SolidWorks® 2005

SolidWorks Routing

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Lesson 1
Explaining Routes

Upon successful completion of this lesson, you will:

- Understand the basics of routing.
- Understand the different types of routes.
- Understand the various routing components and their roles.

Introduction

SolidWorks Routing is an advanced course, requiring a basic knowledge of various SolidWorks operations, including top-down design, general part design, use of configuration and design tables, drawing creation, etc.

This manual assumes this level of SolidWorks skills. If you are new to SolidWorks it is recommended that you refer to the *SolidWorks Essentials: Parts and Assemblies* and *SolidWorks Essentials: Drawings* training manuals. Please contact your reseller regarding these SolidWorks training courses.

Recommended Settings

It is recommended that you begin with the following options settings:

Tools, Options...

- **System Options, Colors**
  Check **Use specified colors when editing parts in assemblies**
- **System Options, Display/Selection**
  Assembly transparency for in context edit set to **Opaque assembly.**
- **System Options, File Locations**
  Under **Design Library**, add the location `c:\solidworks 2005 training files\routing\training design library\routing\electrical\harnessing`, and use **Move Up** to put it at the top of the list.

  (NOTE: This is the default location for unpacking of the training files; adjust path if necessary.)
- **System Options, Routing**
  Set **Library folder** to `c:\solidworks 2005 training files\routing\training design library`

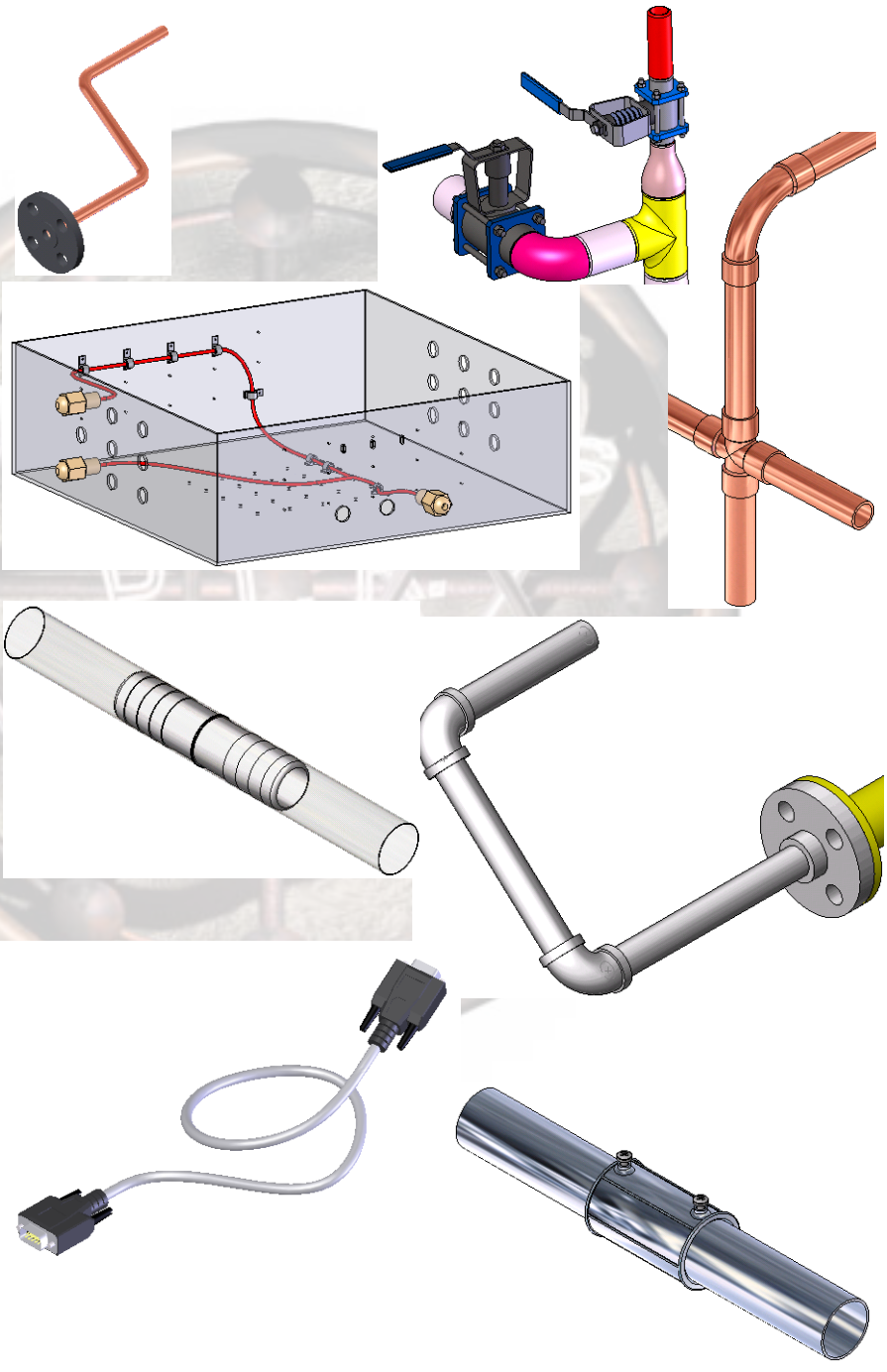
  (NOTE: This is the default location for unpacking of the training files; adjust path if necessary.)

  Under **Component and route defaults**, check **Automatically route on drop of clips**
- **System Options, Large Assembly Mode**
  Under **Display**, uncheck **Hide all planes, sketches, curves,** ...
  Check **Preview when inserting new components**

Types of Routes

SolidWorks Routing enables the designer to create tube, pipe and electrical (cable and wire) routes. Routes fall into several categories; some examples are below:

- Soldered copper, PVC, flexible tubing, electrical wiring enclosure, welded piping, fabricated cable, and associated fittings.



File Naming in Routing

The default naming convention for routing components is intended to be compatible with PDMWorks® and other PDM systems. As always, users may name files according to their own preference or company standard.

The default format for route sub-assemblies is:

`RouteAssy#-`<upper level assembly name>`.sldasm`

and for tube, pipe and cable parts within these route sub-assemblies:

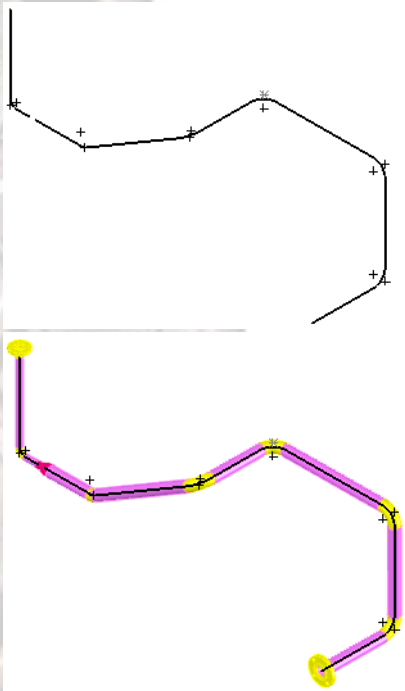
`Tube` or `Pipe` (and configuration) or `Cable-RouteAssy#-`<upper level assembly name>`.sldprt`.

