the wires that make up the cable so you can change the Conductor ID.

In this exercise, you create a library wire and label definition.

Create new library wire and library label definitions



- 1 With the *Enclosure_Assembly.iam* still open, and the harness assembly 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 the New tool.
- 3 On the Library Wire dialog box, General tab, in Properties, specify:
Name: BLU_EE_22
Category: Alpha
Part Number: 60820
Color Style: Blue
- 4 In Physical, enter the following values related to the physical parameters of the wire:
Outer Diameter: 0.0540
Gauge: 22
- 5 Leave Core Size and Bend Radius blank.
- 6 You can click the Custom tab to add additional properties to the wire definition, but it is not required for this exercise.
- 7 Click Save to save the wire to the referenced library.
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.
- 8 Click the arrow and select Label from the object type list.
- 9 Click the New tool.
- 10 On the General tab under Properties, specify:
Name: Wire ID Label

- 11 Accept all other settings under Properties and Physical.
- 12 Under Default Contents, click the arrow to select Wire ID from the Property list, and then click Insert Property.
- 13 Click Save and do not close the dialog box.

Modify Library Wire and Cable Definitions


Using the object types, the listed categories, the lists of individual objects, and the tools in the library, you can select harness objects to copy, modify, or remove.

Copy Library Definitions

In this exercise, you copy a wire that you created.

Copy a library wire definition

- 1 With the Cable & Harness Library dialog box still open, go to the list of wires, and then select the BLU_EE_22 wire that you created.


- 2  Click the Copy tool.

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. You can make up to 9 copies of the same wire before renaming it.

Edit Library Definitions

You can edit any definition in the library except the Default Library definitions such as the Default Library Wire or Default Library Loom. For all harness object types, you can edit the general and custom properties. For cables, you can also add and remove wires from the cable.

Edit a wire definition in the library

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

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_22A

Category: Generic

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

3 Click Save.

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

Delete Library Definitions


You can delete a library definition using the Cable & Harness Library dialog box. If a harness object that is deleted from the library was already added to a harness assembly, any occurrences of that object in the assembly are not deleted. You cannot delete wires used by a cable.

NOTE If a wire or cable with the same name as the deleted one is added back into the Cable and Harness Library before a harness assembly is opened, any occurrences of the wire automatically update with the new definition the next time the harness assembly is updated.

In this exercise, you delete the wire definition you added previously.

Delete a wire definition

1 On the Cable & Harness Library dialog box, select the wire to delete: BLU_EE_22A.

2  Click the Delete tool, 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.

Add Properties to Library Definitions

Use the Edit tool from the Cable & Harness Library dialog box to view, add, or modify library-level properties.

In this exercise, you add custom properties that further define a wire. Use a color property to represent a textured wire. Although textured color styles are supported, they cannot be viewed on wires displayed in the default centerline display mode.

In this exercise, you select a single color wire, and then add the wire color as a property.

Add custom properties to a wire

- 1 On the Cable & Harness Library dialog box, expand the Generic category, and then select the wire named 22AWG-GRN.

- 2 Click the Edit tool.

- 3 Click the Custom tab, and then enter two properties with the values shown. Click Add after entering each property:

Name: Vendor

Type: Text

Value: 5876

Name: Volt

Type: Voltage

Value: 300V

- 4 To modify the existing color property, click the property in the list, and then enter the following values:

Name: Color

Type: Text

Value: Green/Yellow

NOTE In the Help index, go to “wires, striped” for complete directions on creating a two-color, striped wire.

- 5 Click Modify, and then click Save.
- 6 Close the Cable & Harness Library dialog box.

Import and Export Library Data

Cable and Harness Library data can be imported into the library, as well as exported out of the library by specifying two files. A comma-separated value (.csv) data file and a configuration file (.cfg) are used for both processes.

The configuration file describes the input parameters, their associated data types, and corresponding units. The data file contains the library data, which is a description of each harness object imported into or exported from a library.

A blank value is entered by using two consecutive list separators, such as two consecutive commas. Only one list separator is required if the entry is the last one in the row. If the color styles for a harness object are not defined, the default color style is used. Refer to the Help for details on the format and contents of these files.

NOTE You cannot update existing harness objects by importing library data.

Import into the Cable and Harness Library

The workflow for importing definitions into the Cable and Harness Library is the same for all object types, except you must do the import with the appropriate object type selected. The data provided in the files is also different depending on the object you are importing.

Sample import configuration file for library wires:

libwires

C:\libwires.log

Wire Name,Color Style,Outer Diameter,Category,Part Number,Gauge

text,text,length,text,text,text

none,none,inch,none,none,none

Sample import configuration file for library cables:

libcables

C:\libcables.log

Cable Name,Category,Part Number,Outer Diameter,Conductor ID,Wire Name

text,text,text,length,text,text

none,none,none,inch,none,none

Sample import configuration file for library seals:

libseal

C:\libvirtualparts-seals.log

Name,Category,Part Number,Min Outer Diameter,Max Outer Diameter,Diameter Type

text, text, text, thickness, thickness,text

none, none, none, inch, inch, none

Sample import configuration for library splice:

libsplice

C:\libsplice.log

Name, Category,PartNumber,Embedded Length, MyCustom

Splice_L, Size, SP_L_001, 0.3in, 1s

Splice_M, Size, SP_LM001, 0.32in, 12

Splice_LS Size, SP_S_001, 0.31in, 13

Sample import configuration file for library raw ribbon cables.

libribboncables

C:\librawribboncables.log

Name,Category, Part Number, Conductor One Color Style, Ribbon Body Color Style,
Number of Conductors,Pitch,Height,Width,Gauge,Core Size

text,text,text,text,text,text,length,length,length,text,length

none,none,none,none,none,none,inch,inch,inch,none,inch

For this exercise, you import wires. The configuration and data files were already created for you.

Import wires into the library

- 1 Ensure Harness Assembly1 is still active.
- 2 On the Ribbon, click Cable and Harness tab ► Manage panel ► Library.
- 3 Ensure that the Wire object type is selected.
- 4 On the Cable & Harness Library dialog box toolbar, click Import Library

Objects  .

The Cable & Harness Library dialog box remains open during the import.

- 5 On the Import Library Wires dialog box, click the Browse buttons to navigate to the *Tutorial Files\Cable & Harness\Wire Library* folder where you installed Autodesk Inventor, and then open the following files:

Data File (.csv): *LibraryGXLWires.csv*

Configuration File (.cfg): *WireLibrary.cfg*

NOTE The files listed previously use the comma delimiter, which is commonly used in the United States to separate values. Other locales may use a semi-colon. Versions of the files that use the semi-colon delimiter are also provided. For this example use the listed files. Typically you use the file version with the delimiter specified in the Microsoft Windows regional or language settings for your locale.

- 6 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.

- 7 Click Close on the confirmation dialog box.

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


- 8 On the Cable & Harness Library dialog box, click Close.

Export Library Data

When exporting data from a library, you set the name and location for the output file, and then the system writes the data to the comma-separated data file as specified. The data file includes information for each wire or cable contained in the library, and respects the locale-specific list separator for the column delimiter. The configuration file specifies which properties to export.

All property values are exported using the unit preference settings. The same data file (.csv) used for export (along with the corresponding configuration file) can be used for import, which constitutes a complete round trip from and to the library.

Export wire data from the library

- 1 Ensure that the harness assembly is active and the Cable & Harness Library dialog box still displayed.
- 2 On the Cable & Harness Library dialog box toolbar, ensure that the Wire object type is selected, then click Export Library Object  .
- 3 Enter the name and location for the Data File (.csv) that is output, or select an existing file.
- 4 On the Export Library Wires dialog box, select the needed Configuration File (.cfg), *WireLibrary.cfg*.
- 5 Click OK on the Export Library Wires dialog box.
- 6 Click Close on the confirmation dialog box.
- 7 Click Close on the Cable & Harness Library dialog box.

Practice Your Skills

For additional practice, you can:

- Create additional wires
- Copy a cable definition
- Delete a cable definition
- Add properties to a cable definition

Use the skills you just learned along with the following steps to create a library cable definition.

Create a library cable definition

- 1 Select the Cable object type on the Cable & Harness Library dialog box.
- 2 Select the New tool.
- 3 Click the General tab, and then enter a unique cable Name.
- 4 Optionally, provide a Part Number and Category.
- 5 Select the needed Outer Diameter setting:
 - For a fixed diameter, clear the check box and enter a diameter value.

- For a variable diameter, select the Calculate from Wires check box.
- 6 Click the Wires tab, and then click Add to add wires to the cable definition.
- 7 On the Add Wires dialog box, select a wire from the list of library wires, and then click Add. You must add at least one wire.
- 8 Continue adding wires until all wires that make up the cable are added, and then click OK.
- 9 If appropriate, click the Custom tab and add additional properties to the library cable definition, such as vendor and jacket color.
- 10 Click OK.
- 11 Click Save and then click Close.

NOTE The wires referred to by a cable definition must exist in the library before they can be added to the cable. This requirement includes the library cable import. The import file must reference existing wire definitions.

Work with Wires and Cables

13

In this chapter, you learn how to add wires and cables to an assembly and modify them.

About Wires and Cables

Wires and cables are retrieved from the Cable and Harness Library and placed in the assembly using manual or automatic methods. To insert wires manually, you select the wire to insert, and then identify the two pins on the electrical parts or splices. To insert cables manually, you select the cable to insert, and then identify the pins for each wire in the cable. To insert multiple wires or cables automatically, you import a wire list in *.xml* or *.csv/.cfg* format. When using an *.xml* import file, you can also generate virtual parts and bring in additional data such as properties.

To check for and correct missing RefDes information and to verify that the required electrical parts are placed in the harness assembly before importing the file, you can use the Review Harness Data tool from the Cable and Harness tab.

Once the wires and cables are in the assembly you can diagnose and correct certain data errors. You can also add points to control shape and edit, delete, and move wires and cables, and replace wires. The system automatically calculates lengths based on placement in the assembly. To help complete the design, you can also add virtual parts, such as looms, to wires and cable wires if needed. You can also add labels to wire pins and cable wire pins.

Set Modeling and Curvature Behavior

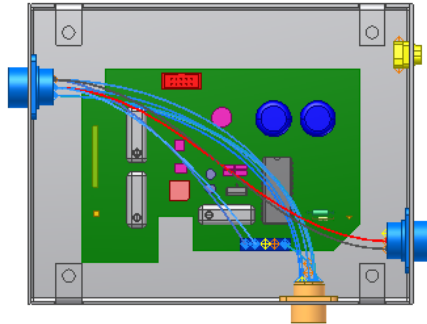
Before you begin creating harness objects such as wires and cables, you can set several default configuration options for forward creation. The defaults are set on the Harness Settings dialog box.

To view and examine the current settings

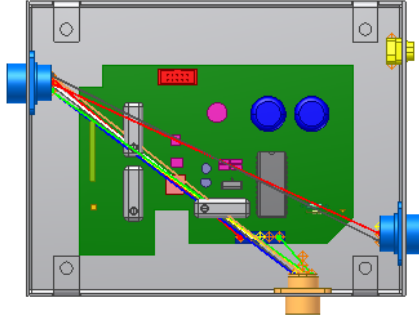
- 1 Activate Harness Assembly1, right-click, and select Harness Settings.
- 2 On the General tab examine the settings of interest, and then ensure the Use Surfaces for All Segments and Wires is selected. It is the default. Use this setting for faster processing times while creating and editing. Clear the check box to use solids.
- 3 Click the Wires/Cables tab and examine the settings of interest.



- 4 Under Natural Curvature click Without Natural Curvature .
 - With Natural Curvature creates wires and cables that are tangent to the outward direction assigned to the associated connector. Wires and cables created with tangency are displayed in a more natural shape that curves as the wires and cables approach the connector pins. Including this natural curve makes the length of the cable more realistic. It is the default.



- Without Natural Curvature creates wires and cables that are not tangent to the outward direction assigned to the associated connector. Wires and cables created without tangency are displayed as straight lines that go from point-to-point.



- 5 Click the other tabs and examine the settings of interest.
- 6 Click OK.

Insert Wires and Cables Manually

To insert individual wires or cables into an assembly, use the Create Wire or Create Cable tool.

Insert Wires Manually

To insert wires, you provide a wire ID, select the wire to connect, and then select the pins for each end of the wire.

Insert a wire manually

- 1 In the browser, double-click Harness Assembly1, if it is not already active.



- 2 On the ribbon, click Cable and Harness tab ► Create panel ► Create Wire .
- 3 On the Create Wire dialog box, accept the default Wire ID (Wire1).
The Wire ID is “Wire” plus an incremental number starting with 1.
- 4 From the list of wires, select the wire to insert. Use the arrows to select the following category and wire name from the lists.

Category: Generic

Name: 22AWG-GRN

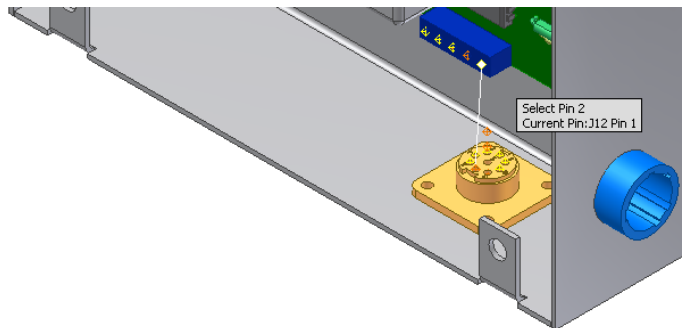
- 5 Verify that the Pin 1 button is depressed. If it is not, click the button.
- 6 In the graphics window, move the cursor over the pins in part 360124. As you select points for a wire, the application displays tooltips and dynamic prompts to instruct you on the next step. When the tooltip is displayed as U3 Pin1, click the work point representing the start pin for the wire.



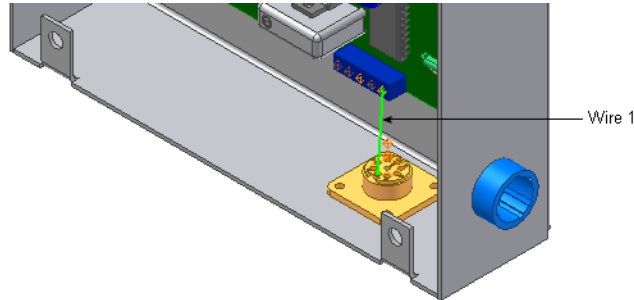
The pin changes color when selected. The Pin 1 button changes from red to white, and the Pin 2 button is activated (depressed).

NOTE To turn off dynamic prompts, click Tools tab ► Options panel ► Application Options ► General tab and clear the Show command prompting (Dynamic Prompts) check box.

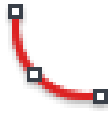
- 7 Pause the cursor over the first pin on part LTP. As you select the second pin for a wire, a preview wire is displayed along with the tooltips and dynamic prompts.



- 8 When the tooltip is displayed as J12 Pin 1, click the point to select the second pin.
- 9 To create the wire, click Apply on the dialog box.
The wire is created as centerline or rendered, depending on the display setting. Display is set to centerline for wires and cables by default.



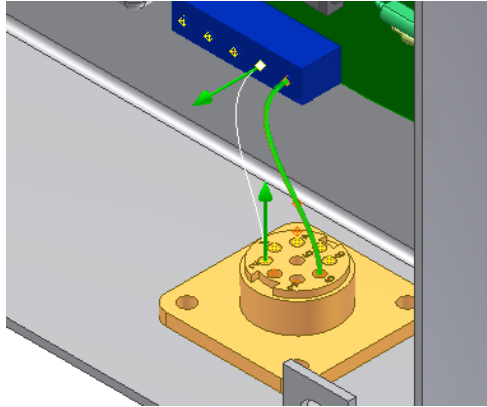
- 10 While the dialog box is still open, click the Properties button to view the properties for the library wire. Click OK.
The wire that is placed is an occurrence wire. The occurrence inherits the properties from the library wire.
- 11 On the Create Wire dialog box, click Cancel.



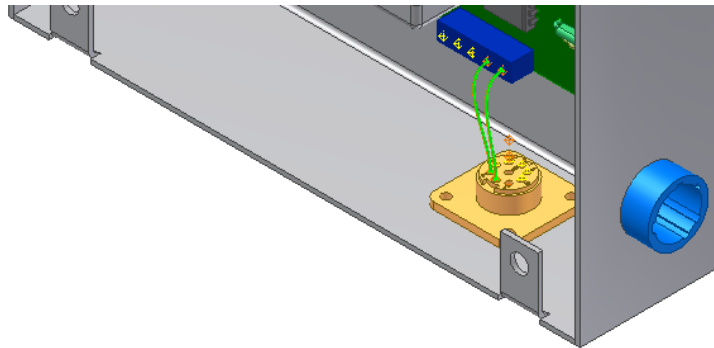
- 12 Before adding the second wire, use the skills you learned previously to change the natural curvature setting for wires and cables back to With Natural Curvature .
- 13 For the wires to display natural curvature, the associated connectors must be authored
Double-click 360124:1.
- 14 Click Manage tab ► Author panel ► Connector.
- 15 Click on the top face of 360124:1 to define the outward normal direction.
- 16 Click OK to author the connector.
- 17 Click OK to close the message box. Return to the harness assembly.
- 18 You need to author the other connectors to display natural curvature.
Follow the steps to author connectors for LTP and 360575.

Note how natural curvature is displayed once you author the associated connectors.

- 19 Add one more wire connecting U3 Pin 2 to J12 Pin 2.
As you select points for the wire, the application gives you visual feedback on the outward direction for each point you select along with the preview wire and prompt.



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



- 21 Click Cancel.
- 22 In the browser, right-click, and then select Finish Edit.
- 23 In the browser, expand the Harness Assembly1 part and the Wires folder to view the added wires in the hierarchy.
- 24 Save your files.

Insert Cables Manually

To insert cables, you provide a cable ID, select the cable to connect, and then select the pins for each wire, or conductor, in the cable. The system automatically advances through the cable wires in sequence, starting with the first cable wire. If needed, you can also manually select the order in which to connect the cable wires.

In this exercise, you add a second harness assembly so that the cable harness can be documented in a separate nailboard drawing, and then you add the cable wires.

Insert a cable manually

- 1 Ensure that nothing is selected in the browser.
- 2 On the ribbon, click Environments tab ► Begin panel ► Cable and Harness, enter the name for the harness assembly and the location as shown, and then click OK (example for Windows® XP):

Harness Assembly File Name: *Harness Assembly2.iam*

Harness Assembly File Location: Enter the path, or browse to *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness\Enclosure_Assembly\AIP\Cable & Harness*

- 3 Add the following reference designators for the connectors:
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. Use the arrows to select from the list:

Category: **Belden**

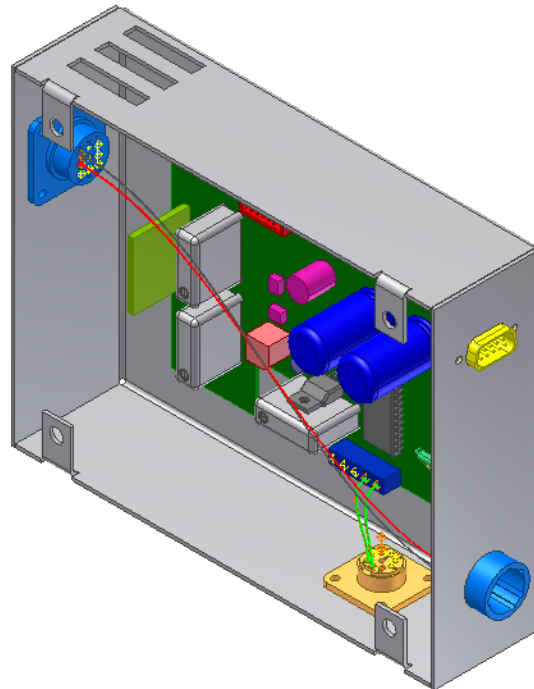
Name: **8441**

This cable contains two wires to connect in the assembly.

- 7 Verify that the Pin 1 button is depressed. If it is not, click the button.
- 8 By default, the first cable wire is automatically selected for connection. If it is not, select the first cable wire in the list.
- 9 In the graphics window, move the cursor over the pins in part 36575:2, rotating as needed to see the part. When the tooltip is displayed as U8 Pin 1, click the work point.
Cable wires are automatically assigned Wire IDs as they are connected. By default, the Wire ID is the Cable ID plus the Conductor ID.
- 10 Connect the other end of the cable wire to part 36575:1. When the tooltip is displayed as U7 Pin 1, click the work point. The status of the cable wire in the list changes to connected, and then the system automatically advances to the next cable wire.

	Conductor ID	Color Style	Gauge
Connected cable wire	1	Black	22
Spare (disconnected) cable wire	2	Red	22

- 11 Connect the second cable wire from Pin 2 on part 36575:2 (tooltip U8 Pin 2) to Pin 2 on part 36575:1 (tooltip U7 Pin 2).
- 12 If appropriate, click the Properties button to view the properties for the cable.
- 13 Click OK.



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

In the browser, cable wires are named with the Cable ID followed by the conductor ID, and the conductor ID in parentheses. Cable wire names are changed by modifying the Wire ID in the Cable Wire Properties dialog box. You cannot remove the conductor ID in parentheses.

Move Wires and Cables

From within a harness assembly, a wire or cable can have one or both of its ends moved and reconnected to different pins. Both routed and unrouted wires and cables can be changed.

A dangling wire can also be moved to reattach detached ends to existing pins. Spare cable wires, cable wires that have no connections, can also be attached to pins, including splice pins.

Move Wires

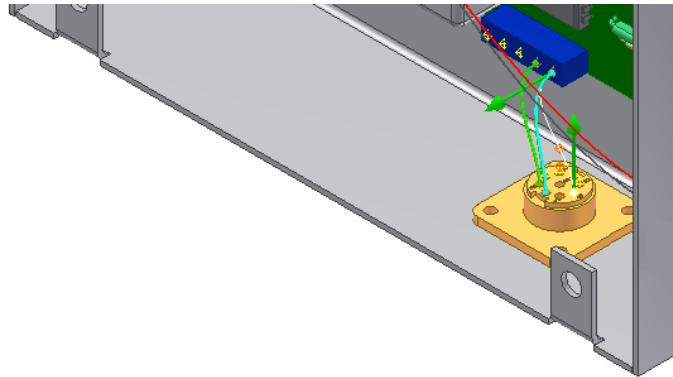
To move a wire, choose the wire to move, select Edit Wire from the context menu, click the selection button representing the pin connection to change, and then select the new pin connection. In this exercise, you move Pin 2 of the wire you added previously to another pin on the same connector.

Move a wire

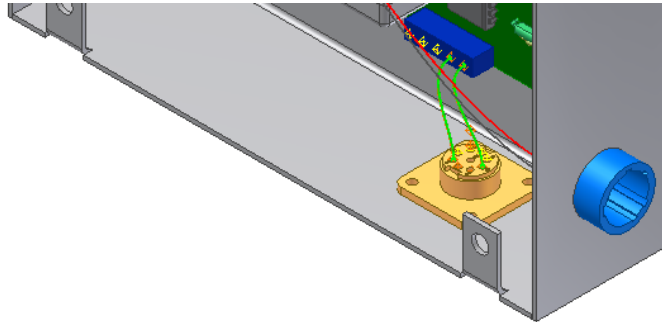
- 1 Activate Harness Assembly1.
- 2 In the browser or graphics window, right-click Wire1, and then select Edit Wire from the context menu.

NOTE To select the pin in the graphics window, you may need to set the Select tool to Select sketch features.

- 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.
The preview wire is a straight white line drawn between the two pins.
- 5 When the tooltip is displayed as U3 Pin 6, click the pin as shown in the following illustration.



- 6 To create the wire in the new location, right-click, and then select Apply from the context menu.
The wire is redrawn using the new pin, and the wire length is updated as needed.



- 7 Click Cancel.

Move Cables

Moving cables and wires is similar. To move a cable, however, you move the wires it contains.

Workflow: Moving a cable

- 1 Select the cable containing the cable wires to move.
- 2 Select Edit Cable from the context menu.
- 3 On the Edit Cable dialog box, click the wire to move.
- 4 Click the selection button representing the pin connection to change.
- 5 Select the new pin connection.
- 6 Continue selecting cable wires and changing pin connections as needed until the desired cable wires are moved.
- 7 Click Cancel.

Delete Wires and Cables

Wires and cables you no longer need in your design can be deleted at any time. Any virtual parts attached to the wire or cable are also deleted.

Delete Wires

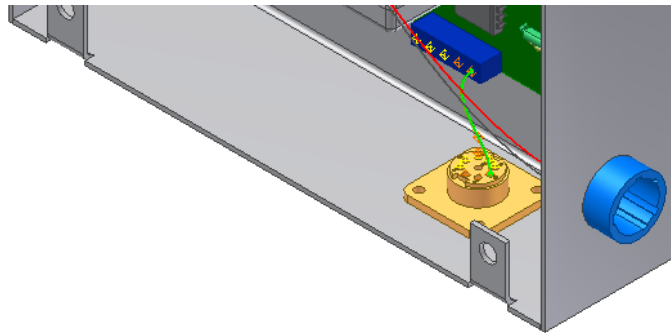
To remove a wire, select the wire to remove, and then delete it using the context menu.

Delete a wire

- 1 Activate Harness Assembly1, if not already active.
- 2 In the browser or graphics window, select Wire2.

NOTE To select the wire in the graphics window, you may need to set the Select tool to Select Sketch Features.

- 3 Right-click and select Delete from the context menu.



The wire is deleted.

Remove Cables and Cable Wires

There are several ways you can remove cables from your design. You can:

- Delete an entire cable.
- Disconnect a selected cable wire from its current connection.
- Delete a selected cable wire from the cable definition.

Like wires, you can delete cables that are no longer used at any time. To delete a cable, use the same workflow used for deleting wires.

To remove a cable wire from a cable, edit the cable definition in the Cable and Harness Library and then delete the cable wire. Removing a cable wire

from a cable definition does not remove the cable from the library. It does remove the wire from any occurrences of that cable in the harness.

Wires that make up a cable can also be disconnected. You can disconnect a wire to reterminate it to different pins, or to leave it as a spare. Disconnected wires are also removed from the graphics window. Each time a cable wire is disconnected, all properties on the cable wire are reset to the library-level property values.

Disconnect a cable wire

- 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 a cable wire to disconnect.
- 5** Click Disconnect, and then click Apply.
Notice how 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.

Replace Wires

To change or replace a wire definition, you select the wire to change, and then select the new library wire definition. The wire is redrawn using the new definition. The wire occurrence property values, including the Wire ID, remain the same and the library-level wire properties are added to the occurrence wire immediately.

Wire color changes take effect immediately. To see the effect of changes to outer diameter values, click the Update tool.

Replace a wire

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

NOTE To select the wire in the graphics window, you may need to set the Select tool to Select sketch elements.

- 3 On the Edit Wire dialog box select:
Category: Belden
Name: 9916-V10
- 4 If necessary, click Properties to view the properties for the library wire.
- 5 To redraw the wire using the new definition, click Apply, and then Cancel. The wire in the graphics window is redrawn in violet.
- 6 Return the wire to the original, green wire definition. On the Edit Wire dialog box, select:
Category: Generic
Name: 22AWG-GRN
- 7 Click OK.

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

Assign Virtual Parts

You can assign virtual parts to various objects in a harness assembly using the Assign Virtual Parts tool on the Cable and Harness tab. You can also assign virtual parts to individual objects through the Property dialog box of an object. You can use one of the virtual parts in the Cable and Harness Library, or you can create and use your own. Refer to the Help for information on types of virtual parts that may be assigned to various harness object types.

For this exercise, you assign the Wire ID label you created previously to a wire pin.

Assign a label to a wire pin

- 1 Activate Harness Assembly1, expand the harness part in the browser to see the Wires folder.
- 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.

Import Harness Data

Use Import Harness Data on the Cable and Harness tab to add (connect) multiple wires or cables in the harness assembly automatically. You can create these files yourself using a text editor or spreadsheet, or you can import this data from other applications, such as AutoCAD® Electrical. You can import The import many times with the same or different import files.

NOTE When importing from AutoCAD Electrical, refer to the Help for tips on preparing your data for use in Cable and Harness.



Import Harness Data command

To import the wires and cables, select the import files to use. The import files include a configuration (*.cfg*) file and a comma separated (*.csv*) data file or an *.xml* file. The data in these files specifies the point-to-point connectivity of each wire and cable wire. Ribbon cables cannot be included in import files.

The configuration file, which respects the locale-specific list separator, describes the format of the input *.csv* data file. An *.xml* data file combines the information provided in both the *.cfg* and *.csv* file. It can also include information on properties and virtual parts that the *.csv* file cannot. Neither format supports the import of a connector, splice, or pin.

Once the wires and cables are imported, the system indicates the status of the import, displays the Imported Harness Data dialog box, and writes a log file containing the details of the import.

The import checks for duplicates, missing, and changed wires or cable wires. If a new wire or cable has the same ID as an existing one, it is not added. A

warning message is displayed if the duplicate entry has different data or connectivity.

To ensure that the import is not interrupted due to missing parts, use Review Harness Data on the Cable and Harness tab to parse the harness data in a harness data import file and generate a list of all the electrical parts defined in that file. The list includes all connectors and splices and their defined properties. An error condition icon is displayed for any electrical part that does not have a RefDes in the assembly.

Use this list to check for and correct missing RefDes information and to verify that the required electrical parts are placed in the harness assembly before importing the file. Refer to the online Help for more details.

Once the harness data is imported, you can use the Imported Harness Data dialog box to:

- View the imported data.
- Identify and correct any issues with the data.
- Add additional information to provide a complete definition.

Configuration File Formats

The configuration file must contain the following information to accommodate both wires and cables. Previous file formats that contain only wires still work for wire import. Cable occurrences must use the format listed:

OBJECT	Type of object contained in the input file. For cable and wire occurrences, the object is cablesandwires. It is not case sensitive.
LOGFILE	The output log file path and name. For example, <i>C:\WireListImport\wirelist.log</i> . If no path is listed, the log file is created in the same directory as the .csv file.
HEADER1 (Property Name)	Specifies the order, data type, and units of the input parameters. There must be a Wire ID, Cable ID, Library Name, Conductor ID, REFDES1, Pin1, REFDES2, and Pin2. The REFDES can be from splices or connectors. The Cable and Conductor IDs are required only when importing cable wires. The custom properties and required properties can be in any order, as long as the minimum required subset is present.

HEADER2 (Data Type)	Specifies the data type associated with the property name in the same column.
HEADER3 (Unit)	Specifies the unit of measure associated with the data type in the same column.

The wire connectivity parameters in the comma-separated data file must have the exact data types and units specified in the configuration file. Units must be appropriate for the data type specified for the given property name.

See the Help for more details on data types and unit handling.

NOTE Configuration files are used for importing wire connectivity data and cable and harness library data, and for producing reports. Each configuration file performs a different task, and requires information specific to that task.

Comma Separated Value Data File Formats

Any number of wires or cables can be included in the comma-separated data (.csv) file. The separator recognized in a .csv file is not always a comma. It is determined by regional settings in Microsoft® Windows®. For example, in another country the separator could be a semicolon.

For each cable and wire included in the file, the following information is required:

Wire ID	A unique identifier for each wire.
Cable ID	(Cables only.) A unique identifier for each cable. Leave this column blank for wires.
Library Name	A wire or cable name in the library.
Conductor ID	(Cables only.) A unique identifier for each cable wire. Leave this column blank for wires.
REFDES1	A unique identifier for the first connector or splice occurrence.
PIN1	A unique pin name on the first pin.
REFDES2	A unique identifier on the second connector or splice occurrence.
PIN2	A unique pin name on the second pin.

Additional wire or cable occurrence properties can be imported on an object-by-object basis after the required input parameters. The corresponding data type and unit for each custom property added must be listed in the associated configuration file.

NOTE The electrical part RefDes value is only recognized by the Import Harness Data function when it is set at the occurrence level. Any RefDes values set in the part file are not used.

XML File Formats

An XML schema is used to describe the format of *.xml* files used to import and export data for a harness. The schema describes most of the data model of the Cable & Harness application for a single harness. Ribbon cables are not included.

See the Help for more details on the *.xml* schema. In the Index, search for and select XML files, and then select “Harness data XML file schema reference.”

Review Harness Data

Electrical parts and pins and their RefDes must exist in the assembly before importing harness data. To ensure that the import is not interrupted due to missing parts, use Review Electrical Parts from the Cable and Harness tab to generate a list of all the electrical parts that are defined in the import file, along with any missing RefDes information. An error condition icon is displayed for any electrical part that does not have a RefDes in the assembly.

NOTE If file or format errors exist that prevent the file from importing, the Import Harness Data message dialog box is displayed with a description of the issue. If errors exist, but the file still imports, click View Log for details.

Ensure that electrical data is complete before import

- 1 In the browser, double-click Harness Assembly1 as the assembly to receive the imported wires.



- 2 With Harness Assembly1 still active, click the arrow beside Import Harness Data and select Review Harness Data .
- 3 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 where you installed Autodesk Inventor.

Data File (.csv): *WireListImport.csv*

Configuration File (.cfg): *WireListConfiguration.cfg*

- 4 Click OK to review the data.
- 5 On the Review Harness data dialog box, you see error-free information on the three connectors.

NOTE If errors exist, use this list to correct the RefDes or other errors.

Import Harness Data

In this exercise, you import seven discrete wires into the harness assembly using import files that were already created for you. The cable columns, cable ID, and Conductor ID, are left blank. To view the file contents, open the file in a text editor.

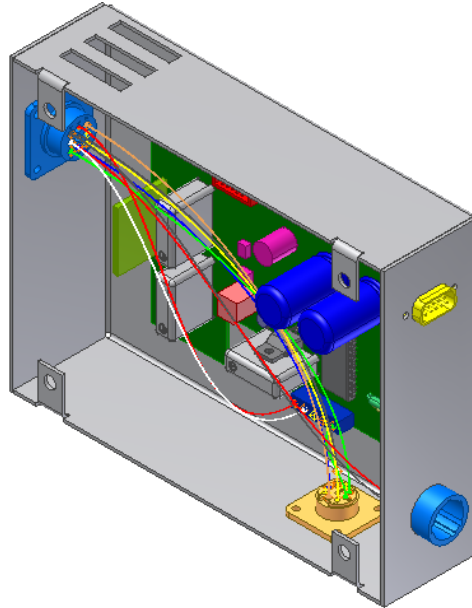
Import a wire list to a harness assembly



- 1 With Harness Assembly1 still active, click the arrow beside Review Harness Data and select 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 where you installed Autodesk Inventor.
Data File (.csv): *WireListImport.csv*
Configuration File (.cfg): *WireListConfiguration.cfg*
- 3 To add the wires to the active harness assembly, click OK.
The Imported Harness Data dialog box is displayed and lists the items that were imported. For this example, there should be no errors. If errors exist, you can right click the items in question for more information on the issue and how to fix it.
- 4 Optionally, expand the folders on the Import Harness Data dialog box to view details on the import.
- 5 Click OK.

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

- 6 Click Close on the message dialog box. The wires are added to the harness assembly.



- 7 Exit Harness Assembly1 and save the files.

The following is an example of a wire list in the .csv format that contains both wires and cables. There are three discrete wires and two cables:

```
//Wire ID,Cable ID,Library Name,Conductor ID,REFDES 1,Pin 1,REFDES 2,Pin 2  
1001,CA1,Belden 4C 22AWG,1,J1,2,J5,6,  
1002,CA1,Belden 4C 22AWG,2,J1,3,J5,7,  
1003,CA1,Belden 4C 22AWG,3,J1,11,J5,8,  
1004,CA1,Belden 4C 22AWG,4,J1,12,J5,9,  
1005,,18 AWG BLU,J3,2,J6,1,,  
1006,,18 AWG BLK,J3,3,J6,3,,
```

1007,,18 AWG RED,J3,4,J6,2,,
1008,CA2,Belden 4C 22AWG1,J2,1,J8,3,,
1009,CA2,Belden 4C 22AWG2,J2,2,J8,4,,
1010,CA2,Belden 4C 22AWG3,J2,3,J8,5,,
1011,CA2,Belden 4C 22AWG4,J2,4,J8,6,,

Add Shape to Wires and Cable Wires

By default, a wire or cable wire is initially created with natural curvature between the pins to which it is attached. If this curvature is not appropriate or if the wires or cables wires were created without natural curvature, you can control the shape by adding and adjusting wire work points on:

- Unrouted wires and cable wires
- Wire stubs and cable wire stubs

You cannot add work points to cables or spare cable wires.

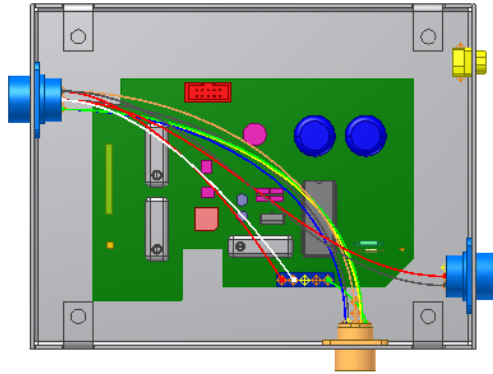
NOTE Because all wire and cable wire work points are deleted when routed, we recommend that you add work points only after routing the wire or cable wire for the first time.

Once work points are added, you can adjust their location and offset value to achieve the shape you need.

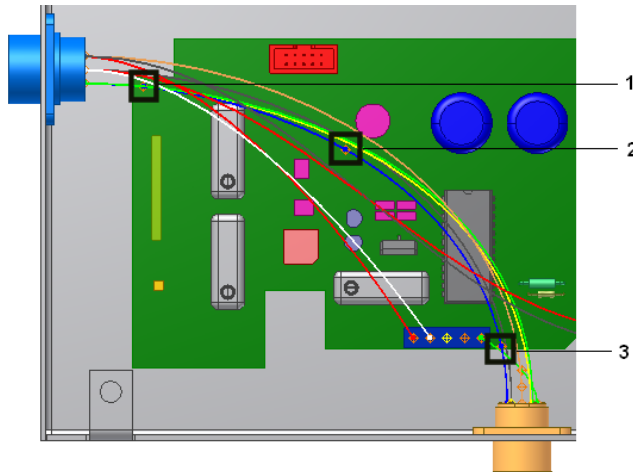
In the following exercises, you add points on a wire, adjust the points to change the shape of the wire, and then delete them.

Add Wire Points

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



- 2 Double-click Harness Assembly1.
- 3 In the browser, right-click wire 2207, and then select Add Points on the context menu.
- 4 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.

- 5 Right-click and select Finish.

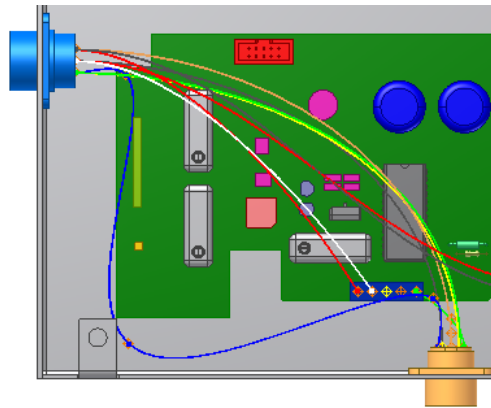
Redefine and Move Wire Points

Once the points are added you can redefine the points to an arbitrary location or feature to achieve the shape you need. Wire points based on an arbitrary point on a face are offset a specified distance and does not update to changes in the model, including changes from positional representations. Like the points used to define pins on parts, points based on existing geometry are not offset and update to changes in the model.

You can also reposition wire work points dynamically or by precise coordinates with the 3D Move/Rotate tool.

Manipulate wire points

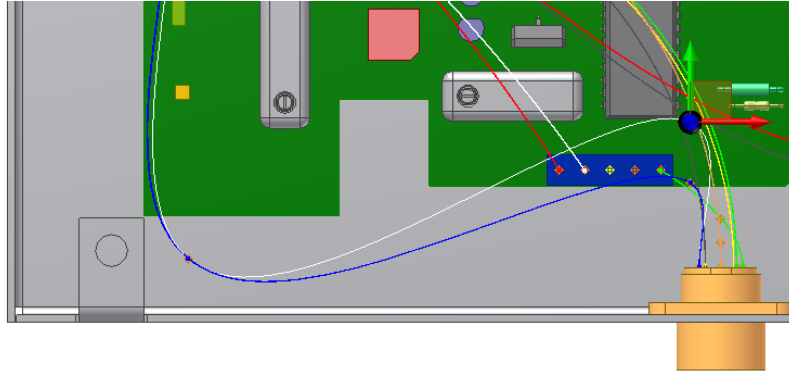
- 1 In the browser, expand wire 2207, right-click the second work point you created and then select Redefine Point from the context menu.
- 2 In the graphics window, pause the cursor near the tab on the enclosure as shown. Notice the tooltip that shows the offset value for the point.
- 3 In the graphics window, right-click (not on the point), select Edit Offset from the context menu, and then enter `.200` 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.



NOTE To create a point that would update to changes in the model, click the circular edge of the tab.

- 5 If appropriate, rotate the view to see the increased offset.

- 6 In the browser or graphics window, right-click the third work point created, and then select 3D Move/Rotate.
- 7 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.



- 8 Click OK and the wire recomputes to the new location.

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.

Set Occurrence Properties

Wire and cable occurrences inherit the general properties that were created at the library level. Occurrence properties can be stored within the assembly in which the wires and cables are placed. You can override certain library-level property values with an occurrence property value.

Wire Occurrence Properties

To access the Wire Properties dialog box, right-click on a wire and select Harness Properties. The Wire Properties dialog box for wire occurrences has several tabs.

Occurrence	Displays the Wire ID property that can be edited on the occurrence. It also includes length information, bend radius, and display options.
	<hr/> NOTE We recommend that you define the bend radius at the library level so that it appears on all occurrences. <hr/>
Display	Lists properties related to the display of wires and any attached looms. It also enables you to view and modify the order of multiple, attached looms.
General	Displays the library-level properties for the selected wire occurrence. These properties cannot be edited at the occurrence level.
From/To	Lists the RefDes and pin name for each end of the wire and provides links to the properties for those pins and connectors.
Virtual Parts	Enables you to add and remove looms and custom virtual parts for the selected wire. It also lists any virtual parts attached to the selected wire.
Custom	Lists and sets the custom properties for the selected wire.