Note

There may be references in this manual that do not conform to this new convention.

Routes

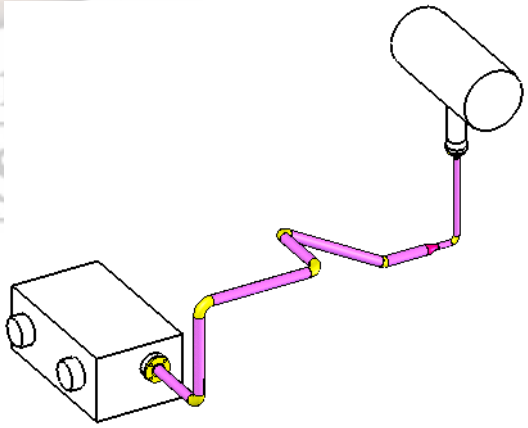
A **Piping** route is used in this description because it uses the most specialized components.



The **Route** is a component of a routing **Sub-assembly**, including a **3D Sketch**, that describes the centerline path of the piping, tubing or cable / harness route from a starting connection to an end connection. The properties of the route includes information to set the pipe, tube or cable nominal size, schedule or gage, and default elbow.

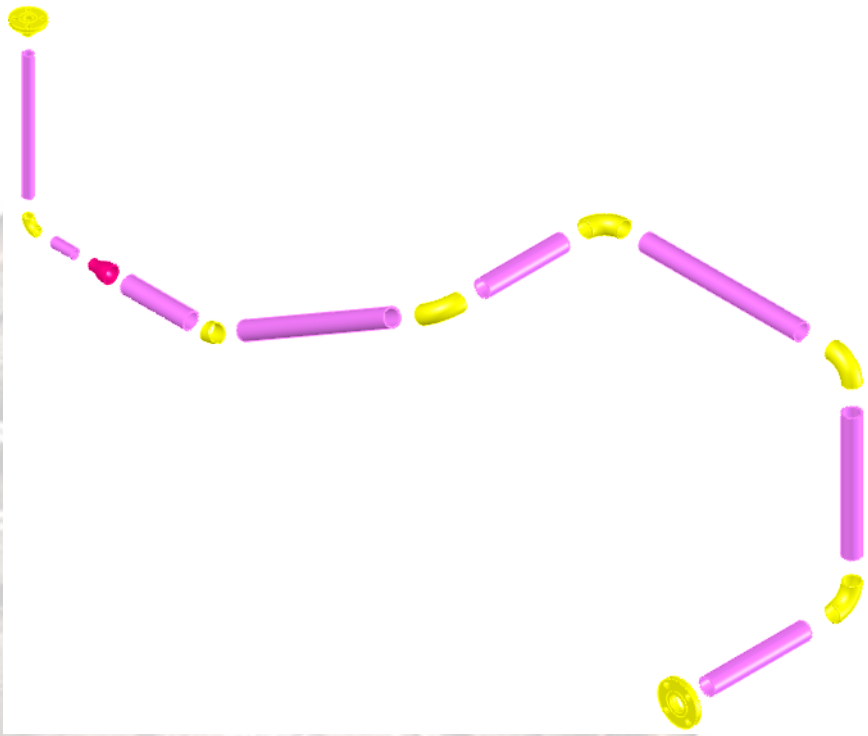
External Components

Routes are **Sub-assemblies** that can be connected to external components such as tanks, cylinders, manifolds or various electrical components. The sub-assembly component keeps the route components separate from the external components and other routes.



Route Components

Using the route properties and sketch geometry, pipes and elbows are added. Based on the elbow locations and geometry, the lengths of pipe between them are determined and added as individual components. In this example all the pipes are purple; elbow and flange fittings are yellow and reducer fittings are red.



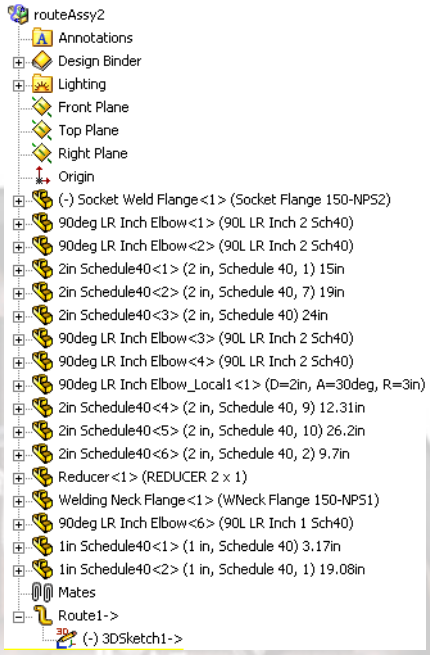
Anywhere the **Fabricated Pipe** changes direction is considered an elbow and is generally represented in the sketch as an arc. The straight lengths are represented as lines.

With rigid tubing, bends replace elbows.

Flexible tubing (hose) and cable typically use spline shapes; it is unusual for there to be truly straight runs of these components.

FeatureManager Listing

The FeatureManager for the **Routing Sub-assembly** lists the pipe and components used in the route. The component types used in this example are: pipes, flanges, elbows, reducers and custom elbows. One of each type is labelled graphically.

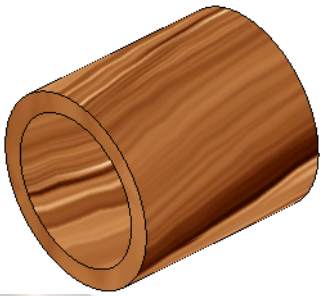
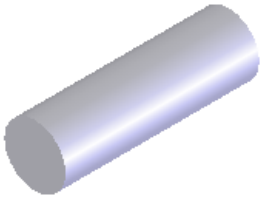
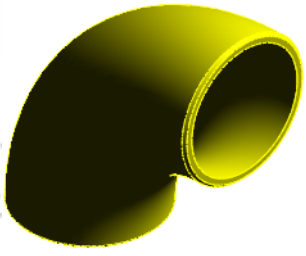