The Custom tab of the properties dialog box for wire occurrences contains several predefined, or recommended, properties. These properties are available in the property Name list and have a preset name and type. You can add them to the occurrences individually and set the value. The recommended properties for wire occurrences are:

Separation Code	Categorizes wires into types or categories for comparison.
Class	Categorizes wires and segments into class groupings for comparison.
Max Length	The maximum allowable length for wire.

Cable Occurrence Properties

To access the Cable Properties dialog box, right-click on a cable and select Harness Properties. The Cable Properties dialog box for cable occurrences also has four tabs.

Occurrence	Displays the Cable ID property that can be edited on the occurrence. It also includes length, bend radius information, and display options.
General	Displays the library-level properties for the selected cable occurrence. These properties cannot be edited at the occurrence level.
Wires	Lists the wires in the cable and enables you to connect and disconnect cable wires from the cable.
Custom	Lists and sets the custom properties for the selected cable.

The Custom tab of the properties dialog box for wire occurrences contains several predefined, or recommended, properties. These properties are available in the property Name list and have a preset name and type. You can add them to the occurrences individually and set the value. The recommended properties for cable occurrences are the same properties listed for wires.

Override Library-level Properties

In this exercise, you change the value of the custom Volt property for a wire.

Override a library-level property

- 1 Double-click Harness Assembly1 if it is not already active.
- 2 In the browser or graphics window, expand the Harness Assembly1 part and the Wires folder.
- 3 Right-click Wire1, and select Harness Properties on the context menu.
- 4 Click the Custom tab. Notice that the custom properties added at the library level are displayed with a dark background.
- 5 Click the Volt property in the table.
- 6 Select the property Value of 300 volts and change it to 200 volts.
- 7 Click Modify to override the property in the table, and then click Apply.

Restore Library-Level Properties

To restore a property value to its original library-level value, position the cursor anywhere in the selected table row, right-click, and then select Restore.

In this exercise, you restore the library-level property value for the custom Volt property with the value override.

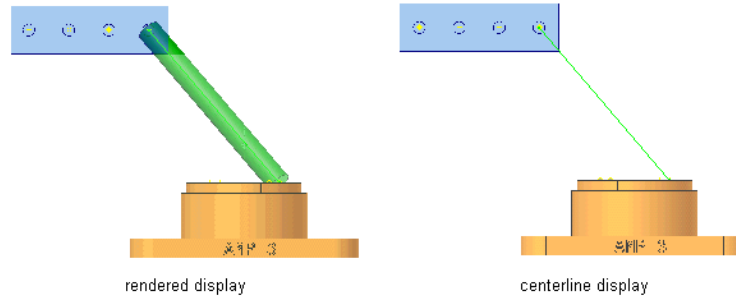
Restore a library-level property value

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

Change Wire and Cable Displays

Wires and cables can be displayed as either rendered or centerline. Centerline display is the default and should be used for optimal performance while creating and routing wires and cables.

If Rendered Display is selected, the harness object is drawn as a three-dimensional shaded solid with the diameter as set in the library definition, which is similar to the physical object appears. With centerline display the objects are drawn as lines, making it easier to see and work on existing model geometry.



The rendered and centerline display options can be set for individual wires or cables or for all wires or cables in a selected harness assembly as shown in the exercise that follows.

Change the display for wires

- 1 Double-click Harness Assembly1 if it is not already active.
- 2 In the browser or graphics window, select any wire from the list of wires.

NOTE To select the wire in the graphics window, you may need to set the Select tool to Select Sketch Features.

- 3 Right-click and select Display as Rendered on the context menu.
- 4 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.
- 5 On the Cable and Harness tab, Visibility panel, select Centerline Display to return the wires to the default centerline display mode.

Work with Segments

14

Segments define virtual paths used for routing wires and cables through assemblies.

In this chapter, you learn how to define and manipulate a segment, how to branch a segment, and how to change the segment display characteristics.

About Segments

Segments define the possible paths a wire or cable can take through an assembly. Each click in the graphic window creates a work point on the segment. Work points are used to manipulate the segment into its desired position and shape.

The segment is created with a default diameter, and is offset a specified distance from selected geometry.

Once a path is defined, it can be modified in several ways, including inserting splices or creating segment branches.

Define Segments

Use the Create Segment tool on the Cable and Harness tab to create a path for routing wires and cables.



Create Segment command

To create a segment you define at least two points, a start point and an endpoint. As you design, you can add additional segment work points where it is critical for the segment to adapt to changes in the assembly, or to control the direction

of the segment around or through existing assembly geometry. Use as few work points as possible, since additional work points slow performance and make relocating work points more difficult.

NOTE You can also use segment points to assign labels and custom virtual parts.

For more precise placement of segment work points, define work features at critical locations before inserting the segment. Work features, such as work points and work axes, are especially useful for directing segments through tie-downs, clips, or other geometry. Segment work points associated to work features update when the model changes. They also react to positional representations. Work features can also help create a more accurate approach to the connectors, or direct the segment around geometry to avoid.

Feedback is provided throughout the creation process to highlight valid points and show the offset for your selections.

Select Work Points for Segments

Like the points used to define pins and wire points, the points you select for segments determine whether certain work points are updated when changes are made to the associated geometry. Segment work points created by selecting arbitrary points on a face are not updated to changes in the model geometry. Segment work points based on the following geometry, do update when changes are made:

- Existing work points
- Center points on any circular component such as a face, a hole, and cylindrical cuts or arc edges
- Existing sketch points
- Model vertices

Select Undo or press ESC while creating a segment to remove the last segment point created and end the create segment operation.

Plan Segment Start Points and Endpoints

When multiple segments exist in an assembly, the placement of the start and endpoints of the segment in relation to the wire or cable connection points is an important consideration. When automatically routing wires and cables,

the system checks for the harness segment ends closest to each end of the wire or cable, and then searches for the shortest path through the identified contiguous segments. Whenever possible, place the harness segments so there is no confusion as to which segment is closest to the connection points.

See [Define Automatic Routes](#) on page 299 for details about how the system automatically routes wires and cables.

Apply Offset Distances

Arbitrary points selected on a face are offset a set distance from the selected geometry. Segment points based on existing geometry are not offset. A segment can require several different offset distances throughout its length. To adjust the segment offset as you move through the assembly, right-click in the graphics window, and select Edit Offset.

NOTE The offset distance is applied only while you create segments. It is not applied when wires or cables are routed through the segment and cause the update of the diameter.

Create Segments

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.

In the following exercises, you define a harness segment between two circular connectors and around the PCB board in the enclosure. You adjust the offset while you work. You define a second harness segment between two circular connectors and around the PCB board on the opposite side of the first segment.

A work point is defined to control the approach to the first connector.

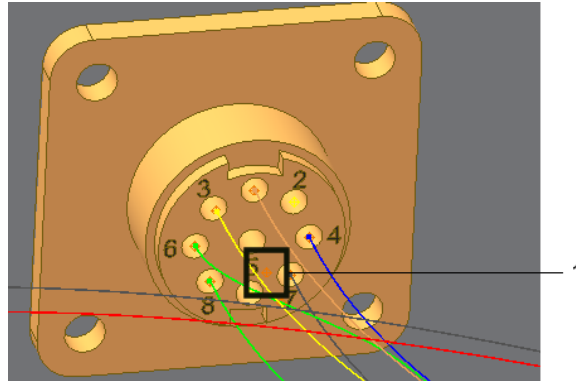
Start a segment

- 1 In the browser, double-click Harness Assembly1 if it is not already selected.



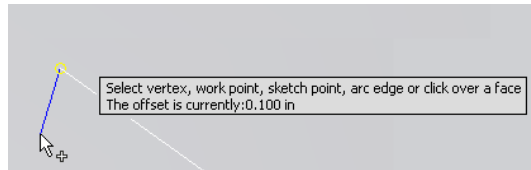
- 2 On the ribbon, click Cable and Harness tab ► Create panel ► Create Segment.

- 3 In the graphics window, orient and zoom the view as shown in the following illustration, and then select point 1.



A segment work point is created at the selected point.

- 4 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.

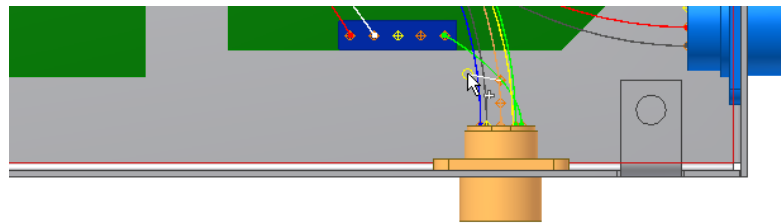


Close-up of offset symbol

- 5 Before you select the second point, increase the offset to prepare for a branch that is created in a later exercise. To increase the offset, right-click, select Edit Offset from the context menu, and then enter 0.270.

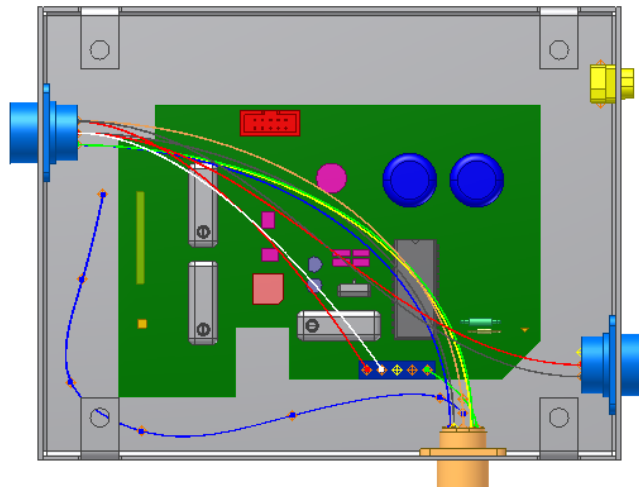
NOTE Units are not required when entering offset values. If no unit is specified the default unit setting is used.

- 6 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, a straight line appears from the selected point to the current cursor position. The system draws the line between the points as selections are made.

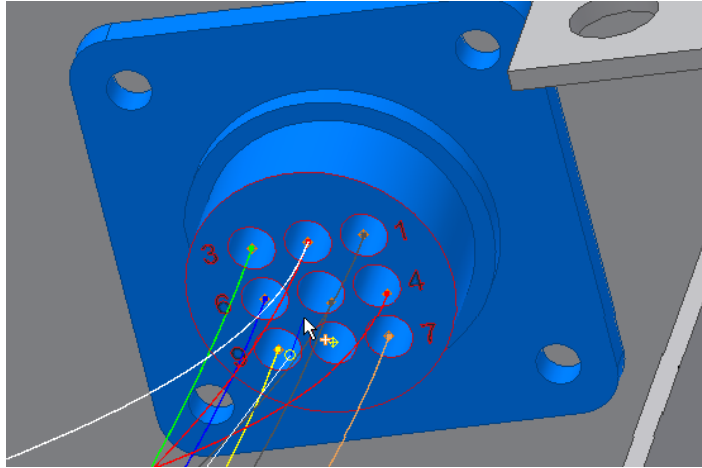


- 7 Continue selecting points around the printed circuit board (PCB) and toward the other connector to create the segment shape like the following image.

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



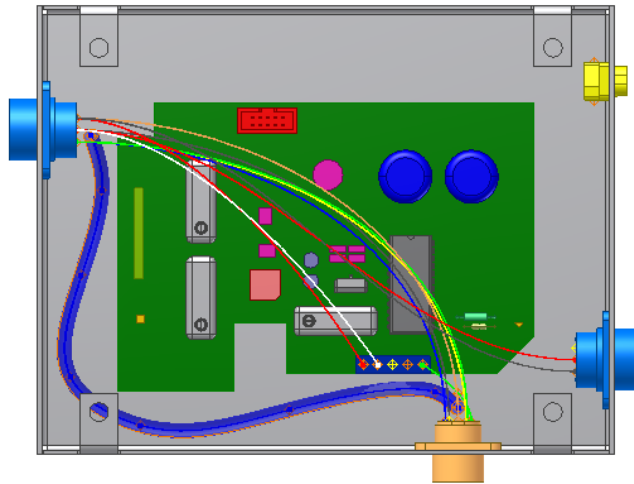
- 8 For the last point, pause the cursor over the connector face near the center hole.
- 9 Right-click, select Edit Offset, and then enter `.200`.
- 10 Click to select the point as shown in the illustration.



- 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. See [Redefine or Move Segment Work Points](#) on page 288 for details about adjusting segment work points.

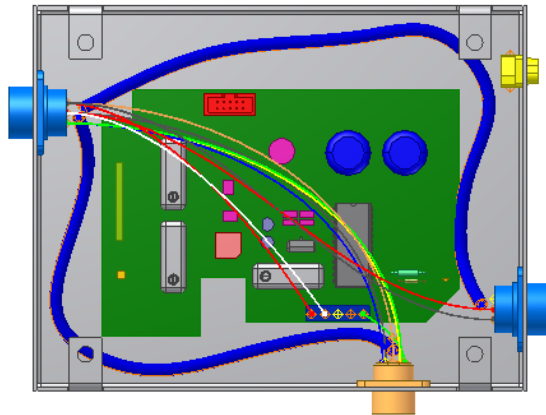
The system generates the segment, as shown in the following illustration.



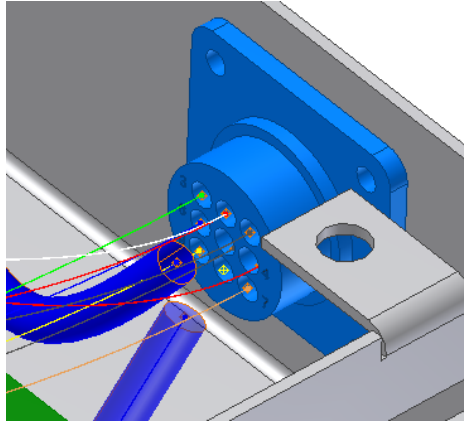
- 12 Save the harness assembly.

Start another segment

- 1 Activate Harness Assembly2.
- 2 Create a segment from connector 360575:2 to connector 360575:1
Click Cable and Harness tab ► Create panel ► Create Segment.
- 3 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 *.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.

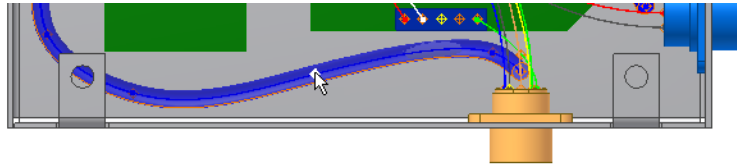
Add Segment Branches

Segment branches are made by starting or ending a new segment on an existing segment. Once completed, there are three segments sharing a common endpoint. The original segment is broken into two separate segments. No tangent constraint is formed between the branch and the two original segments.

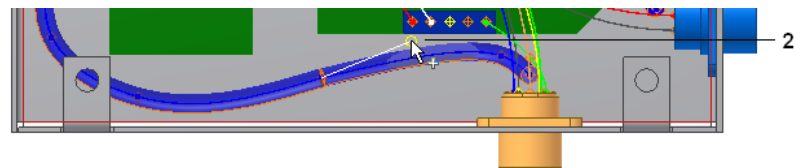
As wires and cables are routed, each of the three segments behave as separate entities and can assume different diameters.

Add a segment branch

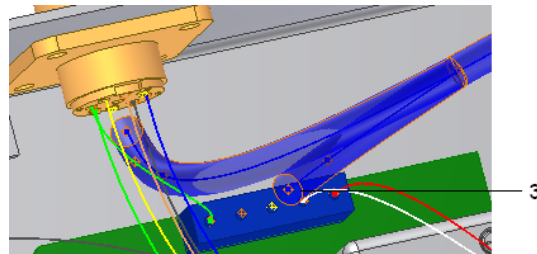
- 1 In the browser, double-click Harness Assembly1.
- 2 On the ribbon, click Cable and Harness tab ► Create panel ► Create Segment.
- 3 Pause the cursor over the harness segment to branch, and then click the segment work point shown as the branch start point.



- 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 $.270$, 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.



Manipulate Segments

Once a segment path is defined, it can be modified in several ways. You can:

- Add additional work points.
- Delete work points.
- Move and redefine work points.

Redefine or Move Segment Work Points

You move and redefine segment work points just as you would move and redefine other work points. To modify grounded and ungrounded work points, right-click the work point in the browser or graphics window, select the Redefine Point option, and then select the new location. The work point becomes associative to new geometry that you select. Changes to the geometry reposition the associated work point.

Grounded work points can also be modified using the 3D Move/Rotate option. To use 3D Move/Rotate, select the 3D Move/Rotate option, and then drag the work point to the desired location. The work point remains grounded.

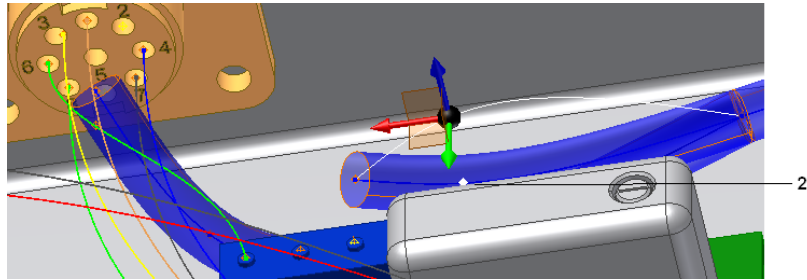
For more information about working with work points in Autodesk Inventor®, search for work points in the Help index.

NOTE When the location of a segment changes dramatically, delete the existing segment and create a new one, rather than try to adjust existing segment work points to accommodate the changes.

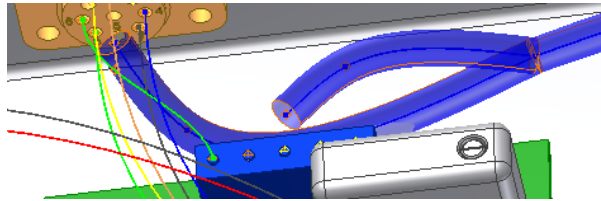
In this exercise, you use 3D Move/Rotate to adjust a work point for the segment branch.

Move the work point

- 1 To get a better view of your data, turn off the visibility for any geometry obstructing your view.
You can create design views that include only the components necessary to wire the assembly. Creating a view that shows only the harnesses or a specific harness assembly can also be useful.
- 2 Double-click Harness Assembly1 if it is not already active.
- 3 In the graphics window, right-click the second branch work point (2) as shown in the following illustration.
- 4 Right-click and select 3D/Move Rotate from the context menu.
- 5 Drag the point vertically until the preview looks similar to the following shape, right-click, and select Apply.



- 6 Continue adjusting until you are satisfied with the shape, right-click, and select OK.



NOTE Once a segment is created, you can select it in the browser, and then select Bend Radius ► Check from the context menu to verify whether any points along the segment violate the bend radius value. Make sure 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.

- 7 Save the harness assembly.

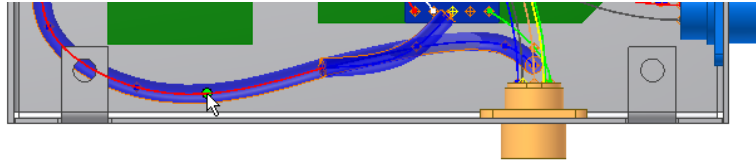
Insert Segment Work Points

If the design changes so that you need more control over the segment shape in certain areas, you can insert additional points.

Add a segment work point

- 1 Double-click Harness Assembly1 to activate it in place, if it is not already active.
- 2 In the browser or graphics window, right-click Segment2, and then select Add Points from the context menu.
- 3 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.



- 4 Click the path to add the point.
The point is added and the segment recomputes.
- 5 Right-click and select Finish.

Delete Harness Segment Work Points

To delete a work point from a harness segment, activate the harness assembly, right-click the point to remove, and then select Delete Point from the context menu. The point is removed and the segment recomputes. Any splice or virtual part attached to a deleted segment point is also deleted.

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, and can be manipulated independently.

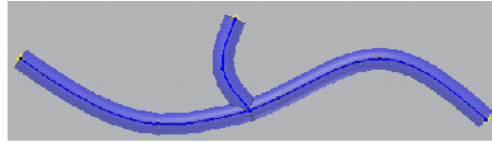
Delete a work point from a single segment

- 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.

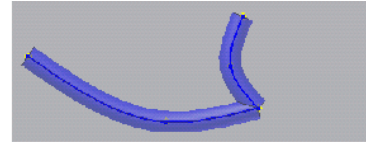
Delete Harness Segments

Before you delete an entire segment, you can choose whether you want the points from the deleted segment to remain on the wires that were routed through it, thus keeping the shape of the segment after the segment is gone. To keep the points, click the Wires/Cables tab on the Harness Settings dialog box and click Inherit Segment Points on Segment Delete. Any virtual parts attached to the segment are also deleted.

To delete a segment, select the segment from the browser or graphics window, right-click and select Delete from the context menu. If the segment deleted is a branch, the two remaining segments are not merged.



segment with branch



segment with deleted branch

When a segment is deleted, any wires or cables routed through that segment unrout from that segment only. If the wire or cable was routed through the deleted segment only, the object returns to an unrouted state. If the wire or cable was routed through more than one segment, then it remains routed through the other segments.

Set Segment Properties

Like wires and cables, segments contain a set of properties that provide a complete definition of the segment in the assembly. Unlike wires and cables, segments do not exist in the context of a library and do not contain library-level properties. They contain only occurrence-level properties.

The Segment Properties dialog box has five tabs.

Occurrence	Lists properties related to the physical aspects of the segment, including segment display and bend radius.
Display	Lists properties related to the display of segments and any attached looms. It also enables you to view and modify the order of multiple, attached looms.
Wires/Cables	Lists properties for wires and cables routed through the segment. Double-click any row to display the properties dialog box for that wire or cable wire occurrence.
<hr/>	
Virtual Parts	Use to add and remove looms and custom virtual parts for the selected segment. It also lists any virtual parts attached to the selected segment.
Custom	Lists custom properties added to the segment.

NOTE You can also attach virtual parts using the Assign Virtual Parts tool on the Cable and Harness tab.

On the custom tab there are several preset, or reserved, properties available from the Name list. See the Help for details about these properties.

Access segment properties

- 1 Double-click the harness assembly.
- 2 In the browser or graphics window, right-click a segment, and then select Harness Properties from the context menu.
- 3 Click the various tabs to view and set the desired properties.
- 4 Click the Virtual Parts tab to add and remove looms and custom virtual parts.
- 5 Click the Display tab to change order and color of multiple attached looms.

For more details on virtual parts search for “virtual parts” in the Help index.

Set Diameter Behavior in Segments

Segments can be configured as fixed diameter segments or as variable diameter segments. Fixed diameter segments, such as semi-rigid tubing, do not change as wires and cables are routed, unrouted, or deleted. The diameter of variable segments is automatically updated as wires and cables are added or removed.

By default, segment diameters are updated automatically as wires and cables are routed, unrouted, or deleted from the segment.

Change the segment diameter behavior workflow

- 1 Double-click the harness assembly containing the segments to change.
- 2 In the browser or graphics window, right-click the segment to change, and then select Harness Properties from the context menu.
- 3 On the Occurrence tab of the Segment Properties dialog box, select the appropriate diameter setting.
 - For a variable diameter, select the Calculate Size from Wires check box.
 - For a fixed diameter, clear the check box and enter a diameter value.
- 4 Click OK.


The segments are updated accordingly.

Change Displays of Segments

Segments can be displayed as rendered or centerlines. Rendered display provides a three-dimensional appearance, while centerline display provides an unobstructed view of the work points and path.

Like wires, the display options can be set for individual segments or for all segments in a selected harness assembly. The display preference can be set in different ways, either with the context menu for the segment occurrence or the Segments folder, the Segment Properties dialog box, or the Display Settings tool on the Cable and Harness tab. The occurrence level display settings always override the current display state.

Change the display for a selected segment

- 1 Double-click Harness Assembly1 if it is not already active.
- 2 In the browser or graphics window, right-click a segment and select Harness Properties.
- 3  On the Segment Properties dialog box, Display tab, click the tool for the Centerline display option.
- 4 To change the display setting for all segments, select the Segments folder in the browser, right-click, and select Display All As Rendered from the context menu.

You can also change the display of a single segment by selecting the Display As Rendered option from the Display Settings tool on the Visibility panel of the Cable and Harness tab or by right-clicking a segment and selecting Display As Rendered from the context menu.

Set Segment Defaults

Several default settings are available for harness segments. With Harness Assembly1 still active, do the following:

Set the segment defaults

- 1 In the browser, right-click the harness assembly, and then select Harness Settings from the context menu.
- 2 On the Occurrence tab of the Segment Properties dialog box, set the diameter behavior.
- 3 If appropriate, set a bend radius value.

NOTE The bend radius multiplier must be set for the bend radius check to operate.

- 4 Set the segment color.
- 5 Select the display setting, including the color style for any assigned looms. These settings are used when creating segments.
- 6 Set the distance to offset segment work points from the selected face during selection.
- 7 Click OK.

Route Wires and Cables

15

When wires and cables are routed, they are inserted into selected segments using manual, automatic, or semi-automatic methods. Unrouting wires and cables removes them from the selected segments.

In this chapter, you learn how to use the different routing and unrouting methods.

About Routing and Unrouting

By default, when wires and cables are routed, or placed into segments, the wire and cable lengths and segment diameters are automatically calculated. All work points on a wire or cable wire are deleted when routed from a point-to-point state. Wire and cable lengths and segment diameters are also calculated automatically when unrouting or removing wires and cables from segments.

When routing and unrouting individual wires, each wire is independent and can take a different path through the assembly. When routing and unrouting cables all cable wires that make up the cable must follow the same path. If a change is made to the routing or unrouting of one cable wire, all associated cable wires also change.

Wires and cables can be routed only into segments that are in the active harness assembly. To route wires and cables, you can use one of three methods:

- | | |
|-----------------------|--|
| manual | Inserts individual wires and cables into selected segments one at a time. Segments can be discontinuous. |
| semi-automatic | Inserts selected wires and cables into continuous segments. |
| automatic | Automatically inserts all unrouted wires or cables into the segment that is closest to the wire connect points (pins) and also represents the shortest path. |

When routing or unrouting you can select the object to use before opening the dialog box, or you can select the objects using the dialog box tools.

Define Manual Routes

Use the Route tool to route individual wires or cables manually into a single segment. The segments can be discontinuous and can be used for incremental routing to create networks with gaps.

The workflow for manually routing cables and wires is the same. The difference is that when you select a wire, it is routed independently of all other wires. When you select a single cable wire, all cable wires associated with the same cable are routed and unrouted together.

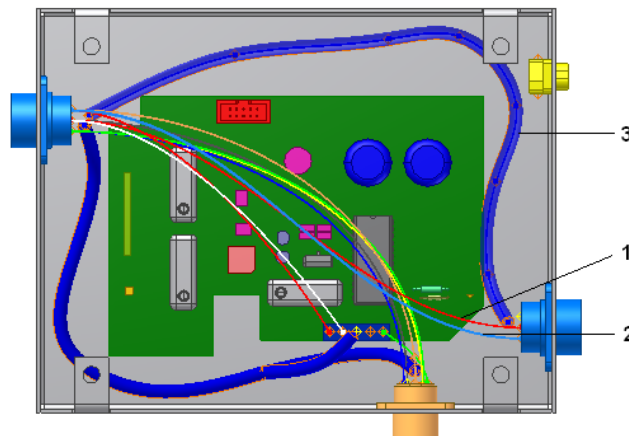
Route a Belden cable into a single segment

- 1 Activate Harness Assembly2.



- 2  On the ribbon, click Cable and Harness tab ► Route panel ► Route.

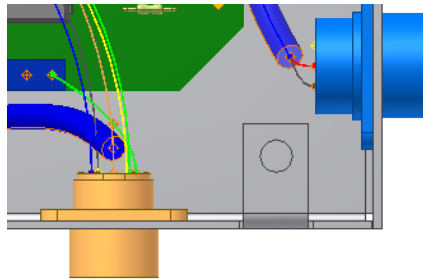
- 3 In the graphics window, select a cable wire (1 or 2) to route.



- 4 Select the Single Segment check box.

- 5 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, and 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.

- 6 Save the harness assembly.

Define Semi-automatic Routes

When creating a semi-automatic route, use the Route tool to select one or more wires or cables, and the first and last segment within the desired network. The system looks for the shortest continuous path connecting the two segments.

If more than one path exists between the two segments selected, a dialog box is displayed for you to cycle through the possibilities, from shortest to longest. The corresponding paths are highlighted in the graphics window.

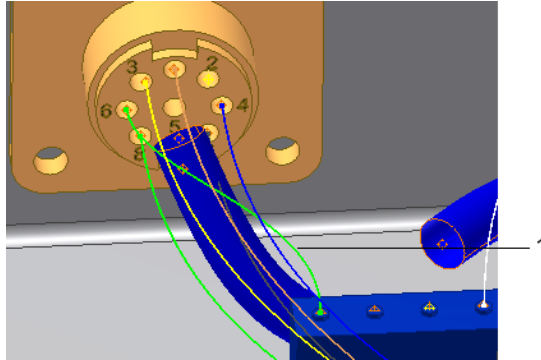
Route Wires

In this exercise, you select a wire and route it manually.

Route a wire into multiple segments

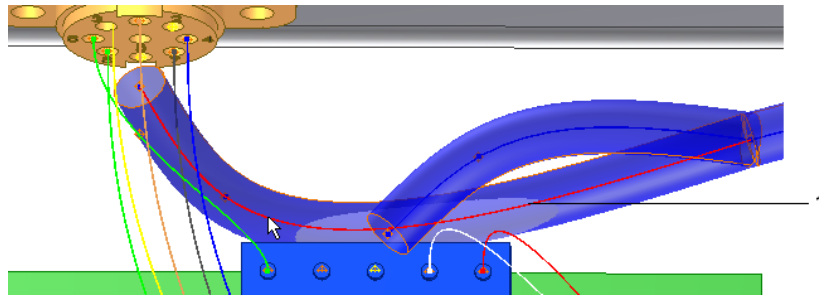
- 1 Double-click Harness Assembly1 if it is not already active.
- 2 Click Cable and Harness tab ► Route panel ► Route.
- 3 On the Route Wires dialog box, verify that the Wires button is selected.

- 4 In the graphics window, click the wire (1) as shown in the illustration. In the browser, this wire is named Wire1.



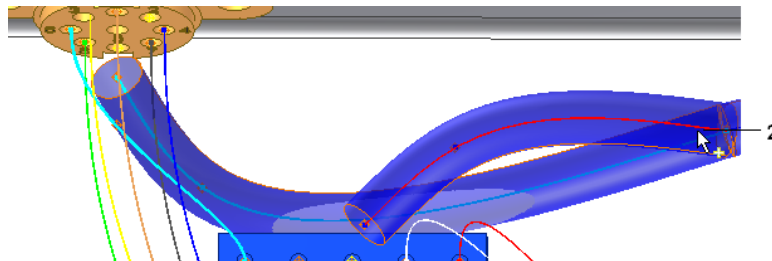
On the Route Wires dialog box, the Wires button changes from red to white to indicate that the wire selection is complete.

- 5 Click the First Segment button, and then click the first segment (1) as shown in the following illustration.

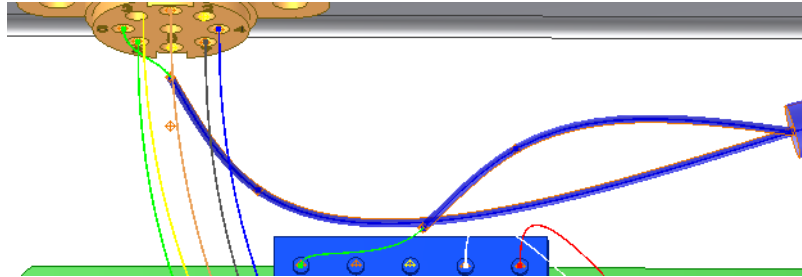


The First Segment button changes color, and the Last Segment button is depressed.

- 6 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.



Define Automatic Routes

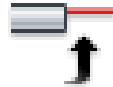
You can route all unrouted wires and cable automatically, or you can route selected wires and cables. When automatically routing, the system checks for the harness segment ends closest to each end of the object being routed. When the segment ends are identified, the system examines all segments found, and then looks for the shortest path. After finding the shortest path, the objects are routed.

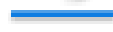
If the harness segment endpoints found are within 0.005 meters of one another, they are considered to be the same distance. If there is no connection between any of the identified start and end segment points, such as when the endpoints identified are from different harness segments, or when the same segment point is identified for both the start and endpoints, the route fails. In this case, you can route the objects using manual methods.

In this exercise, you route any remaining unrouted wires and cable wires into the segments.

Automatically route all wires and cable wires

- 1 With Harness Assembly1 still active, click Cable and Harness tab ►



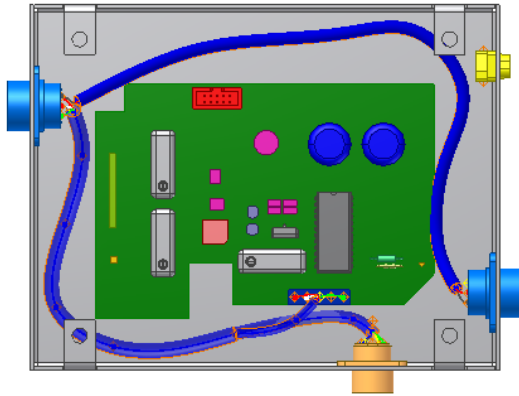
Route panel ► Automatic Route  .

- 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.



View Wire and Cable Paths

As a result of routing and unrouting wires and cables in a complex assembly, the path a wire or cable takes is not always easy to see. To highlight all of the segments a routed object passes through, select View Path from the context menu.

View a wire path

- 1 With Harness Assembly1 still active, select Wire1. You can select the wire from the graphics window or browser.

NOTE To select the wires from the graphics window, set the Select tool to Select Sketch Features.

- 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.

Unroute

Using a combination of tools on the Unroute Wires dialog box, there are several options for unrouting wires and cables. You can unroute:

- Selected wires or cables from all segments (default)
- Selected wires or cables from selected segments
- All wires or cables from all segments

When wires or cables are unrouted from the specified segments, the wires behave as if the segment was deleted, and return to their point-to-point connection position.

All work points on wires or cable wires are deleted when the wires are unrouted from all segments. The work points remain when a wire or cable wire is unrouted from selected segments only.

When the last object is unrouted from a segment that is set to calculate size from wires (including cable wires), the segment diameter does not change from the diameter it had with that last object in it. The current segment is not resized when all wires and cables are unrouted at once.

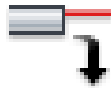
Rather than unroute an object entirely, it is often necessary to unroute it from one or more individual segments. In the following exercises you unroute a selected wire, and then unroute all remaining wires. You also unroute a cable.


Unroute Wires

Individual, or discrete, wires are unrouted independent of one another.

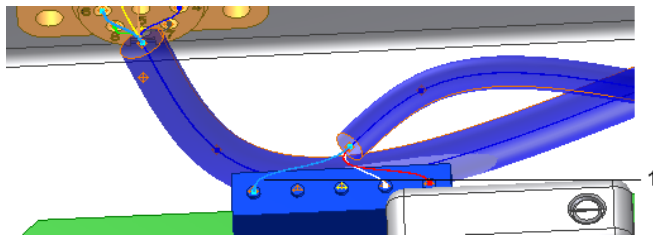
Unroute selected wires from selected segments

- 1 With Harness Assembly1 active, click Cable and Harness tab ► Route



panel ► Unroute  .

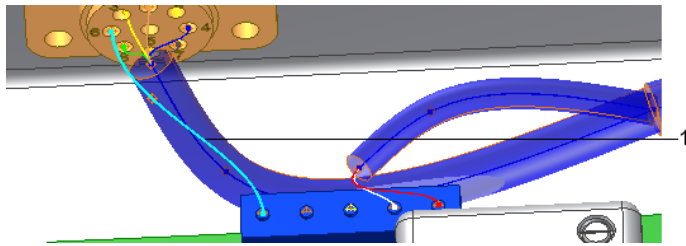
- 2 Verify the All Wires check box is not selected. In the graphics window, select Wire1 (1) as the wire to unroute.



NOTE To select the wires in the graphics window, set the Select tool to Select Sketch features.

The Wires button changes color and the Segments button is depressed.

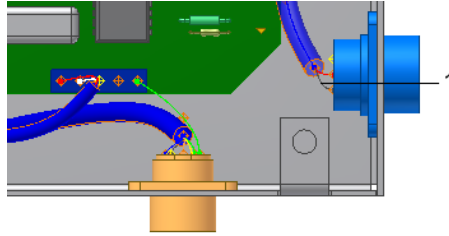
- 3 Clear the All Segments check box.
- 4 In the graphics window, click the segment closest to connector 360124.
- 5 Click the segment branch, and then click OK.
Wire1 (1) returns to its unroured state.



Unroute Cables

To unroute a cable, you select a single cable wire, and then all cable wires associated to the same cable are unroured together.

- 1 Activate Harness Assembly2.
- 2 Click Cable and Harness tab ► Route panel ► Unroute, if the Unroute dialog box is not already open.
- 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.

Unroute All Wires or Cables from All Segments

In this exercise, you unroute the remaining wires. If the dialog box is not still open from the previous exercise, select the Unroute command.

Unroute all wires

- 1 Activate Harness Assembly1.
- 2 On the Unroute dialog box, verify that both the All Wires and All Segments check boxes are selected.
- 3 Click OK.
The wires are unrouted and wire lengths update to reflect the change.

Practice Your Skills

Use the skills and tools learned in the previous examples:

- 1 Reroute the wires and cables you just unrouted.
- 2 Save your work.
The data files with routed wires and cables are required to create the nailboard in [Work With Nailboards and Drawings](#) on page 341.

Work with Splices

16

In Cable and Harness, splices are a virtual representation of the splice location and the wires that are joined together to make a path for electricity to flow.

In this chapter you learn how to create and manipulate splices, add properties to the splice or splice pins, and set embedded length to indicate splice type.

About Splices

Each splice contains two pins, is created with a default size, and is offset a specified distance from selected face geometry. It can also be associated to selected geometry. Splices are typically placed on segments with little or no bending at the splice location. The embedded length is set on splices or splice pins.

A default splice is provided in the Cable and Harness Library. You cannot change settings on the default splice, but you can define your own splice, and then set different defaults as needed. To change settings on splice occurrences, right-click, select Harness properties from the context menu and change settings on the Splice Properties dialog box.

Once a splice is positioned in the harness assembly, it can be included in standard assembly drawings and nailboard drawings. In the nailboard drawing the splice is represented with a unique color and symbol. Splice information is also included in reports and when saving data to the XML format.

Recommended Workflow

- Add a splice definition to the Cable and Harness Library.
- Activate the harness assembly.

- Create a splice.
- Accept or change the splice Refdes and the selected splice.
- Optionally, click the Properties button to view the properties on the selected splice.
- Locate the splice offset from a face.
- Optionally, add or modify custom properties on the splice.
- Import the wires onto the connectors.
- Splice the wires.
- Redefine the splice onto the segment.
- Route the wires.
- Optionally, add virtual parts.

If you prefer, you can create the splice on a segment, import wires onto the splice, and then route the wires. You can also connect wires to splices manually.

Create Splices

Use the Create Splice tool on the Cable and Harness tab to add a splice to a harness assembly. Valid geometry for creating a splice includes:

- An arbitrary point offset from a face
- A wire, including wire work points
- A segment, including the segment endpoints and work points
- Wire stubs (the portion of the wire from the end segment to the pin), exposed wire pieces (sections of wires displayed between gaps in segments), unrouted wires

Like segments, you can also associate a splice to other model geometry such as a vertices, sketch points, or center points.

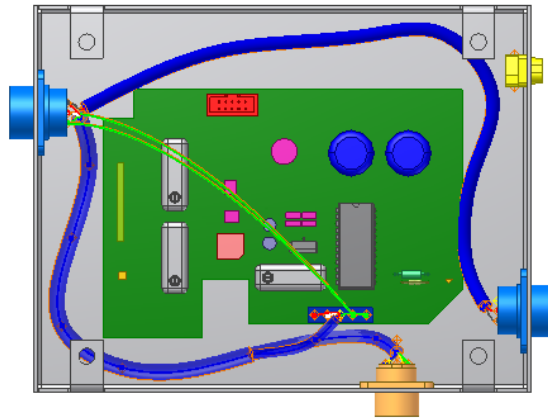
NOTE You cannot place splices on cable wires, ribbon cables, connector pins, another splice, or dangling wires. We recommend that you do not place splices at segment junctions.

Insert Splices

In this exercise you create the splice at an arbitrary location, and then add two wires to the splice.

Insert a splice

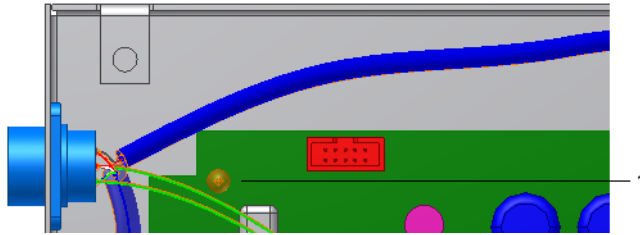
- 1 In the browser, double-click Harness Assembly1.
- 2 Create a Generic 22AWG-GRN wire from U7 Pin 3 on Connector 360575:1 to J12 Pin 2 on LTP:1.
- 3 Create another Generic 22AWG-GRN wire from U7 Pin 5 on connector 360575:1 to J12 Pin 2 on LTP:1, and then click OK. The new wires are shown in the following image.



- 4 On the ribbon, click Cable and Harness tab ► Create panel ► Create Splice .
- 5 On the Create Splice dialog box, accept the defaults.
You can accept or change the default RefDes, accept the default splice, or select a custom splice from the library. You can also click Properties to view and change existing custom properties or add your own.
- 6 Click the Select Location tool, if not already selected.
A green circle that represents the splice is attached to the cursor. A line representing the offset distance extends from the splice object.

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.