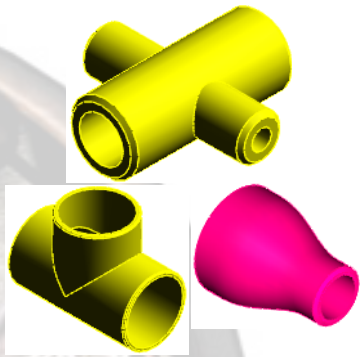
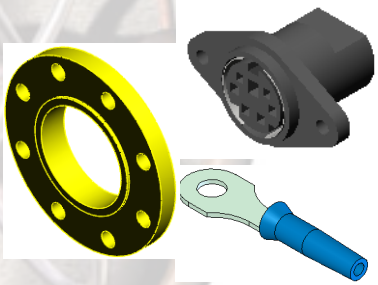
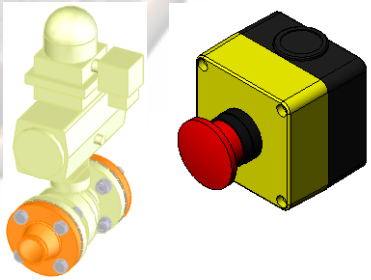
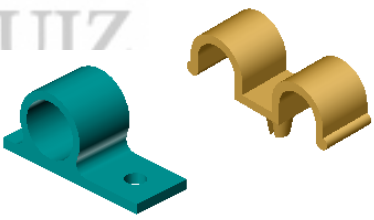
Note

The **Route Components** are attached to the route sketch directly and do not require mates to each other. Only the flange type, which connects to components outside the route, is mated.

Individual Components

Many of the common piping components, both parts and assemblies, are supplied with the SolidWorks **Design Library**. You can create your own custom components and libraries.

Tubes are parts that follow the length of the route, to the end of the sketch or to a fitting. The part includes all bends whether they are orthogonal or free form.	
Pipes, or more specifically **Fabricated Pipes**, are parts that are placed between elbows and fittings following the route. The FeatureManager listing includes the name, configuration and length. *Rigid* copper tubing would be considered fabricated pipe.	
Cables are parts that follow the length of the route, to the end of the sketch or to an electrical connector. Unlike Tubes and Pipes, there is no cable "seed" part; the cable is generated within the route, with specifications extracted from a default or user specified Microsoft Excel or XML file.	
Standard Elbows are part components that are placed at changes in direction along the route. They are placed automatically at 90 and 45 degree bends. The FeatureManager listing includes the name and configuration.	

Custom Elbows are used where the change in direction is less than 90 degrees but not 45. The system will prompt you to allow the modification of a standard elbow to match the angle. The FeatureManager listing includes name, configuration and sizing.	
Fittings is a general classification of *part* components that are *not* added to the route automatically like pipes and elbows. This includes tees, reducers and crosses.	
Flanges and **Connectors** are special part fittings that generally connect to both the route and equipment *outside* the route. Because of this, flanges and connectors generally contain **Mate References** for equipment connections.	
Assembly Fittings is a general classification of *assembly* components that are *not* added to the route automatically like pipes and elbows. These include valves, switches and other multiple-part route components.	
Clips are routing components for electrical or flexible tubing routes that help to locate the route as desired. Clips can be pre-placed and used as reference locations, or dropped into the route "on the fly" during route generation.	

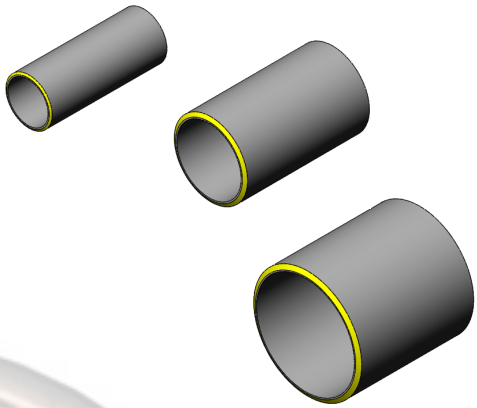
Lesson 2
Review of Configurations

Upon successful completion of this lesson, you will:

- Understand how Routing and Design Library parts use configurations.
- Understand the role of design tables.

Review of Configurations

Configurations, specifically those created by **Design Tables**, are an integral part of Routing. They are used to create and store library parts such as tubes, pipes and elbows. For example, one part can be used to represent multiple tubes, each having different diameters and wall thicknesses.

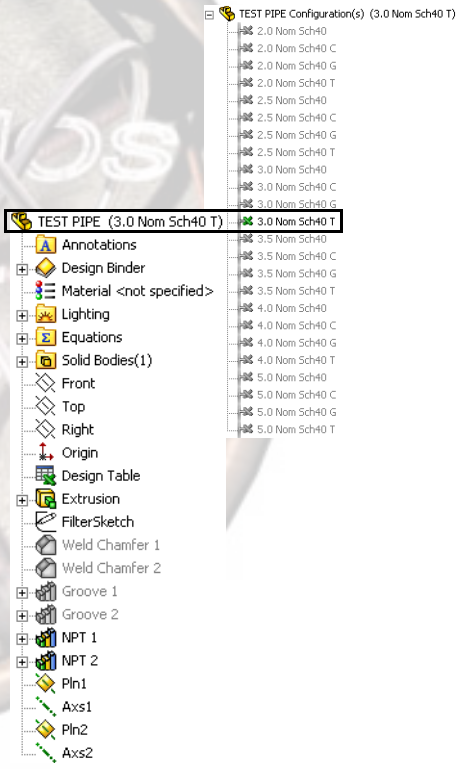


How Routing Uses Configurations

Routing uses configurations to select matching tubes or pipes and related components to size the route.

Tube/Pipe Components

When a tube/pipe route is started with a fitting such as a **Flange**, the configuration that represents a specific nominal diameter is selected. The pipe that connects to that fitting now has limited set of configurations to choose from based on that nominal diameter. The choices are limited to the type of pipe or the schedule in that diameter.



The tube/pipe part is copied to the local folder and configurations are used to represent the required lengths.

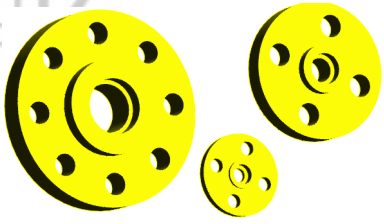
That tube/pipe part is used throughout the route. Added components must match the sizing that has been selected for the route.

Note

Electrical routes do not use configurations to determine route and component size.

Fittings

Fitting configurations are chosen by matching the configuration of the tube/pipe. If the fitting does not contain a matching configuration, an error is produced.



Unlike tubes and pipes, fitting parts are *not* copied to the local folder. They remain referenced to the Design Library.

A Note About File References

Referenced files do not have to be stored with the document that references them. In most practical applications, the referenced documents are stored in multiple locations on the computer or network. SolidWorks provides several tools to determine the references that exist and their location.

Find References

Find References provides the exact locations of referenced part and assembly files. **Find References** will display the Search Results dialog box which lists the component files used, including the full path names. This is useful if you have several versions of the component files.