- 7 Click in the graphics window to place the splice.
The splice (1) is shown in the following image.



- 8 Click OK.

Modify Splices

After you place the splice, you can:

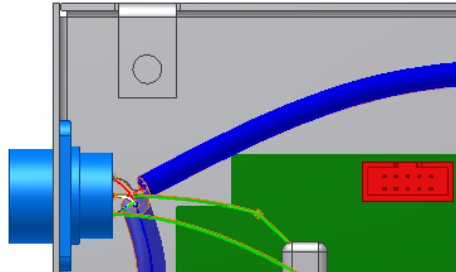
- Add wires to the splice manually or by importing a wire list.
- Add or modify Custom properties on the splice occurrence or splice pins.
You cannot change splice pin names or default properties on the splice.
Default properties must be changed on the splice definition in the library.
- Redefine the splice location.
- Delete the splice.

Splice Wires or Add Wires to the Splice

To visualize the best placement of the splice in the harness, place the splice on the wires before redefining the splice onto the segment.

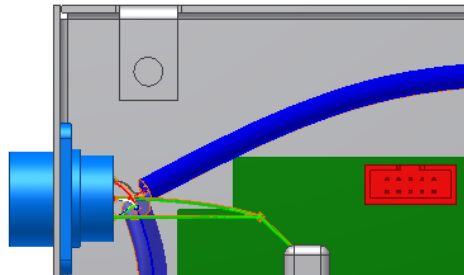
- 1 In the browser or graphics window click the splice, right-click, and then select Splice Wire on the context menu.
- 2 Click Wire 2, the wire to splice.

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 Click OK on the confirmation dialog box to remove redundant wires. Redundant wires (wires that share a common point when spliced) are deleted.

NOTE Redundant wires are only removed when manually splicing wires. Importing wires or placing wires on splices using Create Wire or Edit Wire does not remove redundant wires.



Redefine the Splice

After visualizing the splice on the wires, you can redefine the splice to a new location. You can redefine a splice on a wire:

- Onto another wire.
- Onto a segment

- At another location, including another location along the same wire

You cannot redefine the splice to a location off of the wire, because the wires connected to the splice move with the splice. You can only delete the splice.

You can also redefine a splice on a segment:

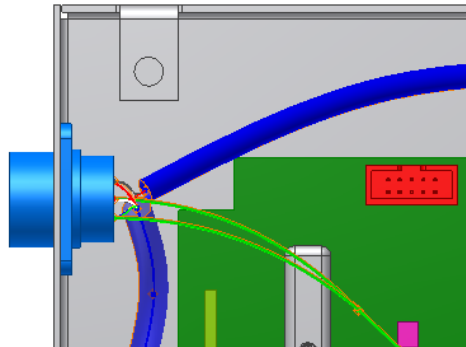
- To a location off of the segment.
- Onto another segment.
- At another location along the same segment.

If the new location is on a different wire or segment:

- The new wire or segment is split, and each object gets a unique name. For example, Segment1 becomes Segment1 and Segment1_1.
- If the initial location was on a segment, the existing segment is healed and any wires attached to the splice are unrouted.
- If the initial location was on a wire, the wires move with the splice to the new location.
- Any work points on the spliced wires are retained.

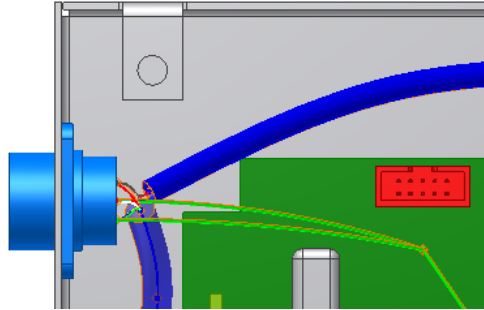
Redefine a splice along the wire or to an arbitrary location

- 1 Right-click the splice, and then select Redefine Splice on the context menu.
- 2 Click a new location along the same wire.

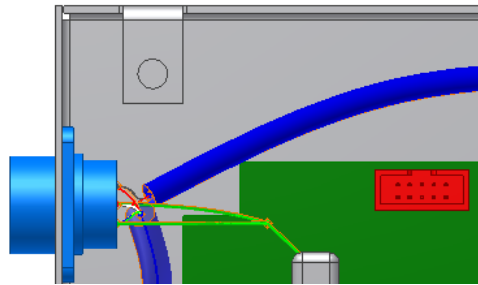


- 3 Right-click the Splice again, select Redefine Splice, and then select a new location in the enclosure.

Notice how the wires move with the splice.



- 4 Use Undo twice to return the splice to the original location.

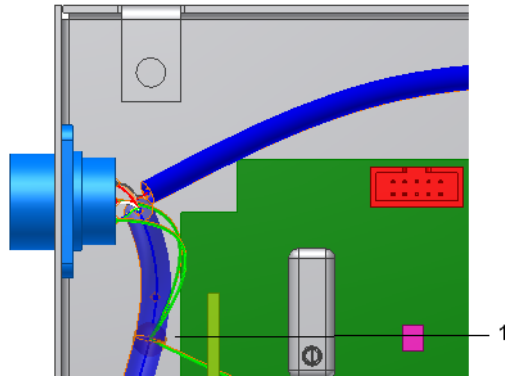


Redefine the splice onto a segment

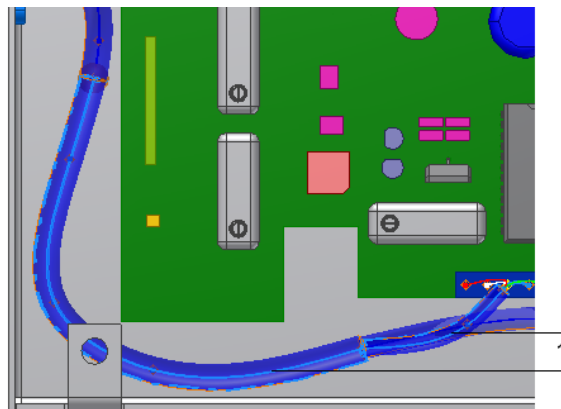
- 1 Right-click the splice in the graphics window, and then select Redefine Splice on the context menu.

NOTE We recommend that you place splices on segments. In this exercise, you redefine the splice onto a segment, and then route the wires.

- 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.



- 6 Save the harness assembly.

Splice Properties

Like all other harness objects, splices and splice pins contain a set of properties that provide a complete description of the splice. Splice occurrences inherit the default properties that are set on the library definition and the general properties set at the part level. Default settings can only be changed on the definition in the library. When unique properties are needed for the splice definition, you can create and add a new splice definition to the library. When unique properties are needed for the splice occurrence in an assembly you can add custom properties.

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. Notice that there is an embedded length value of 0. Also notice that 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.

Control Length for Splices

The embedded length property is key for splices, because it indicates the different splice types. There are two types of splices:

- | | |
|----------------------------|--|
| Butt splices | A physical object where the wires are inserted into the body of the splice to form the electrical connection. Wires do not overlap, thus the embedded length is zero or a negative number. |
| Overlapping splices | Wires are either soldered or ultrasonically welded together. The embedded length value is the amount that the wires overlap. |

For both splice types, the splice pins are located in the center of the splice. All wire lengths are calculated to the splice pins and the embedded length of the splice is added to the wire length to determine its adjusted length value.

This ensures that the proper wire length is added to each wire connecting to the splice.

To change the embedded length for a splice, create a splice in the Cable and Harness library, and then enter the needed embedded length value. You cannot change the embedded length on the Default Library Splice.

Search for “length, control in splices” in the Help index for more information and examples on setting embedded length and negative length conditions.

Delete Splices

If you delete a splice with two wires attached, the wires heal. If there are three or more wires, redundant wires are deleted along with their associated properties. The logical connection of the signal carried by the wires is maintained.

If you delete a splice from a segment, the segment heals and all routed wires remain routed. If a segment or wire that has a splice on it is deleted, the splice remains. Any custom virtual parts assigned to the splice are also deleted.

Practice Your Skills

- 1 Create a splice.
- 2 Import wires onto the splice.
- 3 Manually attach a wire to the splice.
- 4 Delete a splice from a segment.
- 5 Delete a splice from a wire.

Work with Ribbon Cables

17

In this chapter you learn how to add a raw ribbon cable definition to the Cable and Harness Library, create a ribbon cable between start and end connectors, and then modify it to control the shape and direction through the assembly.

About Ribbon Cables

Like wires and cables, ribbon cables are retrieved from the Cable and Harness library and placed in the harness assembly. To insert a ribbon cable, you select the ribbon cable to insert, identify the start and end connectors, specify how the ribbon cable engages with the connector, and then add intermediate work points to control shape or indicate the location for a fold.

Once the ribbon cable is in the assembly, there are many ways to modify it.

- Change connectivity: select different connectors, change direction in which the ribbon cable attaches to the connectors, move conductor one to a different pin, change the orientation.
- Add, delete, move, and redefine work points.
- Add single and double folds.
- Adjust twist at selected points.

In the following exercises, you create a ribbon cable with a single fold between D-Sub Ribbon on the enclosure and RC Male Connector on the PCB board.

Define Raw Ribbon Cables

Like traditional cables, there is no default ribbon cable. Before creating a ribbon cable in the harness assembly, the definition for the raw ribbon cable must exist

in the Cable and Harness Library. You add raw ribbon cable definitions to the library the same way you add other harness objects.

- 1 Activate the Ribbon Cable harness assembly.
- 2 Click the Cable & Harness Library tool.
- 3 Select Raw Ribbon Cable as the harness object type.
- 4 Click the New tool and examine the properties that are unique to ribbon cables such as the ability to set different colors for conductor one and the ribbon cable body.
- 5 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: .05 in.
Height: .035 in.
Width: .450 in.
Gauge: 28
(Core size is optional. Leave it blank.)
- 6 Optionally, click the Custom tab to add additional properties.
- 7 Click Save and the new raw ribbon cable is added to the library in the Tutorial category.
- 8 Click Close.

Place Connectors from Content Center

The Content Center contains some generic connectors for your use. You can use these connectors or create custom connectors that you author and publish to Content Center for reuse.

For more information on authoring and publishing connectors and using Content Center, see the Help.

In this exercise, you use a 10-pin ribbon cable connector from Content Center.

Locate and place the connector

- 1 Activate the Ribbon Cable harness assembly, if not already active.



- 2 On the ribbon, click Assemble tab ► Component panel ► Place from Content Center .

- 3 On the Place from Content Center, Category View pane, navigate to Cable & Harness ► Connectors ► Ribbon Cable.



- 4 Click Family Preview to view the family members for the ribbon cable connector.

- 5 In the preview table, note the Member name for the 10-pin connector. In this case, it is the first member in the list.

- 6 Double-click the image of the ribbon cable connector to select it.

- 7 On the Ribbon Cable Connector dialog box, ensure that the following are selected:

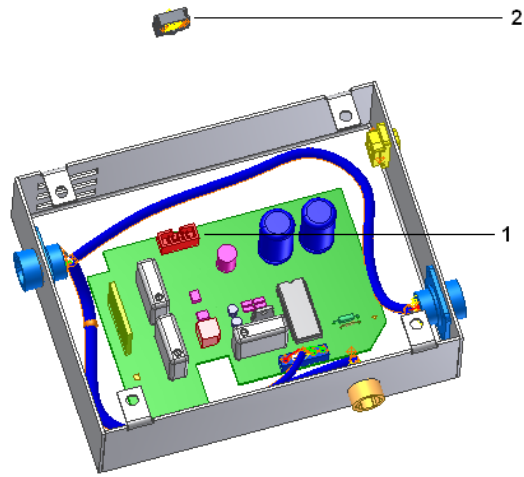
- Ribbon Cable Connector-01
- As Standard

- 8 Click OK.

In the graphics window, the part is attached to the cursor.

- 9 Rotate the enclosure to get a better view of the RC Male Connector (1).

- 10 Click in the background of the graphics window to place the part (2). Exact placement is not important.

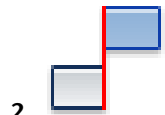


11 Right-click and select Done.

In this exercise, you place three mate constraints to constrain the ribbon cable connector in place.

Constrain the connector

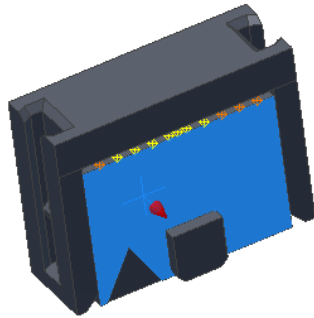
1 Activate the top-level assembly, *Enclosure_Assembly.iam*.



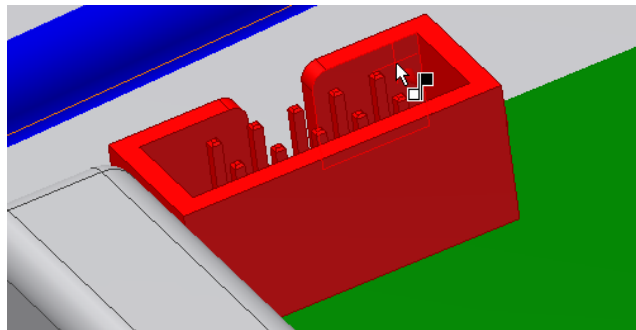
2 On the ribbon, click Assemble tab ► Position panel ► Constrain.

3 Ensure the Show Preview check box is enabled.

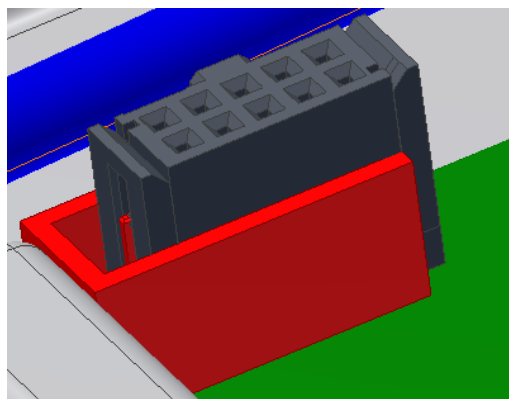
4 Ensure the Mate constraint tool and the first selection button is selected, and then click the face of the ribbon cable connector.



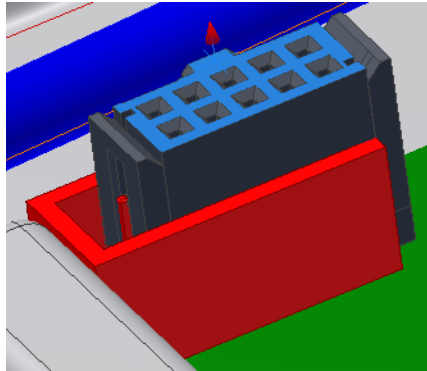
- 5 The second selection tool is now active. Click the wall of the RC Male Connector as the geometry to mate to.



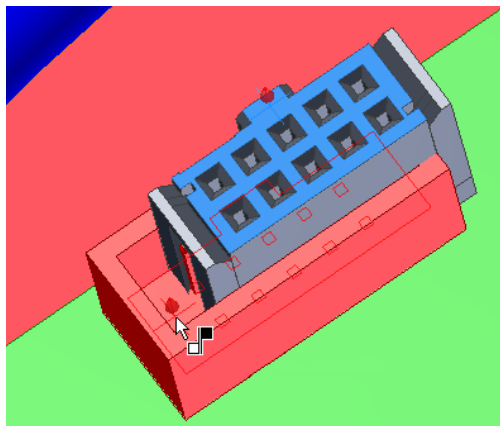
- 6 When the preview shows the connector has moved into place, click Apply to create the first mate constraint.



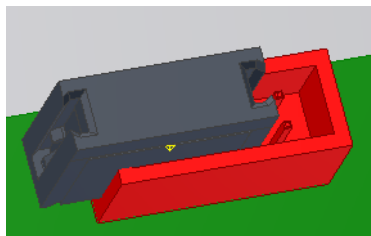
- 7 For the second mate constraint, click the top face of the ribbon cable connector.



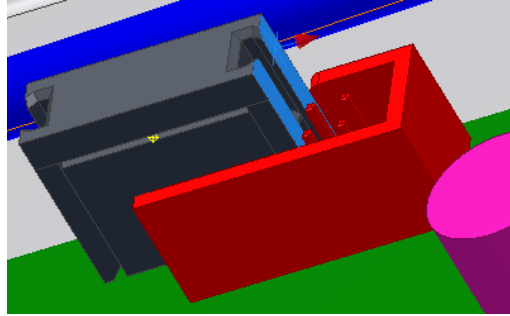
- 8 Click the bottom face of the RC Male Connector.



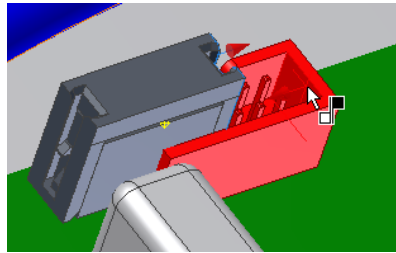
- 9 When the preview shows the connector has moved into place, click Apply to create the second mate constraint.



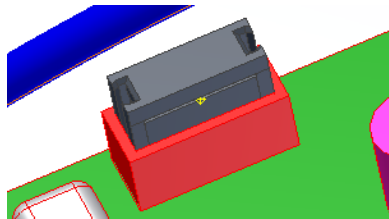
- 10 For the final mate constraint, click the right side of the ribbon cable connector.



- 11 Click the inside face of the RC Male Connector.



- 12 When the connector moves into place, click OK.



Create Ribbon Cables

You create a ribbon cable between a start and end connector. When selecting the connectors for a ribbon cable, only valid (authored) ribbon cable connectors highlight. Once you select the connectors, each mouse click in the graphics window creates a work point on the ribbon cable. As with other harness objects, you can offset work points from a face and adjust the offset

as you work. Or you can associate work points with existing geometry for more precise placement.

Create the ribbon cable

- 1 Activate the Ribbon Cable assembly.

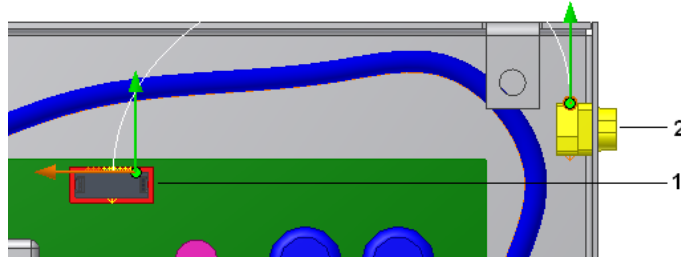



- 2 On the ribbon, click Cable and Harness tab ► Create panel ► Create Ribbon Cable.

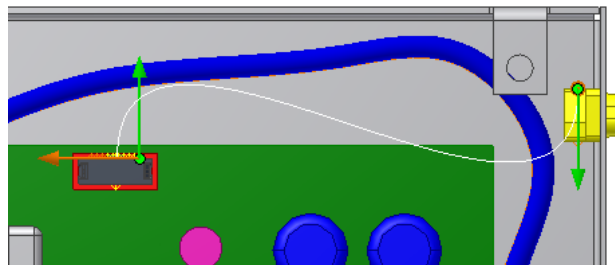
- 3 Ensure that the raw ribbon cable definition you created earlier is the ribbon cable that is selected in the dialog box.

- 4 In the graphics window, select the start connector (1) and the end connector (2).

Notice the directional arrows and the spline that 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.

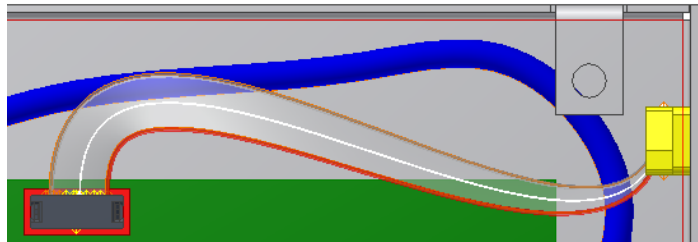


- 5  The outward direction on the end connector is not correct. Click the outward direction tool for the end connector to change the direction.



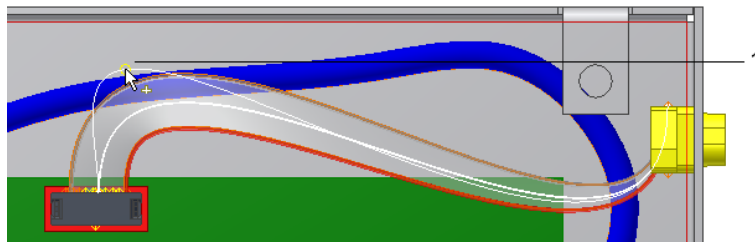
- 6 Click OK.

You are still in ribbon creation mode and can add intermediate points to change the current shape. Add the points in sequence from the start connector to the end connector.

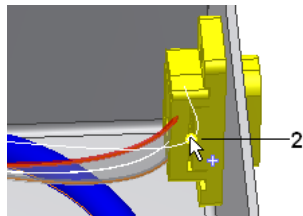


- 7 Right-click, select Edit Offset, and enter .500 in.

- 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.

Adjust Ribbon Cable Orientation and Shape

The ribbon cable must align with the slot at connector D-Sub Ribbon. To control the orientation of the ribbon cable, you can add single or double 90-degree folds at work points that exist along the ribbon cable.

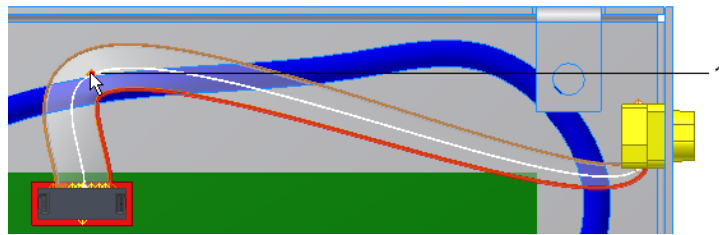
A ribbon cable with a fold is a single piece of raw ribbon cable. For modeling purposes, the ribbon cable is not continuous through the fold. When you add a fold, three separate entities are created: the fold entity and the ribbon cable piece to each side. Each entity is represented as a node in the browser.

For this exercise, you add a single fold and then adjust the twist.

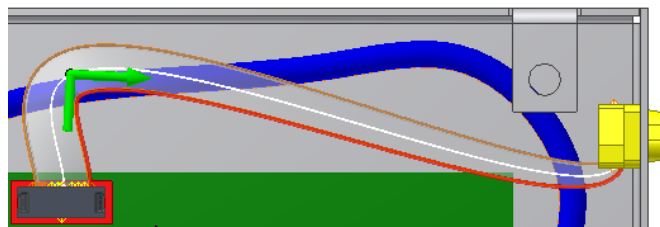
Add a fold




- 1 Click the Cable and Harness tab ► Create panel ► Create Fold, and then click the first point (1) you created.

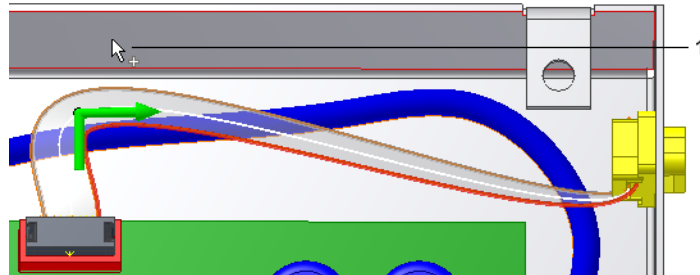


An arrow appears at the fold point indicating the direction of the next work point on the path of the ribbon cable, which is the correct direction for this fold.



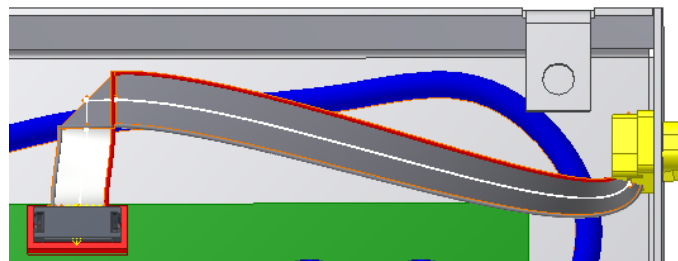
- 2  Under Alignment in the dialog box, click the Shaft tool, and then click the face of the enclosure (1).

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. Ribbon cable work points cannot be rotated. For folds, only the work point used to create the fold can be manipulated.



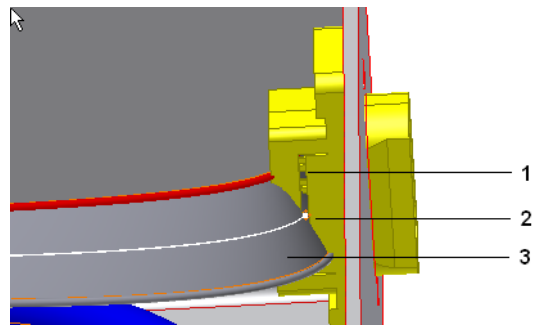
- 4 In the browser, expand the Ribbon Cables folder and the ribbon cable. You should see Ribbon1, Fold1, and Ribbon2.

Adjust twist

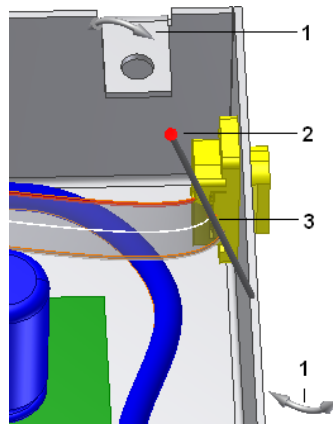
The ribbon cable body aligns parallel to the slot in the D-Sub Ribbon Cable connector. To adjust the ribbon cable at a selected point, use Twist Control.



- 1 With the ribbon cable harness assembly still active, zoom into the D-Sub Ribbon cable and notice how 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.

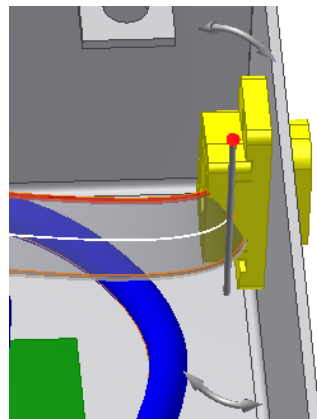


- 3 The Twist Control appears at the selected point.
The rotation handles (1) enable you to change the current orientation.
The red ball (2) is the conductor one indicator. The orientation bar (2) represents the orientation of the surface on the ribbon cable at the selected location.

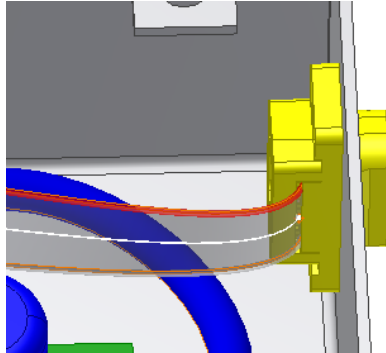


- 4 Click and drag a rotation handle until the ribbon body is aligned with the slot, and release the mouse button.

NOTE Click the minus (-) and plus (+) on the keypad to decrease and increase the size of the Twist Control, if appropriate.



- 5 Right-click and select Apply.
The ribbon cable body recomputes to the new location.



- 6 Save the assembly.

Practice Your Skills

- 1 Edit the ribbon cable and change the orientation and direction of the ribbon cable as they attach to the connectors.
- 2 Edit the fold. Change the fold type, experiment with the direction and alignment.
- 3 Add and modify the work points on the ribbon cable.

Generate Reports

18

Harness assemblies can be documented in reports. These reports are dependent on specified properties and can be configured to meet your needs.

In this chapter, you learn about the Report Generator user interface, and how to format and generate reports.

About Generating Reports

Several standard report types, such as wire run lists, bills of material, and connector tables, can be run for a harness assembly. The information that is processed is based on the property set for each harness component.

When a report is generated, the output file is saved to a specified location, and one or more reports are displayed in the Report Generator document window for viewing. Report output files can be imported into an Excel® spreadsheet or similar application. Reports can also be added as table annotations in a nailboard drawing.

A set of sample configuration files is installed with the application. You can rename and customize the sample configuration files to produce reports that meet your needs. You can also create your own configuration file from a blank file.

NOTE The Report Generator can only view report configuration and .csv files that are created in the Report Generator.

Workflow for Harness Reports

The following are the basic steps to create and generate a harness report.

Workflow: Generate reports

- 1 Activate the harness assembly to report on.
- 2 Select Report from the Cable and Harness tab.
- 3 Specify a configuration file (.cfg) to format and process the data.
- 4 Modify the configuration file to meet your needs.
- 5 Select the files to generate.
- 6 Generate the reports.

Use the Report Generator Dialog Box

Before you begin processing reports, explore the commands on the Report Generator dialog box. To display the Report Generator dialog box, click Cable and Harness tab ► Manage panel ► Report.



Report

The Report Generator user interface consists of a menu bar, toolbar, and document window. When the dialog box is first displayed, the document window is empty. After a report is generated, the text files containing the output from generated reports are displayed. The toolbar and menu bar contain features for generating and modifying reports, and several standard Windows features.

The following commands are available on the Report Generator toolbar. Pause your cursor over the toolbar to view the tooltips.



New Text File	Creates a new, blank text file.
Open Text File	Opens an existing text file.
Save Text File	Saves the selected text file using the name and location specified.
Cut, Copy, Paste	Cuts, copies, or pastes the selected text.
Print	Sends the selected text files to print.

Create Report	Sets the files to use for processing reports on the active harness assembly, and creates a report output file. Also removes reports from the list, and can rename and re-locate the output file.
Edit a configuration (.cfg) file	Selects the configuration file to edit, and then displays the configuration file dialog box with options to modify the format and specify information to include for the selected file.
Cascade Text Files	Arranges text files in the document window to overlap slightly.
Tile Text Files	Arranges text files in the document window to display with no overlap.
Close Text Files	Closes all files displayed in the document window.

Format Reports

Each report type requires a configuration (.cfg) file to process the information from the harness assembly and create a report. To create a configuration file or modify an existing configuration file, use the Edit configuration file command on the Report Generator toolbar.

Sample .cfg files for each of the main report types are provided in the *Tutorial Files\Cable & Harness\Report Generator* folder where you installed the Autodesk Inventor® software. We recommend that you use these files whenever possible to reduce input time and errors.

You can also create a new blank configuration file.

Use Sample Configuration Files

The easiest way to create a report is to use one of the sample configuration files installed with the application. If the sample file does not meet your needs, use the sample file as a base, modify, and organize the data to include the information you need, and then save the file using a unique name.

All sample report types are set to print as comma-separated files. To view the settings while reading the descriptions, select the required configuration file, and then click each tab.

The following are sample files and their output:

PartsList.cfg

Lists the electrical parts attached by a wire or cable wire in the harness assembly. The format is set to include a placeholder for repeated part names, and to print the column headers and the text displayed in the Info tab.

Part_Splice_Wire_Cable_BOM.cfg

Outputs the quantity of each part type that is attached by a wire or cable wire in the harness assembly. The report also includes a total length calculation for each library wire type used in the harness. The filter is set to exclude objects from the BOM (denoted by a specific property BOM=FALSE name/value pair), and to use the Part Number to total the number of like objects. Although length is required to total the amount of wire in the harness assembly, the column is not printed. The total amount of each wire and cable is included in the Quantity column.

To create a full electrical bill of materials (BOM) for a harness assembly, you can use the Report Generator or the Autodesk Inventor bill of materials. The Autodesk Inventor parts list includes only the harness assembly, and none of its contents. To include harness parts in a BOM, add them as virtual components in the Cable and Harness assembly.

NOTE Virtual parts and ribbon cables are not included in bill of material reports created with the Report Generator. All harness objects, however, are included in bill of materials and parts lists created using standard Autodesk Inventor features.

Wire Run List.cfg

Outputs a from/to list for each wire, including cable wires, in the design. It includes the connection, RefDes, and pin name at both ends of each wire, as well as the length of the wire in the specified units. The table uses several link types to get the RefDes and pin name information for each wire end. The wire lengths include a round-off value that is used before converting the wires to feet. The format is set to sort multiple columns.

Connector.cfg

Creates one file for each electrical component or splice with a RefDes value, and wires or cables wires attached to them in the selected harness assembly.

The output file lists the pin name, Wire ID, and wire color. The filter is set to query only the parts with a part number property. The table is set to include, but not print, the RefDes property. The format is set to output a file for each RefDes and *<RefDes>.txt* is used as the default output name. If multiple wires or cable wires connect to the same pin, each wire appears on a new line, but the pin name is not repeated.

Segment Table.cfg

Creates a table with information about the physical characteristics of each segment in a harness.

Edit Configuration Files

To modify an existing configuration file, click the Edit configuration (.*cfg*) file command, select the file to modify, and then change the format and other aspects of the report using the configuration file dialog box.

Before editing a configuration file, click each tab of the dialog box to explore the available options. The following tabs are available:

Info	Displays and sets the header information for the configuration file.
Table	Sets the columns and information to use for the report.
Filter	Controls the types of harness components to include in the report.
Format	Refines the report output format including sorting, file format, and file print options.

Configuration File Options

The parameters on each of the four tabs in the Edit Configuration file dialog box are described in this section. Click each tab to view the parameters being described.

Info Tab

On the Info tab, view and set the information to appear as the file header. The file header often provides a brief description of the report. Other

information such as the name of the project or model used, and a date can also be included. Use the scroll bar to view all the information, if necessary.

Table Tab

On the table tab, set the columns and properties to use for parsing the report. Click the buttons to add and remove columns, and click the input boxes to add property values to each field. If an arrow is displayed, click the arrow to select from a list.

Keywords are used to perform certain functions, such as counting the number of like objects based on a specified property name.

See the Help for details about key words.

In the Table tab on the Edit Configuration dialog box, link types are used to obtain information on an object type, such as a pin or part, when running a report on a different object type, such as a wire. The following table shows the link types you can use for the different harness component object types.

Link Types	Segment	Wire	Pin	Part	Cable Wire	Splices
None	X	X	X	X	X	X
GETPART			X			
GETPART1, GETPART2		X			X	
GETSEGMENTS		X			X	
GETWIRES	X					
GETPINS				X		X
GETPIN1,GETPIN2		X			X	
GETPINSGETWIRES				X		X
GETPINSGETWIRESGETPIN2				X		X
GETPINSGETWIRESGETPART2				X		X

The rows of the table are defined as follows:

Column header	Sets the text for column headings. Enter any string.
Link type	Specifies whether to derive a property value from the main object or an associated object. When set to a link type, the property value is derived from an associated object.
Name	Sets the name of a single property whose value is written to the report. You can enter a property such as Color Style or Gauge, or a keyword. Keywords enable functions within the report generator. See the Help for details on keywords.
Column width	Sets the width of the column.
Subtotal function	Sets the value to place in the last row.
Property data?	Specifies whether the data displayed in this column is a harness property. "No" means that the property is a constant or special keyword.
Units	Specifies the units to use for that column. Autodesk Inventor unit settings are used by default. When the data type is a length type, round-off values are presented in the Round Off Row.
Round Up	Indicates the round up value used for wire and segment lengths. Values are dependent on the Unit setting.
<hr/>	
NOTE We recommend that you do not adjust the Unit and Round-up settings from the default. If adjusted, the lengths in harness assembly and nailboard could be different from the generated reports.	
<hr/>	
Print units suffix?	Sets whether to print a units suffix in the report.
Print column?	Sets whether to print this column in the report.

Filter Tab

The Filter tab options determine the harness objects and property name value pairs used to process data for the report.

Select objects to report	Sets one or more harness component objects to include in the report.
Object filters	Sets the property name and value pairs to include or exclude from the report. Also specifies the property value used to group and calculate like objects.

Format Tab

On the format tab, you set options that fine tune the report output. There are several options.

Multiple files	Enables a single report to output multiple files, and determines how to handle them. You can specify a property to use as the naming convention for created files. Valid properties include any property that appears as a column in the Table tab.
Sort by column	Specifies, in descending order of importance, columns used to sort the report data. Valid column numbers are dependent on the columns that appear on the Table tab. Enter at least one column number.
Placeholder	Specifies a placeholder when blank or repeated values are used consecutively in a single column. You set the text or character string to use as a placeholder in the report.

There are also several print options listed:

Print filter information	Writes the options selected in the Filter tab to the report.
Print header	Writes the name of the <i>.cfg</i> file and the directory information to the report. Clear the box to exclude this information from the report.
Print information	Writes the text in the Info Tab to the report. Clear the box to exclude the Info Tab text from the report.
Print column headers	Includes the column headers as defined on the Table Tab in the report. Clear the box to exclude the column headers from the report.

NOTE We do not recommend that you enable the Print filter information, Print header, or Print information options when you create reports for tables on the nailboard drawing since they affect the parsing of the data when the table is created.

Generate Reports

To generate a report, you select one or more configuration files to use, and then click OK to generate the report. The output file is saved to the specified

location, and one or more reports are displayed in the Report Generator document window for viewing.

A data type consistency check is performed as the reports are processed, and a log file is generated. The consistency check is performed for all properties on all harness objects associated with the selected harness assembly. For each property used on an object in a harness assembly, only one data type can be associated with the property name. For example, an inconsistency can occur if a connector part with a specific property and data type is placed into a harness assembly that contains a wire or segment with the same property name, but a different data type.

The log file details any properties with conflicting data types. If there is a failure in the data type consistency check, the location of the log file is listed, and you are prompted whether to continue.

Create Part and Wire Bills of Material

In this exercise, you generate a simple bill of materials report for the parts and wires in the harness design using the sample file *Part_Splice_Wire_Cable_BOM.cfg*.

Before you begin, verify that the harness assembly is saved and fully up to date (Update button is not active).

NOTE You can also create a complete bill of materials and parts lists using standard Autodesk Inventor features.

Create a BOM for parts and wires in a harness design



- 1 Double-click *Harness Assembly1*, and then click *Cable and Harness* tab ► *Manage panel* ► *Report*.
- 2 On the Report Generator dialog box, click *Create Report*.
- 3 On the Create Reports dialog box, click *Add File to List*, navigate to *Tutorial Files\Cable & Harness\Report Generator\Part_Splice_Wire_Cable_BOM.cfg*, and then 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.

- 4 To rename the file, double-click the output file name in the list, and then enter *Part_Splice_WireCable_BOM1*.
Click Open.
- 5 Click OK.
The report is processed and the output file is generated with the specified name and location, and then displayed in the Report Generator document window.
The path of the last selected configuration file is used as the default for subsequent file add operations.

Create Wire Run List Reports

A wire run list is a from/to list for each wire and cable wire in the design. It typically includes the reference designator and pin name or number for each end of the wire, and the wire part number and length information.

In this exercise, you modify the sample wire run list report to include more information, including link types.

Before you begin, verify that the harness assembly is saved and fully up to date (Update button not active).

Modify a wire run list

- 1 Make sure Harness Assembly 1 is still active.
- 2 On the ribbon, click Cable and Harness tab ► Manage panel ► Report, if Report Generator is not already open.
- 3 Click the Edit a configuration (.cfg) file command on the Report Generator toolbar, select *Tutorial Files\Cable & Harness\Report Generator\Wire Run List.cfg*, and then click Open.
- 4 On the Info tab, click in the input area before the existing text, and then enter Modified 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 Click the Filter tab, and then select the items to process for the report. Verify that Wire and Cable Wire are selected as the object types.

- 6 Click the Table tab. Scroll to the right and notice how columns 5 - 10 use link types to get the RefDes and pin name for each wire end.
- 7 Leave the Format settings as they are, and 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, navigate to the *Modified_Wire Run List.cfg* file in the Report Generator directory, and then click Open.
- 10 Click OK.

Create Custom Reports

In this exercise, you create a new, custom report by first creating a configuration file from a blank configuration file.

Create a custom report

- 1 Double-click *Harness Assembly1*, and then click Cable and Harness tab
➤ Manage panel ➤ Report.
- 2 Click the Edit a configuration (.*cfg*) file command on the Report Generator toolbar.
- 3 Browse to the appropriate file location.
- 4 Enter the name of the new configuration file as *custom_partbom.cfg* and click Open.
- 5 On the Info tab, enter *Customized Part Bill of Material, no quantity*.
- 6 Click the Filter tab, and then in Select Objects, select Part.
- 7 In Object Filters on the same tab, select All for both the Property Name and Property Value.
- 8 Click the Table tab, and then click Insert Column to add two columns. Insert each column before column 1.
- 9 Enter the values in the columns, as indicated previously. To enter data, click in the input box, and then enter the value. If an arrow is displayed, click the arrow and select from the list.

The first column lists the reference designator for the part, followed by the part number.

Each column can be configured, and any property name can be displayed.

	Column1	Column2
Column header	REFDES	PartNumber
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

- 10 On the Format tab, enter *two* as the first column to sort.
- 11 Select the following print options if they are not already selected:
 - Print filter information
 - Print header
 - Print information
 - Print column headers

- 12 Click OK to save the new configuration file.
Using the skills learned earlier, generate the report using this configuration file to view the resulting output. Each of the parts in the design display in an individual row.

Work With Nailboards and Drawings

19

You can document a harness assembly in a nailboard drawing or a standard Autodesk Inventor® assembly drawing.

In this chapter, you learn how to create a nailboard drawing, manipulate the harness shape, annotate the drawing with dimensions, and add connectors and other key information. You also learn to create a simple assembly drawing with the harness objects represented as centerlines.

About Nailboards and Drawings

A nailboard is a 2D flattened representation of the harness assembly that is used in the manufacture of a wire harness, cable, or ribbon cable. In a nailboard all harness wires, cables, and segments are flattened and drawn as straight lines in their original display colors. Ribbon cables are flattened and drawn as rectangles with appropriate lines indicating any folds. The work points defined in 3D, translate to 2D points that can be used to arrange the harness shape. The relative positions of the work points from 3D to 2D are maintained, including the distance between any of the two points.

Along with the 2D view, the drawing often contains annotations such as a bill of materials, parts list, a wire list, views of connectors, dimensions, pin numbers, and other attribute data.

Any changes made to the harness are automatically reflected in the nailboard the next time it is opened within the harness assembly, unless the assembly is set to defer updates.

You can also document the cable and harness assembly in a standard assembly drawing and either sweep the harness objects or include them as centerlines.