Where to Find It

- From the **File** menu, select **Find References**.

Copy Files

The **Copy Files** button can be used to copy the files to another, common, directory. Find references can be used as a "Pack and Go" by copying the parent and all referenced files to a single location. This can be useful when creating a ZIP file with a drawing, assembly and all the parts.

File Management

As noted previously, pipe and tube parts are copied and saved in the working directory with only the required configurations. Fitting parts are not copied, so the assembly will reference the file in the Design Library directory.

If the entire assembly is to be copied, moved, archived or zipped, consider saving all the reference files (Design Library parts) with it. **Find References** will accomplish this.

It is also a good idea to include a copy of the original pipe or tube file from the Design Library with the assembly. In this way, should a new pipe or tube route of the same type be added to the assembly after it has been moved, the same seed part can be used.

The configured pipe or tube files that exist in the assembly do not have all the required information to be used in a *new* route and hence cannot be used this way. **Find References** will NOT copy this file, it must be done manually.

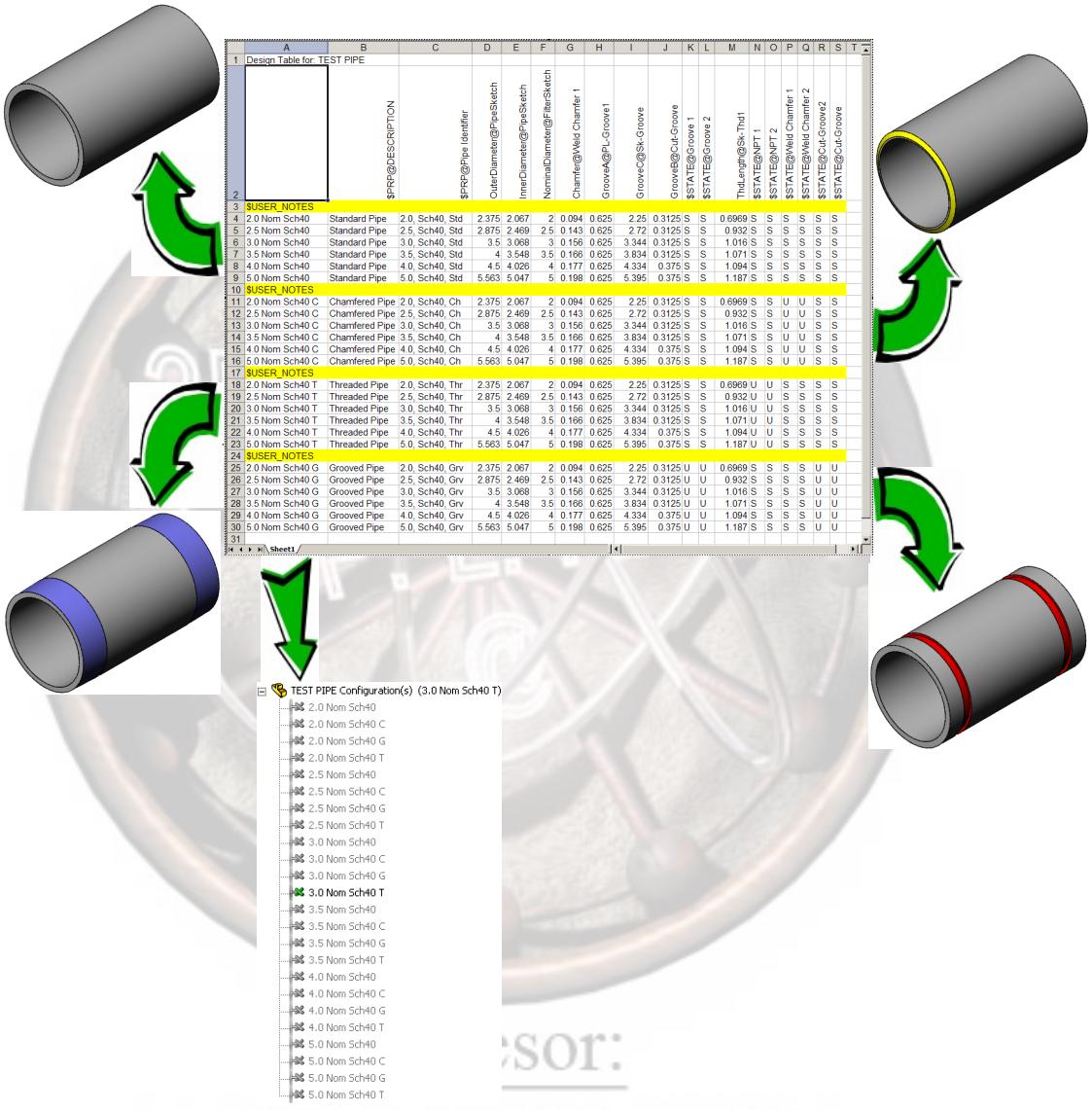
PDMWorks®

SolidWorks Office Professional includes PDMWorks®, which provides a powerful way to manage library parts and project files.

For more information, see the File Management training manual.

How Libraries Use Configurations

Library Parts are supplied by SolidWorks through the **Design Library**. These parts and assemblies, as required, use design tables as the engine to create configurations. Design tables can set the values for dimensions and suppress features by configuration name. The result of the design table is a set of configurations shown in the Configuration Manager.



Design Tables

Design tables are the most efficient method of creating and maintaining configurations. Using an Excel spreadsheet, the configurations and the variations between them are recorded and stored.

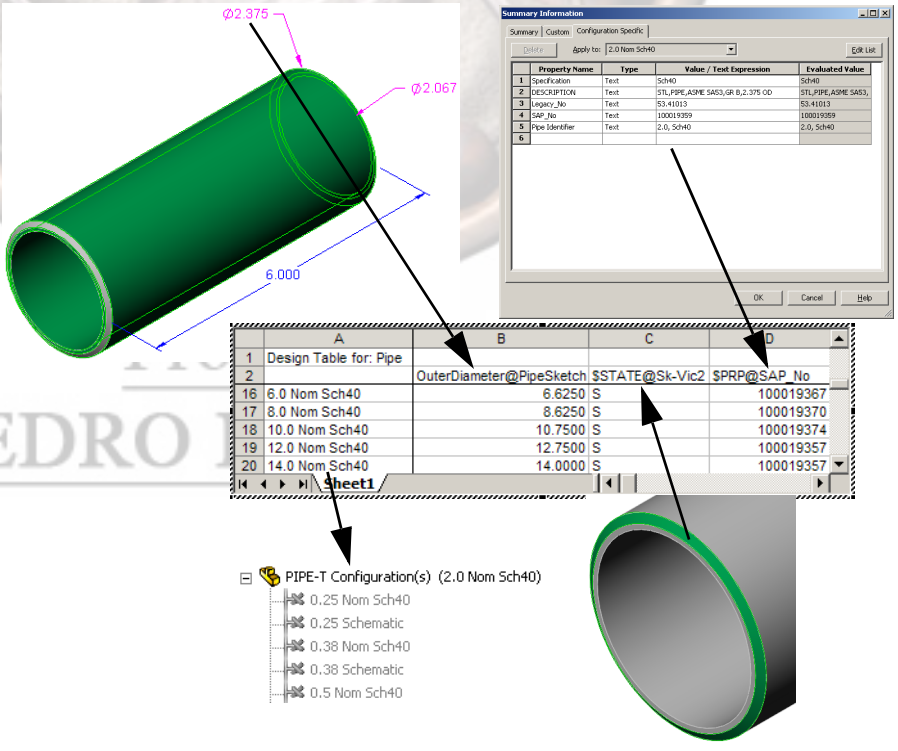
The **Design Table** contains columns for configurations, dimension values, suppression of features, configuration specific properties and notes. Using the **Auto-create** function allows the table to be generated automatically from the input.

Design Table Input and Output

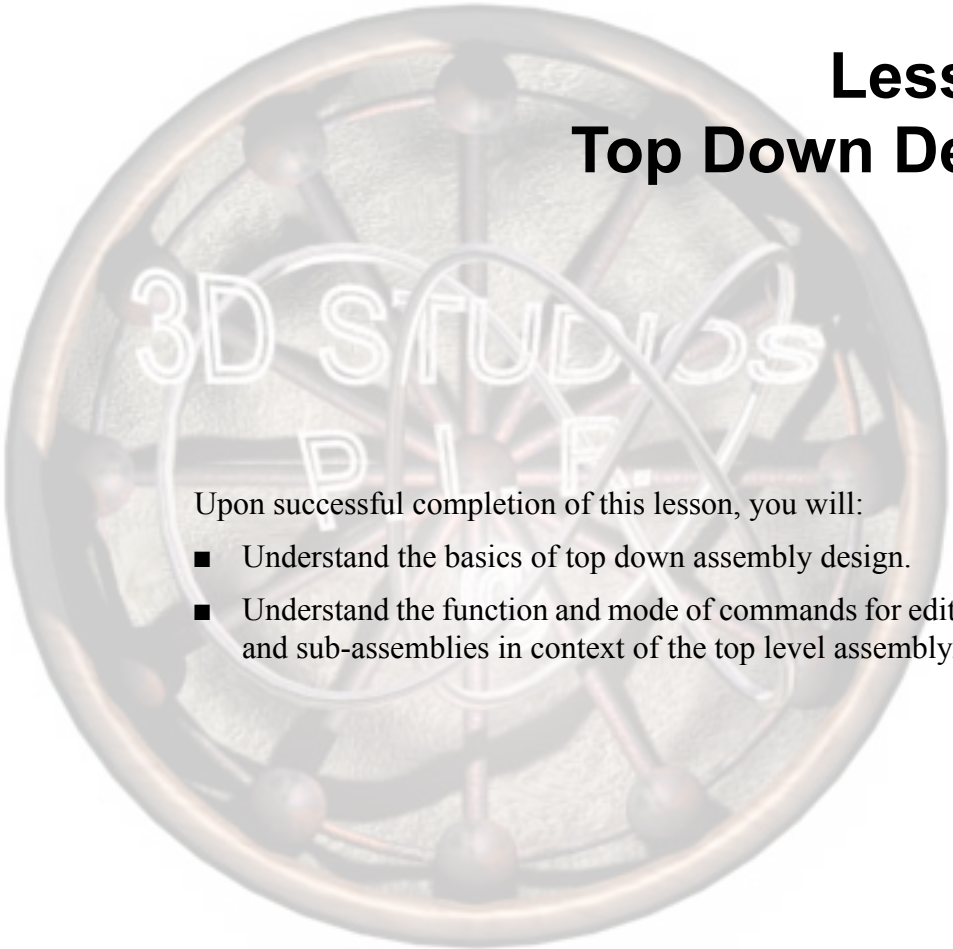
The input options include the parameters dimensions, features and properties. The output is a set of configurations that are added to the ConfigurationManager of the part. As always, descriptive names should be used for dimensions and features included in the design table.

If the design table settings allow it, changes can be bi-directional.

- **Dimension Values**
  A **Dimension Name** appended with `$<feature name>` as a column header controls the value of that dimension by configuration.
- **Feature Suppression**
  Prefixing a **Feature Name** with `$STATE@` allows that feature to be suppressed or unsuppressed by configuration. `Suppress`, `S`, `Unsuppress` or `U` appear in the cells below the header. A blank cell assumes unsuppress.
- **Configuration Specific Properties**
  Properties added to the part through **File, Properties Configuration Specific** tab appear in the design table with the prefix `$PRP`.
- **Notes**
  User defined notes can be added in rows or columns with `$USER_NOTES` as the row or column header.
- **Configuration Names**
  Configuration names added in the header row become configurations that appear in the ConfigurationManager.



Lesson 3
Top Down Design



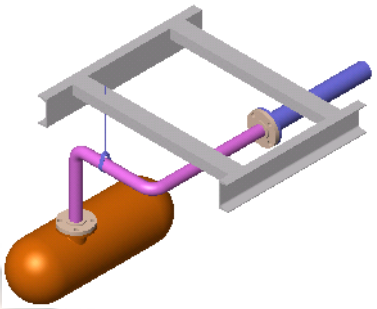
Upon successful completion of this lesson, you will:

- Understand the basics of top down assembly design.
- Understand the function and mode of commands for editing of parts and sub-assemblies in context of the top level assembly.

Top Down Introduction

Top Down Assembly modeling involves the switching between the editing of parts and assemblies.

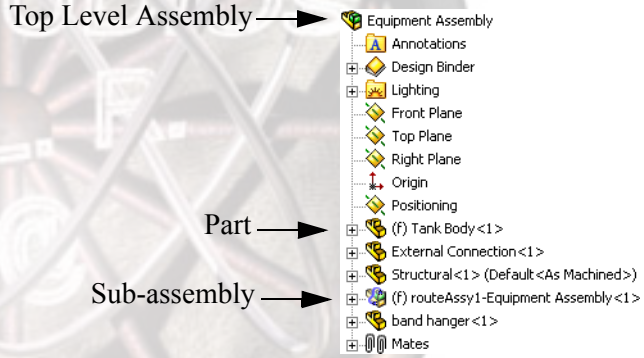
1 **Open assembly.**
Open the assembly `Equipment Assembly`.