Nailboard Features

With the nailboard tools, you can:

- Create an accurate, associative 2D representation of a 3D harness.
- Set default display for segments.
- Set default display and distribution for cable wire stubs and wire stubs.
- Set default style, offset, orientation, and scale for connector views.
- Set whether to show or hide looms and labels. (If looms are shown, wires, cables, and segments to which a loom is attached are displayed in the loom color by default.)
- Rearrange the harness, segment, wire and cable wire stubs, ribbon cables, and labels (if displayed).
- Create base views.
- Create connector base views.
- Annotate the drawing with pin, wire, splice, ribbon cable, and connector properties.
- Add dimensions to actual wire and bundle lengths.
- Add dimensions to actual ribbon cable lengths and various points that identify a fold in a ribbon cable.
- Add other data needed for manufacturing.

Nailboard Environment

There are three ways to enter the nailboard environment:

- Click Nailboard on the Cable and Harness tab in an active harness assembly.
- Click Nailboard View on the Place Views tab in an open drawing.
- Open an existing nailboard.

In the nailboard environment, the Nailboard tab and browser are displayed and you can begin manipulating and annotating harness data.

To provide for maximum flexibility, the standard Autodesk Inventor Annotate and Place Views tabs are available from the nailboard environment. Using these features, however, is not recommended. We recommend that you create another view and make annotations as needed until the view contains the entities you require.

Pause your cursor over the commands to view the tooltips.



The commands on the Nailboard panels include:

- Edit Nailboard Sketch** Enters edit mode allowing easy access to the nailboard sketch.
- Pivot** Fixes a point as a pivot point for the purpose of arranging the wires, segments, and ribbon cables.
- Fan Out** Equally distributes wire or cable wire stubs about the endpoint of the segment based on the specified angle and sorting direction.
- Fan In** Collapses all wires or cable wires onto each other to form one entity that emerges from the endpoint of the segment. It is displayed using the characteristics of the longest wire or the segment.
- Broken Sketch Entity** Removes the specified length from the selected wire or segment to shorten the object to fit in the drawing.
- Harness Dimension** Like the standard General Dimension command, however, the dimension values on the wires, segments, and ribbon cables are overridden by the adjusted length values from the harness assembly.
- Property Display** Inserts the harness properties onto the nailboard.
- Report** Accesses the report generator from within the nailboard for creation of report files that can be used for annotating the drawing.
- Place Connector Views** Places one, some, or all connector views. You can specify the display characteristics for placement.
- Base View** Accesses the standard Autodesk Inventor Drawing Manager command within the nailboard environment for placement of connectors.

Create Table	Inserts a table onto the nailboard drawing. Not available in nailboard sketch.
---------------------	--

Nailboard Browser

The Cable & Harness and Model browsers in the nailboard environment contain the same items, except for the nailboard sketch, which exists only in the model browser.

Create Nailboards

By default, when you create a nailboard drawing:

- The nailboard view (or draft view) is created and placed in the nailboard sketch.
- The harness is placed in the center of the drawing.
- The wires, cables, segments, and ribbon cables are drawn with their original display color.
- The title block and border are placed on the nailboard based on the selected template.
- Splices are represented with a unique symbol and color.
- Wire and cable wire stubs are equally distributed (fanned out) in a clockwise direction about the segment endpoint at a 90 degree angle.
- No automatic scaling is done, which means that the harness objects are placed on the drawing even if they exceed the border size. It causes the nailboard to be a true one-to-one harness drawing.
- Looms and labels are not shown.

NOTE If needed, use the standard Autodesk Inventor Edit command on the draft view to scale the nailboard view.

The Enclosure Assembly contains three harness assemblies. You can create a nailboard drawing for each.

Set Display Behavior

Before you create a nailboard, you can verify or change the defaults that control:

- Lines for segments and wires, including cable wires
 - Fan state (fan in or fan out)
 - Display behavior for fanned in wire stubs and cable wire stubs
 - Sorting direction and angle for fanned out wire stubs and cable wire stubs
 - Color style for looms assigned to segments and wires/cable wires.
 - Offset, orientation, display style, and scale for connector base views.
-
- **NOTE** Once the nailboard is created, you can control the display state for looms and labels. Right-click the harness assembly in the Cable & Harness browser and select Nailboard Settings, Display tab.
-

Defaults for new nailboards are set using either the Harness Settings or Nailboard View dialog boxes, depending on how you create the nailboard.

In this exercise, you view the default display settings for the active assembly and create a nailboard drawing.

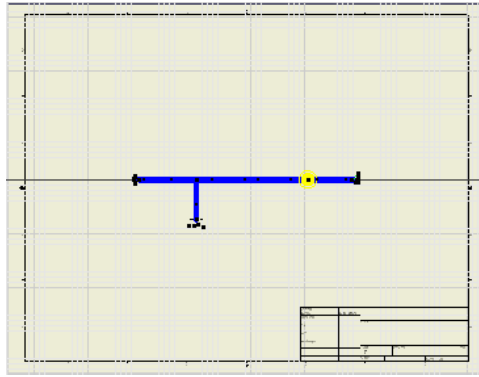
Create a nailboard for the first harness assembly

- 1 Double-click Harness Assembly1 as the assembly to document.
- 2 To view the default settings, right-click Harness Assembly1 in the browser, select Harness Settings from the context menu, and then click the Nailboard tab.
- 3 Review the loom color style settings on the Wires/Cables and Segments tabs, and then click OK or Cancel.



- 4 On the ribbon, click Cable and Harness tab ► Manage panel ► Nailboard.
- 5 On the Open Template dialog box, select the default template, *Standard.idw*, and then click OK.

The nailboard drawing view is created and the nailboard sketch is active. The harness is placed in the drawing as shown.



- 6 Right-click in the graphics window, and select Finish Sketch.
- 7 Save your work using the file name, *Harness Assembly1.idw*.

Manipulate the Harness

Although the default arrangement of the harness and harness objects require little manipulation after initial placement, options are provided to rearrange the harness objects.

For example, in a nailboard drawing, you can move the entire harness and rearrange segments or wire or cable wire stubs to position them for ease in annotating and viewing in the drawing. Any displayed labels are initially centered on the pin or point to which they are associated. You can also rearrange them for ease in viewing in the drawing.

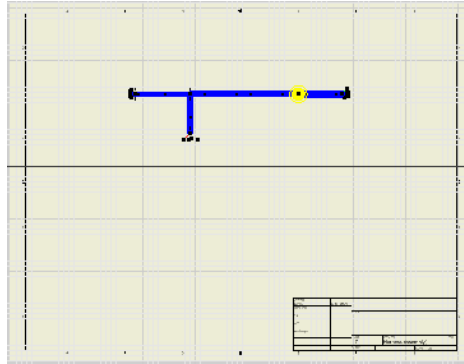
Move the Harness

You can move the entire harness to a new location on the drawing. You can move the harness in the drawing view or the nailboard sketch.

NOTE For a move to occur, no pivot points can exist.

Move the harness assembly in the drawing view.

- 1 Click the harness segment, and then drag the cursor to the location shown in the illustration.
- 2 Release the mouse to complete the move.
The harness is moved and redrawn in the selected location.




- 3 Undo to return the harness to its original position in the center of the drawing.


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.

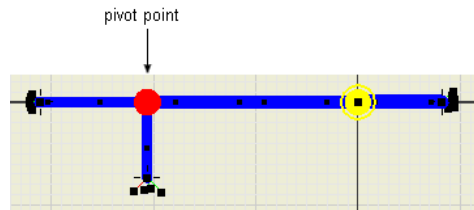
In this exercise, you arrange the harness segments in vertical and horizontal directions using grid snap for precision placement.

Arrange the harness segments

- 1  On the ribbon, click Nailboard tab ► Edit panel ► Edit.

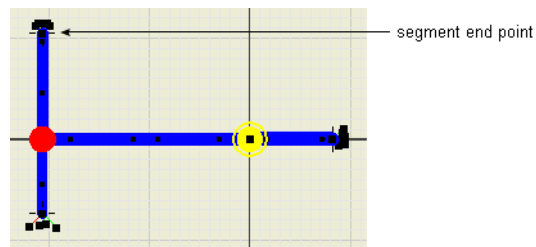
- 2  In the active nailboard drawing sketch, click Nailboard tab ► Edit panel ► Pivot.

- 3 In the graphics window, click the junction point for the harness branch.

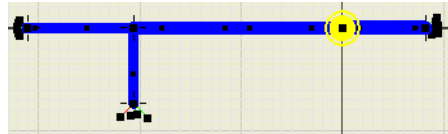


- 4 Click the point at the left end of the segment, and then drag until the segment snaps to the vertical position.

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.

Arrange the Wire Stubs

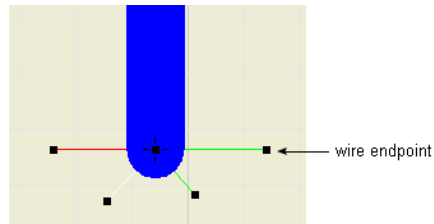
In a nailboard drawing, you can manually rearrange individual wires and cable wires by dragging them. You can also arrange multiple wire stubs automatically using Fan In and Fan Out. Exposed wires and wires ending on splices cannot be fanned in or out.

NOTE The wire stubs in this harness are short. If the wire stubs are important in your nailboard drawings, you can return to the harness assembly and move the final segment point farther from the connectors. Longer wire stubs are more visible in the drawing. The drawing updates automatically the next time it is opened or activated for editing.

Manually arrange 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.

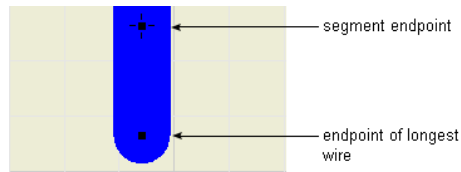


Automatically arrange multiple wire stubs (fan in and fan out)

- 1 Right-click the segment endpoint or any of the four wire endpoints, and then select Fan In/Out ► Fan In.

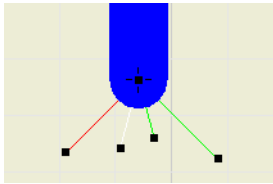
The wires are displayed using the color and diameter of the segment and the length of the longest wire.

NOTE If a loom is assigned to a wire or segment, they can display using the color of the loom or the wire. Color style must be set using the Wires/Cables and Segments tabs of the Harness Settings dialog box before creating the nailboard.



- 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.

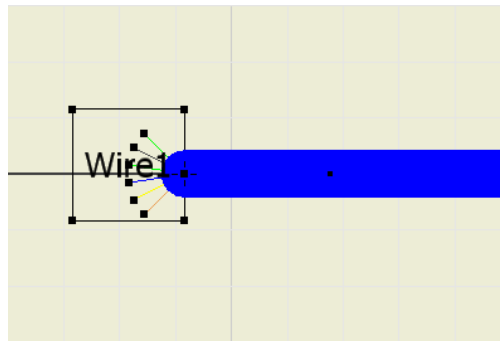
NOTE Right-click a wire in the fanned out state, and then select Fan Out from the context menu to change the sorting direction or angle between the outermost wires.



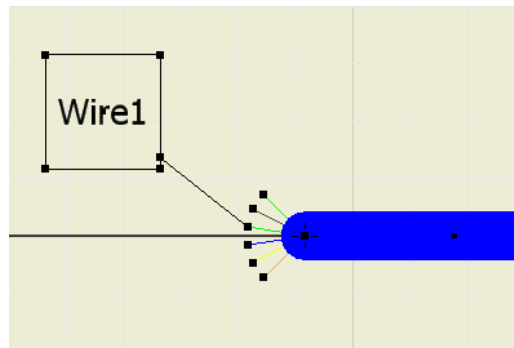
Arrange the Label

To arrange the label, first set the label to show in the nailboard. Once labels are set to show, you can manually rearrange individual labels by dragging.

- 1 In the browser, right-click Harness Assembly 1, and then select Nailboard Settings.
- 2 On the Nailboard Settings dialog box under Virtual Parts, click the Show Labels check box, and then click OK.



- 3 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.

- 4 For the purpose of this exercise, change the default label display setting back to hide. (Right click the harness assembly, select Nailboard Settings, click the Display tab, and clear the Show Labels check box.)

Change Nailboard Displays

Once the nailboard is created and the harness is placed, you can change the display for segments, wires, cable wires, wire stubs, and cable wire stubs using the context menu, and tools on the Cable and Harness tab. You can also change the display for connector base views.

Change Segment and Wire Line Display

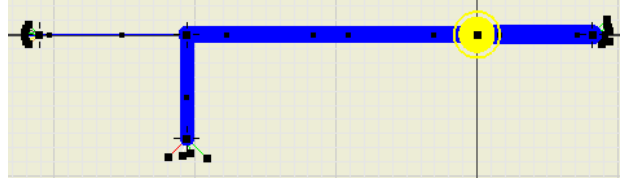
By default, nailboard wires and cable wires are set to display as thin lines. Segments are set to display as actual diameter. You can switch the display for an individual object in the nailboard drawing or all harness objects of one type.

To change the segment, wire, or cable wire display once the nailboard is created, right-click and use the display options on the context menu.

Change segment display

- 1 In the active nailboard drawing sketch, select the segment to change.
- 2 Right-click and clear the Display as Actual Diameter check box.

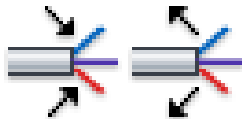
Notice how the segment has changed.



- 3 Right-click again, and click the Display as Actual Diameter check box to return the display to the actual diameter.

Change Fan State and Displays

You can change the fan state of wire stubs and cable wire stubs using Fan In and Fan Out on the Nailboard tab, Edit panel, or right-click to use the context menu.

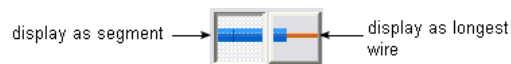


Fanned out wire stubs are always displayed using the current wire display setting. Once the nailboard is created, you can change the sorting direction and angle using Fan Out on the Nailboard tab, or right-click and use the context menu.



There are two options for displaying wire and cable wire stubs that are fanned in:

- Display as segment - use the diameter and color of the associated segment and the length of the longest wire.
- Display as longest wire - use the diameter, color, and length of the longest wire.



In this exercise, you create a nailboard view of the harness assembly from an open drawing and change the fan state and display behavior.

Fan in wires and change display settings



1 Click  ► New.

2 In the New File template dialog box, select the default template, *Standard.idw*, and then click OK.



3  On the ribbon, click Place Views tab ► Create panel ► Nailboard.

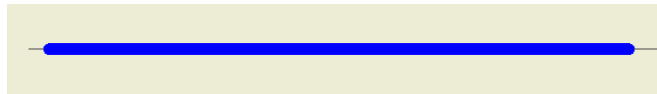
4 On the Nailboard View dialog box, ensure that the Nailboard View is *Harness Assembly 2.iam*.

5 Click the Display tab and click Fan In.

6 Under Appearance, click Display as Longest Wire.

7 Click OK and close the dialog box.

8 Right-click and select Finish Sketch for a better view of the cable wire stubs.

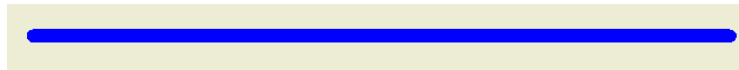


9 On the ribbon, click Nailboard tab ► Edit panel ► Edit.

10 In the browser, right-click *Harness Assembly2*, and select Nailboard Settings.

11 On the Nailboard Settings dialog box, click the Display tab, and then click Display as Segment.

12 Click OK.



NOTE This technique is the only way to change the fan in the display once the nailboard drawing is created.

Annotate Nailboard Drawings

Add annotations to the nailboard drawing to provide the needed manufacturing information. You can add dimensions and selected properties on the designated harness objects. You can also add tables including information such as a wire run list, or harness bill of materials.

Dimension the Nailboard

Nailboard dimensions are like driven dimensions where the values update when the harness segments and wire lengths update, but they do not affect the harness geometry. Unlike standard Autodesk Inventor dimensions, the nailboard dimension values show an adjusted length. The adjusted length is the actual calculated wire and segment length that factors in such things as service loop, global slack, and embedded length.

NOTE Search for and select “adjusted length” in the Help index to learn more about adjusting the length in a harness assembly.

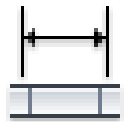
You can place dimensions between two selected points on segments, wire or cable stubs, or splices. The dimensions can span wires and segments if needed. Dimensions can be moved and deleted in the nailboard sketch, but they cannot be edited.

Dimension Styles control the text style, format, and display properties of the dimensions. All nailboard dimensions are aligned and parallel to the selected geometry.

When you exit the nailboard sketch, the parentheses are removed from the dimension and automatically retrieved into the view.

Add some dimensions to the harness segments

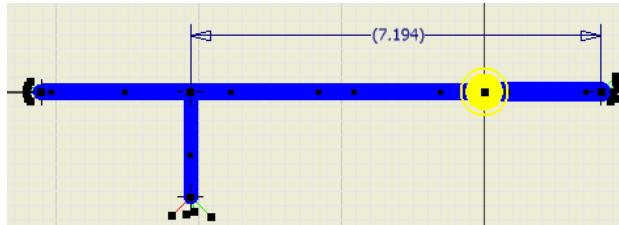
- 1 Activate the nailboard for Harness Assembly1.



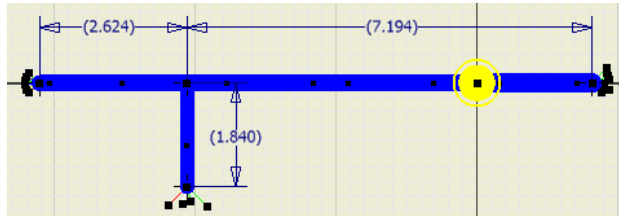
- 2 On the ribbon, click Nailboard tab ► Dimension panel ► Harness Dimension.
- 3 In the graphics window, click the point at the right end of the horizontal segment, and then click the segment point at the branch.

- 4 Drag to display the dimension, and then click to place as shown.

NOTE Your dimensions may differ depending on your placement of the segment and segment points.



- 5 Dimension the other two segments as shown, right-click and select Done.



Add Properties

Use the Property Display tool to view and display harness properties for wires, cable wires, pins, splices, and segments displayed on the drawing. You can select individual or multiple objects for display.

To place the properties in the drawing, you select the location for the first item in the selection set. The system automatically places all other properties offset from the associated harness object based on this initial selection.

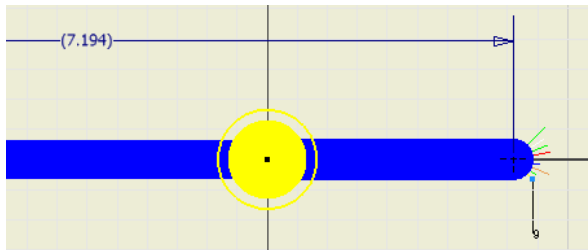
Once placed you can delete them, or move and adjust the displayed properties to their desired position by right-clicking and rotating 90 degrees in the clockwise or counterclockwise directions. You can also edit the text to change the font. If the harness object to which a property is associated is moved, the property is also moved.

Add properties to pins, parts, and a selected wire

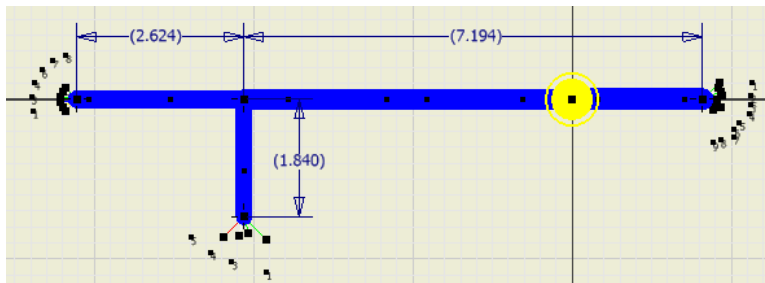
- 1 In the graphics window, zoom in on the harness.



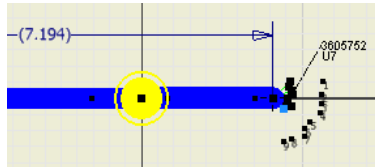
- 2 On the ribbon, click Nailboard tab ► Manage panel ► Property Display.
- 3 On the Property Display dialog box under Select Filters, click All Pins.
- 4 In the Property Name list, select Pin Name.
- 5 Under Display Options, select Value Only, and then click Apply.
- 6 In the graphics window a rubberband line appears from the property text box, which is attaches the cursor to the associated object. Click to place the property as shown.



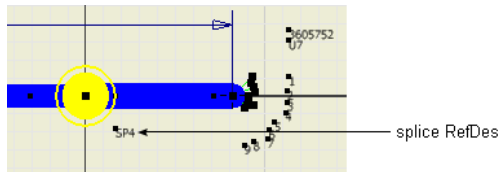
All properties are automatically placed based on this selection.



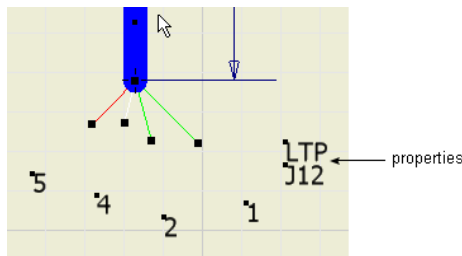
- 7 To place the part properties, select All Parts, select RefDes and Part Number, select Value Only, and then click Apply.
Use CTRL + click to select multiple items in the list.
- 8 In the graphics window, click to place the part properties.