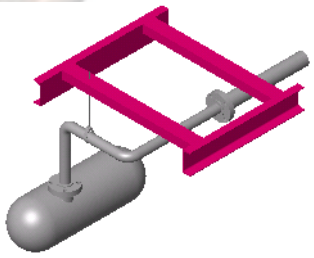
Parts and Assemblies

When using routing, several types of components are used. Each can be edited in turn to make effective use of top down design and complete the routing assembly. The types of components are labelled below.

Note that the sub-assembly in this case is a specific type, a *Routing* sub-assembly.



Edit Part



While you are in an assembly, you can switch between editing the assembly — adding mate relations, inserting components, etc. — and editing a specific part. Editing a part while in the context of an assembly allows you to take advantage of geometry and dimensions of other components while creating matching or related features. Using geometry outside the part creates **External References** and **In-context Features**.

Four commands, **Edit Part**, **Edit Assembly**, **Edit Sub-assembly** and **Edit Route** are used to switch back and forth between editing one component in an assembly and editing the assembly itself. When you are in edit part mode, you have access to all the commands and functionality the part modeling portion of SolidWorks. Plus, you have access to other geometry in the assembly.

Introducing: Edit Part, Edit Assembly and Edit Sub-assembly

Edit Part/Edit Assembly/Edit Sub-assembly is used to switch between editing a part and an assembly. The right-mouse menu will display the proper command.

Where to Find It

Select the part or sub-assembly you wish to edit. Then:

- Click **Edit, Part**, or **Edit, Assembly** or **Edit, Sub-assembly**.
- Or, from the right-mouse menu, select **Edit Part**, **Edit Assembly** or **Edit Sub-assembly**.
- Or, from the Assembly toolbar, click the  tool.

Note

The colors are set through **Tools, Options, System Options, Colors** as **Assembly, Edit Part** and **Assembly, Non-Edit Parts**. In order to see these colors, the option **Use specified colors when editing parts in assemblies** must be enabled. Otherwise, the edited part's color will not change. This option is unchecked by default.

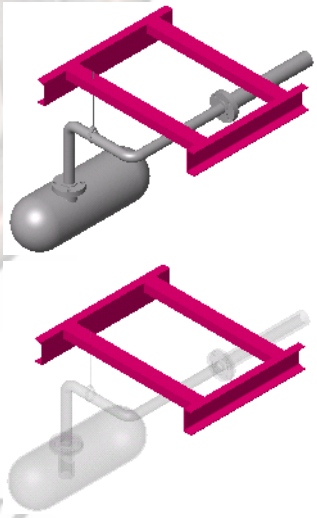
Appearance of Components While Editing

When you edit a part in the context of the assembly, if the option above is turned on, the component you selected turns opaque pink (or any user-specified color). The appearance of the other components depends on the assembly transparency settings you choose.

Introducing Change Assembly Transparency

The transparency of components that are *not* being edited can be set to one of three conditions:

- **Opaque assembly**. All components become opaque gray, except for the component you are editing, which becomes opaque pink (recommended, particularly for complex designs.)
- **Maintain assembly transparency**. All components maintain whatever their current transparency is, except for the one you are editing, which becomes opaque pink.
- **Force assembly transparency**. All components become transparent except the one you are editing, which becomes opaque pink.



Where to Find It

- From the menu click **Tools, Options, System Options, Display/ Selection, Assembly transparency for in context edit...**.
- Or, from the Assembly toolbar, click the  tool.

Note

Use the slider to adjust the transparency level for **Force assembly transparency**. When you move the slider to the right, the components become more transparent.

Editing Options

Each editing option is useful in its own way in the assembly context. By editing parts in context, you can take advantage of any entity in the assembly for sketching, offsetting, converting and dimensioning.

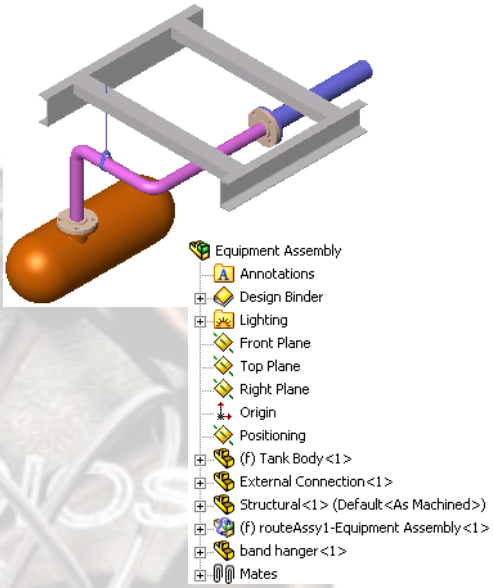
Edit Assembly

Edit Assembly is the default state of an assembly when it is created or opened.

Also, when **Edit Part** or **Edit Sub-assembly** is "turned off" the top level assembly returns to this state.

Use **Edit Assembly** to:

- Add or delete components from the top level assembly.
- Add or delete mates from the top level assembly.

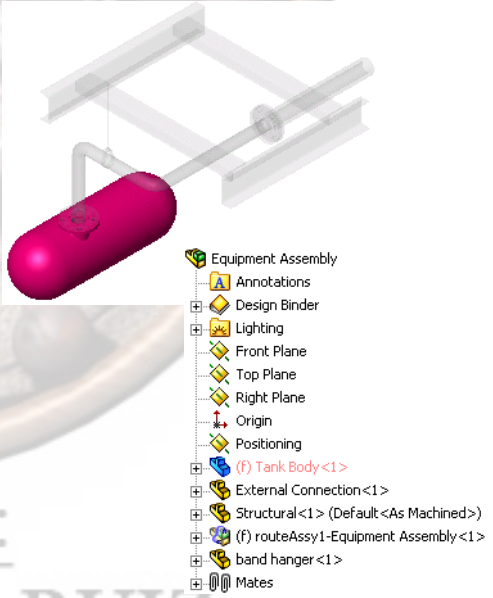


Edit Part

Edit Part is used to access a specific part and edit it while remaining in the top level assembly.

Use **Edit Part** to:

- Add or edit individual sketches of a part within the assembly.
- Add or edit features of a part within the assembly.

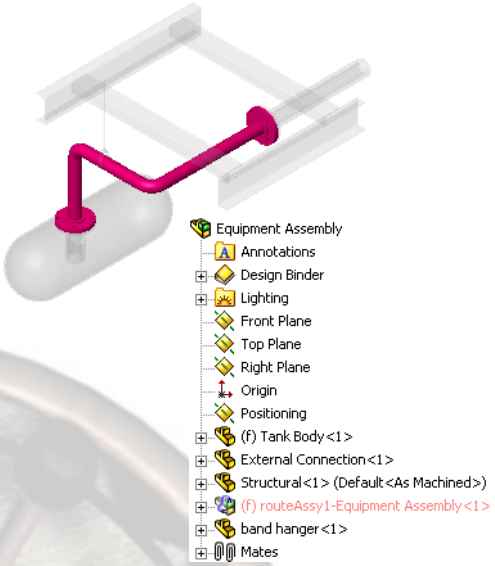


Edit Sub-assembly

Edit Sub-assembly is used to access a sub-assembly and edit it while remaining in the top level assembly. In routing it is often used to make changes to the route sub-assembly .