- To place the splice properties, select All Splices, select RefDes and Value Only, and then click OK.

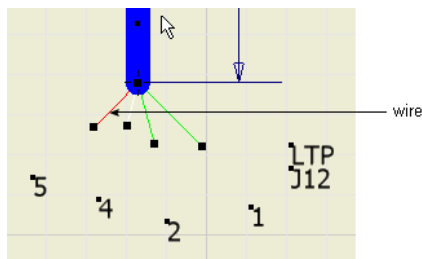


- Use CTRL + Click to select the LTP, J12 properties and drag them closer to the end of the branched segment.

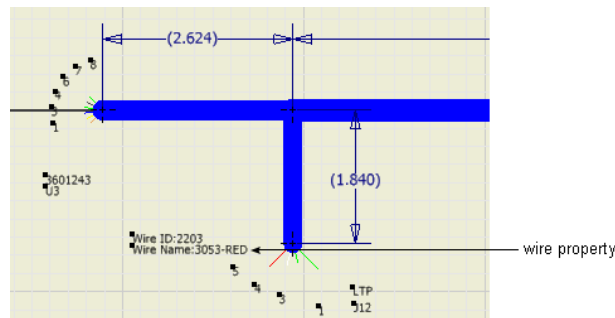


Place a property for a single object

- On the ribbon, click Nailboard tab ► Manage panel ► Property Display.
- 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 Apply.
In the graphics window, notice that 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.

Add Tables

To add a table with additional manufacturing information you first generate a report or a standard Autodesk Inventor bill of materials or parts list with the required information.

If a report is not generated before entering the nailboard environment, use the Report tool to generate it. For details on generating a parts list, search for “bill of materials, in nailboard” in the Help index. Once the report is generated, you use the Table tool to select the report output file and place the table in the drawing. You place parts lists and tables in nailboard just as you place parts lists and tables in other drawings.

Once the table is placed, you can move or resize the table by dragging the edges. You can also edit the table to change:

- Cell justification and size
- Cell and grid line weights and color
- Title position
- Text style

NOTE You cannot edit report data within the table. The external file controls the data so you cannot insert, delete, or change the order of rows or columns.

Typically, tables include wire run list, bill of material, or connector information. In this exercise, you place a cable and harness bill of materials that was created using the Report Generator in the nailboard drawing. The cable and harness bill of materials is unique to the harness assembly and may not contain standard Inventor parts list information. It does not include virtual part or ribbon cable information.

If changes are made to the external report file while the drawing is open, right-click the table in the browser, and then select Update to update the table data. The location and filename must be the same for the update to work.

Add a table to the nailboard drawing

- 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 the Browse button.
- 4 On the Open dialog box, browse to and select the *Harness Assembly1_Part_WireCable_BOM1.csv* file, click Open and then OK. The default location is: *Program Files\Autodesk\Inventor <version>\Tutorial Files\Cable & Harness\Report Generator*.
- 5 Click in the upper left corner of the drawing to place the table.
- 6 To modify the physical characteristics of the table, select the table in the browser or graphics window, right-click, and then select Edit.

- 7 Click Table Layout, and then use the Table Layout dialog box to change the title of the table to Enclosure Harness Assembly BOM. If appropriate, make additional changes to the table, and then select OK.

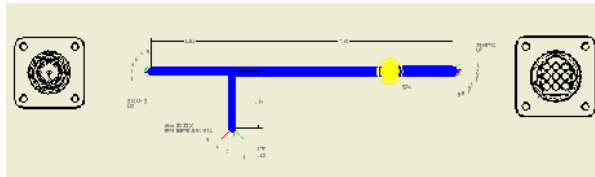
Placing Connector Views

You can also add connectors to complete the harness information using the Place Connector Views tool. You can set the default display characteristics before placing the connectors, or you can override the defaults on selected connectors while placing them.

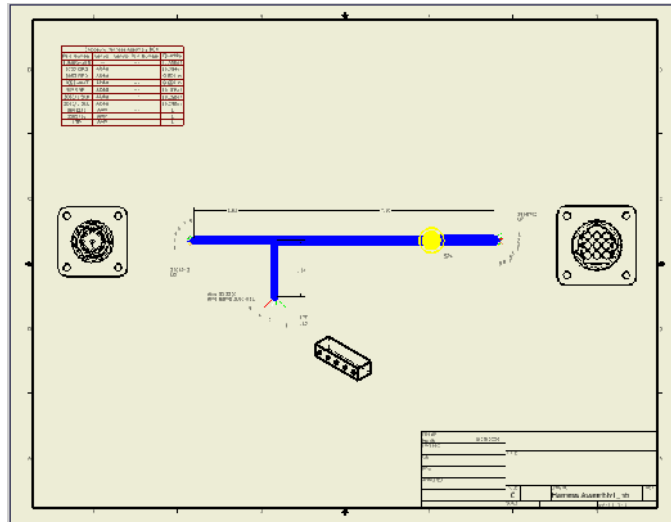
For this exercise, you place two connector views using default display settings and then override the default display settings for the other connector.



- 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 the Place Connector Views tool 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.

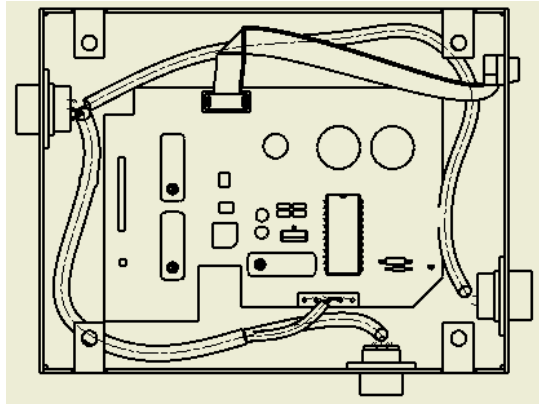
Create an Assembly Drawing

In a standard assembly drawing, cable and harness information is treated like other parts and subassemblies. You use normal drawing and manager methods and tools to detail it. You can sweep or recover harness objects into the drawing as centerlines.

1 Click  ► New.

2 On the Open template dialog box, select the default template, *Standard.idw*, and then click OK.

3  In the drawing view, click Place Views tab ► Create panel ► Base.



To display the portion of the splice that is hidden by other geometry, right-click the view in the graphics window, select Edit View, and then select the Hidden Line tool in the Drawing View dialog box.

- 8 Optionally, add dimensions and other annotations.
- 9 Save your work.

Print Nailboards and Drawings

You print nailboards and standard drawings that include harness assemblies using Autodesk Inventor printing capabilities. When printing nailboards and other large drawings, use the tiling capability to:

- Print a single drawing across multiple pages.
- Print registration marks on page corners for alignment of printed pages.
- Include page identifiers that contain the drawing and sheet name and a table cell number to help keep pages in order.

You can print or plot a single sheet, a range of sheets, or all of the sheets in the active drawing.

Workflow: Print a nailboard or drawing

- 1 Click  Print.

- 2 In the Print Drawing dialog box, set the print range, scale, and number of copies. If necessary, you can click Properties to open the Print Setup dialog box, and then change the paper size and orientation.
- 3 If the drawing is too large to fit on one sheet, in the Scale box, select Model 1:1, and then select the Tiling Enabled check box.
- 4 Click the Preview button to confirm your expected prints.
- 5 Click OK.

NOTE Check the All colors as black option to print the drawing in black and white. Embedded images and shaded views are still printed in color.

Refer to Help for details on other printing options.

Practice Your Skills

Create a nailboard drawing of the ribbon cable.

- Add dimensions for: conductor one, length of the ribbon cable, fold angle, internal work points.
- Place connector base views.
- Edit the ribbon cable harness assembly and change the fold to a double fold. Return to the nailboard and see the results of the change.
- Add properties.

Create a nailboard view from an open drawing, and then:

- Generate reports or a parts list and place additional tables.
- Update the external report file, and then update the table data.
- Move and adjust the table settings.
- Display additional properties.
- Create and place a parts list with roll-up.
- Experiment with angle and sorting direction for fanned out wire stubs and cable wire stubs.
- Change the orientation and font size of existing properties.

IDF Translator

Part 3 of this manual provides information about the Intermediate Data Format (IDF) translator for the exchange of printed circuit assembly (PCA) data between mechanical design and printed circuit board (PCB) layout in Autodesk Inventor® Routed Systems.

Use the IDF Translator

20

The Intermediate Data Format (IDF) is an industry-standard specification designed specifically for the exchange of printed circuit assembly (PCA) data between mechanical design and printed circuit board (PCB) layout.

In this chapter, you learn about the IDF, what types of IDF data are supported by the translator, how to import IDF data using the translator, and how the data can be used.

About the IDF Translator

The Autodesk Inventor® IDF Translator uses the IDF format to read files containing printed circuit assembly data and create an assembly or part file in Autodesk Inventor® containing the PCB and all of its components.

The IDF Translator enables the transfer of PCA data between PCB layout and mechanical design within Autodesk Inventor.

When you first open an IDF board file using the translator, general information about the file being opened is displayed. The items on the board are summarized so you can import all items or a subset that you select. You can import the data as a new Autodesk Inventor assembly or part file, or place the IDF data as a component in an existing assembly.

Once the data is imported, the board appears with basic board and component geometry information, along with design constraint information, such as keep in and keep out regions. Assigned component colors provide a visually correct representation of the PCB. The browser and BOM are updated with the IDF data.

NOTE The IDF does not provide a full design representation of a PCA, such as full functional or electrical descriptions, and should not be used as such.

With the Autodesk Inventor IDF Translator you can:

- Import industry standard files (*.brd*, *.emn*, *.bdf*, *.idb*) in the Intermediate Data Format (IDF) as new part or assembly files, or as subassemblies in existing assemblies.
- Display a summary of board items before importing.
- Automatically place all or selected board items, such as drilled holes, keepouts, outlines (including cutouts), and components on the PCB.
- Automatically group outlines, keepouts, and components based on board side (top, bottom, inner), when imported as an assembly.

NOTE Via keepouts are not grouped since they travel through the board.

- Assigns colors on import for an accurate board representation. You can change the colors for the entire board, other outlines, and all or selected components.

Exchange Data

The Autodesk Inventor IDF Translator supports IDF versions 2.0 and 3.0, and PCB design files with the extensions *.brd*, *.emn*, *.bdf* and *.idb*.

The following table summarizes and describes the types of IDF entities supported by the Autodesk Inventor IDF Translator.

Board entity type	Description
Outlines: <ul style="list-style-type: none">■ Board and Panel■ Other■ Routing■ Placement	Each of the four outline types provides a different function, such as communicating board shape and cutouts, or regions for routing and placing components. When imported as an assembly, outlines are represented in the browser as parts with single or multiple sketches. If multiple sketches exist, you can expand the part to view them. When imported as a part, outlines are represented as sketches.
Keepouts: <ul style="list-style-type: none">■ Routing■ Via■ Placement	Each keepout specifies a region of the board where a type of item is not allowed. If multiple placement keepouts exist, a sketch is created for each one. All routing keepouts on the same side of the board (Top, Bottom, or Inner) are stored together in a single sketch. Only one sketch is created for via keepouts, since only through vias are supported. When

Board entity type	Description
	imported as an assembly, all sketches for each type of keep-out are stored in a part. Expand the part to view the sketches.
Group Areas	Specifies an area where a group of related components are to be placed. If a group area exists on both sides of the board, a sketch is created for each side. When imported as an assembly, all sketches are stored in a part. Expand the part to view the sketches.
Drilled Holes	Drilled holes are distinguished from circular cutouts and are placed onto the same sketch as the board outline.
Components (Component Placement)	Board components such as connectors, switches, and displays are listed with their package name and part number. Each component contains placement information that specifies its location on the board. The display name for component occurrences is the placement reference designator. When imported as a part, each instance of a component is represented as a sketch. When imported as an assembly, each component is represented as a part, and there are multiple occurrences of that part instance for each component.
Annotations	Notes added in the IDF data are created and stored with the file, but are not displayed.

The entities created by the IDF Translator are not only graphical representations of design entities. They are normal Autodesk Inventor parts with additional attributes that are applied to the created sketch. Attributes for reference designators defined in the IDF file are stored with the component representation.

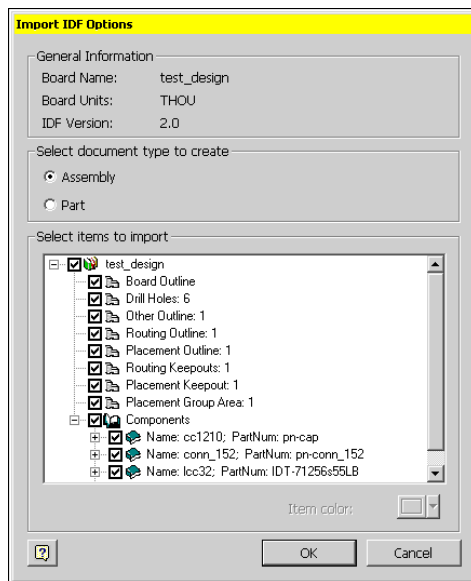
All part files and sketches are named automatically, based on the board name and other existing board information.

Understand Import IDF Options

Once you select a file to import, the IDF data is read into Autodesk Inventor and the Import IDF Options dialog box is displayed. It provides basic

information about the board being imported, options for how to create the data, and a summary of all board items.

You can click Cancel to exit the dialog box without importing any data. Click OK to import all checked items.



General Information

Displays the board name, the units used, and the IDF Version for the selected board data.

Select document type to create

Sets the type of document (or file) type to create.

- | | |
|-----------------|---|
| Assembly | Select to create a new assembly file containing multiple part files that represent the imported data. |
| Part | Select to create a single part file for the imported data, which reduces the number of files to manage. |

Select items to import

Lists the items contained on the board. Use this list to determine whether to import all items or a selected set. To prevent an item from being imported, clear the check box before that item.

Item color

Shows the color for a selected item in the list so you can assign new colors. You can assign a color to the entire board, other outlines, and all or individual components.

To select a new color, click the item name in the list, and then select a color from the palette.

Import IDF Board Files


In the typical design process, a PCB designer lays out the components on a board, and then the completed board assembly is passed through the IDF to the mechanical designer to verify fit (size, shape, height) in the context of an existing assembly.

Another common workflow involves the design of PCB boards to be used as standard components in other assemblies, or as stand-alone parts or assemblies.

To accommodate these common workflows, the IDF Translator provides two ways to import IDF data. You can:

- Import the data as a new part or assembly file.
- Place the data as a component in an existing assembly in Autodesk Inventor.


Import IDF data into an existing assembly

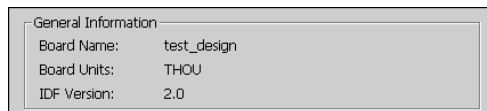
- 1 Open a destination Autodesk Inventor assembly file of your choice.
- 2  On the ribbon, click Assemble tab ► Component panel ► Place.
- 3 On the Open dialog box, in Files of type, select IDF Board File (*.brd*, *.emn*, *.bdf*, *.idb*), and then select the file to import.
- 4 Click Open to begin reading IDF data.

- 5 On the Import IDF Options dialog box, view the summary data, and then select part or assembly as the type of file to create.
- 6 Select the items to import.
All items are imported by default. Clear the check mark from the items you do not want to import.
- 7 To import other outlines, components, or reference designators in a different color, click the component name, and then select a color from the palette
- 8 Click OK.
To exit the dialog box without importing any data, click Cancel.
The assembly or part document for the imported IDF data is displayed, with the IDF board components. The browser and BOM are populated with the IDF data.
- 9 Return to the destination assembly to place the component as usual.

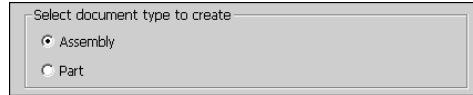
In this exercise, you import a board as a new assembly.

Import an IDF board file as an assembly

- 1 Click  ► Open.
- 2 On the Open dialog box, change the file type to IDF Board File (.brd, .emn, .bdf, .idb).
- 3 Navigate to the *Autodesk\Inventor <version>\Samples\Models\Translation\IDFTranslator* folder, and then select *idf.brd*.
- 4 Click Open to start reading data.
The status bar is displayed indicating the progress as files are being read.
- 5 On the Import IDF Options dialog box, review the general information displayed about the data being imported.
The information is displayed as shown in the following illustration.

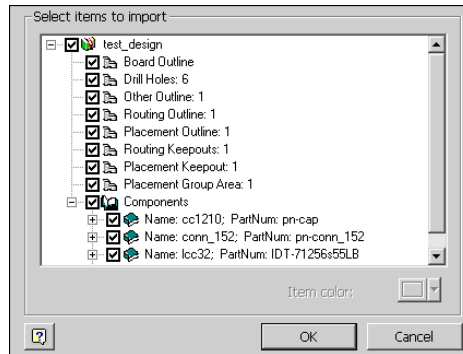


- 6 Select Assembly as the type of document you want to create.



- 7 Review the summary of board items to import to get an idea of the items included on the board.

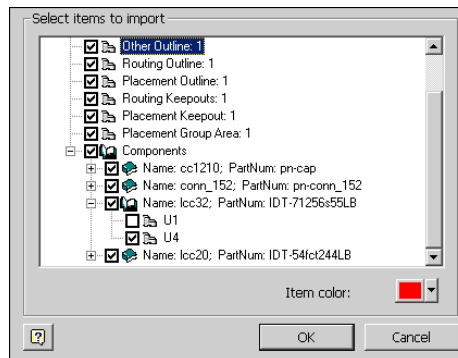
The summary of items is displayed, as shown in the following illustration:



- 8 Click the plus (+) sign before the component named Icc32;Partnumber:IDT-71256s55LB to expand it.

- 9 Clear the check box before U1 so that it is not imported.

- 10 Select Other Outline so the name is highlighted, and then verify that the assigned item color is red.



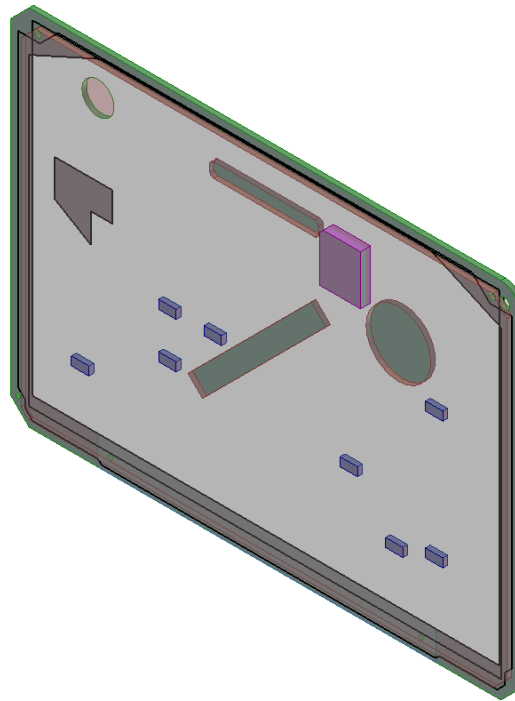
If multiple other outlines exist, all are displayed in the same color. Colors can be assigned to group components visually or to call attention to a

particular item. To assign a new color, click the arrow beside Item Color and select from the palette.

11 Click OK.

To exit the dialog box without importing any data, click Cancel.

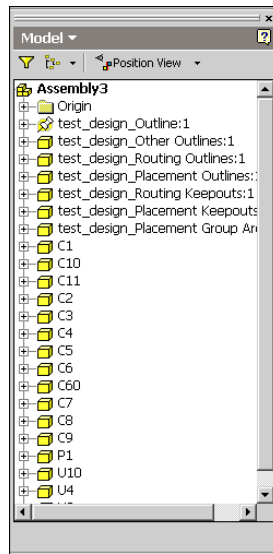
When the selected data is imported, a standard *.iam* file with the IDF board components is created, as shown in the following illustration.



The browser and BOM are updated with the IDF information.

12 Look at the browser to see the results of the import.

When imported as an assembly document, the board components are translated into multiple parts that are contained in the new assembly. Each part file contains the data for one or more component instance. The following illustration shows the results of importing the *idf.brd* file in the browser.



When imported as a part document, the board components are translated into the sketches and extrusions that make up the part model in a hierarchy. Importing IDF data as a single part simplifies file management and reduces the number of files that must be opened to display an accurate representation of the PCB assembly.



- 13 Click  ► Save, and accept the defaults.

For additional practice, try importing the same file as a part to see the differences in the browser information.

Use IDF Board Data

Once IDF data is imported, you can treat it as you would treat normal Autodesk Inventor parts, assemblies, and subassemblies. With the PCB data you can:

- Verify important features of the board including mounting locations, major components, connector locations, and keepout areas.
- Verify the fit and shape of the board in the context of the assembly, when placed in an existing assembly.
- Edit design constraints (keepouts) as 2D sketches.

- Create drawings, presentations, and design views.
- Manually define reference designators on individual components and pins.
- Use 3D measure between board items.

NOTE If the Autodesk Inventor IDF Translator is not installed on the system viewing the imported data, new IDF data cannot be imported. There are no other restrictions on interaction with the IDF data.

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