Use **Edit Sub-assembly** to:

- Add or delete components within the sub-assembly.
- Add or delete mates within the sub-assembly.

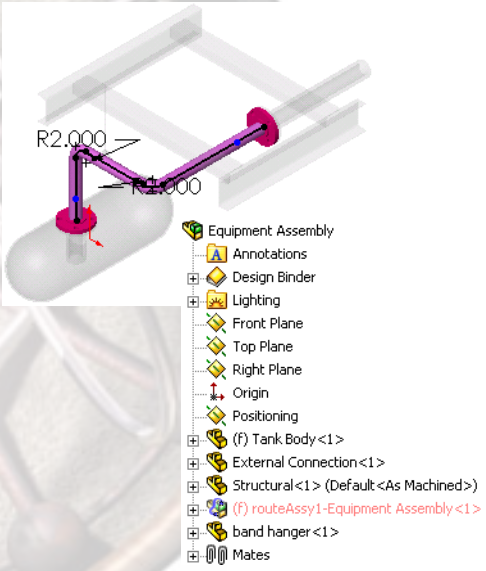


Edit Route

Edit Route is used to edit the **3D Sketch** that defines the tube or pipe route. This automatically switches to **Edit Sub-assembly** mode using the route sub-assembly.

Use **Edit Route** to:

- Create and edit the 3D Sketch used as the route.
- Add inline components such as reducers and tees.

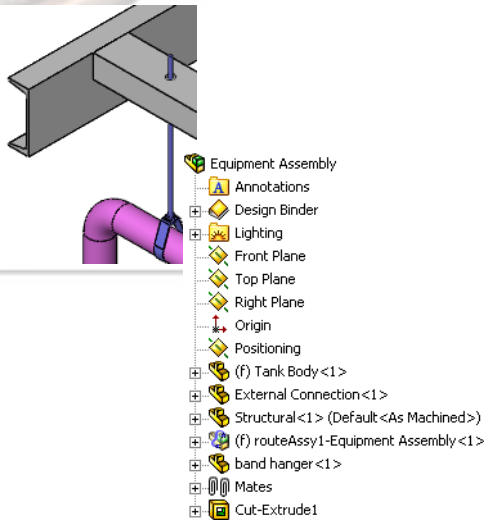


Assembly Feature

An **Assembly Feature** is used to add a cut feature or hole to the top level assembly. The feature uses a **Feature Scope** to determine which components are affected by the cut.

Use **Assembly Feature** to:

- Add a cut feature that exists *only* in the top level assembly.



Lesson 4
Design Library Introduction

Upon successful completion of this lesson, you will:

- Understand how Design Library parts are used in Routing.
- Understand the function of Routing Points in Design Library parts.
- Understand the Configuration Specific Properties required for Routing components.

Design Library Parts Introduction

Design Library Parts are standard components of various types that can be stored on the Part Design Library for insertion into an assembly quickly and easily. SolidWorks supplies some, but users may also make their own.

The Design Library parts supplied with the SolidWorks software are configured with established standard sizes. They include some standard fasteners and mechanical components, but in this case we will work with standard pipe, tube and electrical components.

There are both geometric and non-geometric (property) requirements in building design library parts.

Routing Points

There are specific geometric requirements for Design Library parts including the **Route Point** and **Connection Point** features.

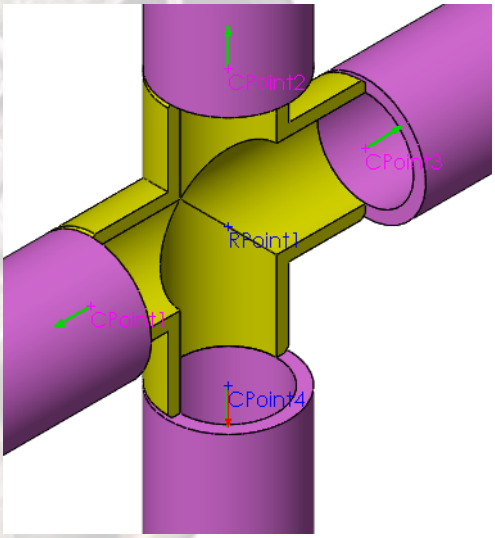
Note

Route Points (**RPoint**) and **Connection Points** (**CPoint**) are collectively referred to as **Routing Points**.

Connection Points

Connection Points are required in fittings and electrical connectors such as elbows, tees, crosses and crimp terminals. They are used to determine where the routing ends and the direction that it enters the fitting or connector.

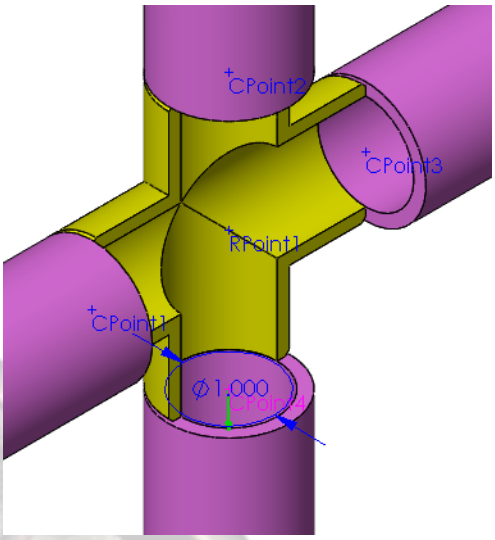
They are also used to specify the nominal diameter of the route and the type of route: **Electrical, Tubing** or **Fabricated Piping**.



Tip

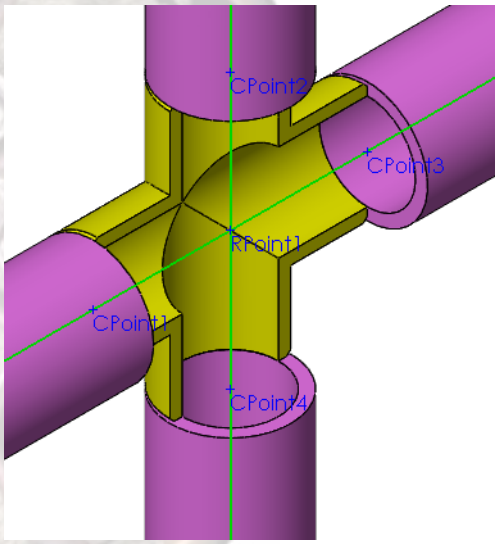
Double-clicking a CPoint displays the nominal diameter assigned to the CPoint.

In an **Electrical** route, the nominal diameter is defined as a maximum cable size for that component.



Route Points

Route Points are required in fittings such as elbows, tees, crosses and route clips. They are used to place the fitting on an endpoint on the 3D Sketch route line, or in the case of clips, to help define the path of of a flexible route.



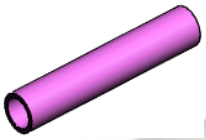
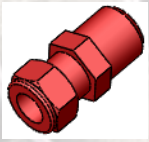
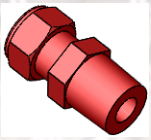
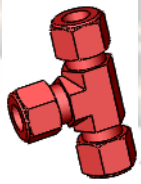
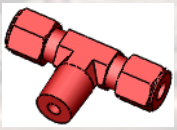
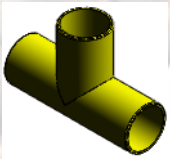
Tip

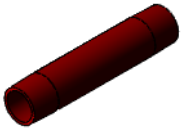
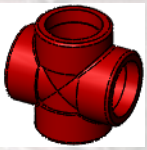
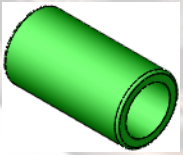
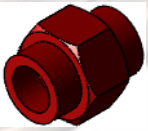
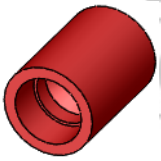
Route line components (tube, fabricated pipe, or electrical cable) do not have routing points in them.

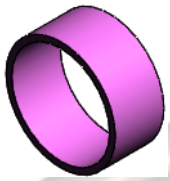
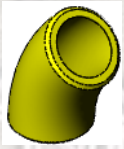
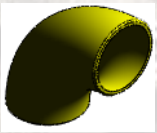
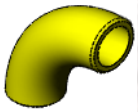
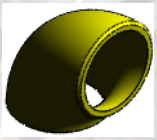
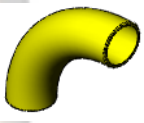
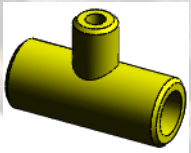
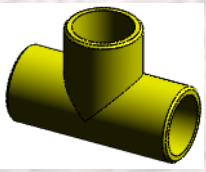
Libraries

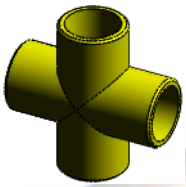
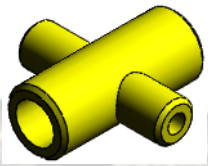
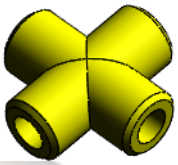
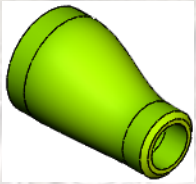
SolidWorks provides Design Library parts and assemblies that include common tube, wiring and pipe sizes and associated fittings.

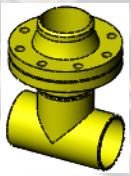
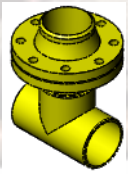
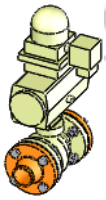
For more vendor-supplied routing components, visit 3DContentCentral℠, **www.3dcontentcentral.com**.

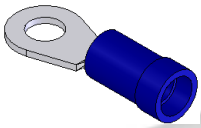
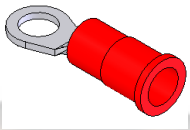
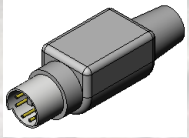
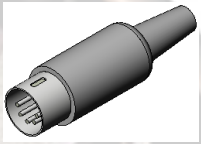
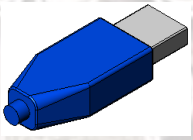
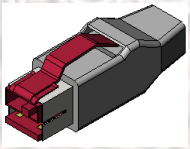
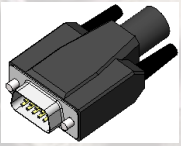
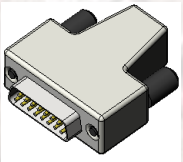
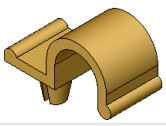
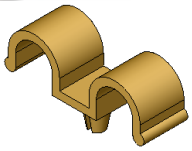
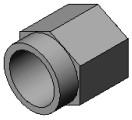
Tubing	C:\Program Files\SolidWorks2005\data\design library\routing\tubing			
Tubes	tube-ss 			
Flanges	slip on tube flange-ss 			
Tube Fittings (Connectors)	solidworks -lok male pipe weld connector 	solidworks -lok male connector 		
Tube Fittings (Tees)	solidworks -loktubing branch tee 	solidworks -lok male branch tee 	tee-ss 	

Threaded Piping	C:\Program Files\SolidWorks2005\data\design library\routing\piping\threaded fittings (npt)			
Pipes	threaded steel pipe 			
Elbows	threaded elbow--45deg 	threaded elbow--90deg 		
Tees, Laterals and Crosses	threaded tee 	threaded lateral 	threaded cross 	
Couplings and Unions	threaded coupling 	threaded half-coupling 	threaded union 	
Reducers and Cap	threaded reducer 	threaded cap 		

Piping	C:\Program Files\SolidWorks2005\data\design library\routing\piping			
Pipes	Pipe 			
45° Elbows	45° LR Inch 	45° LR Metric 		
90° Elbows	90° LR Inch 	90° LR Metric 	90° SR Inch 	90° 3R Inch 
Tees	Reducing Outlet Tee Inch 	Straight Tee Inch 		
Flanges	Socket Weld Flange 	Slip On Weld Flange 	Welding Neck Flange 	

Piping	**C:\Program Files\SolidWorks2005\data\design library\routing\piping**			
Crosses	Straight Cross Inch 	Reducing Outlet Cross Inch 	Straight Cross Metric 	
Reducers	Reducer 	Eccentric Reducer 		

Assemblies	**C:\Program Files\SolidWorks2005\data\design library\routing\assembly fittings**			
	assembly fitting 	assembly fitting without acp 		
Valves	2in control valve 			

Electrical	C:\Program Files\SolidWorks2005\data\design library\routing\electrical			
Ring Crimp Terminals	ring_term_awg-14-16_awg-x 8 	ring_term_18-22_awg-x_6 		
DIN Connectors	socket-6pinmindin 	plug-6pin-minidin 	plug-5pindin 	
USB Plugs	plug-usb1 	plug-usb2 		
DB Connectors	db9 male 	db15-e 		
Clips	pclip2 	richco_hurc-4-01-clip 	richco_dhurc-4-01-dualclip 	cable constraint (no visible geometry)
Other	plug-sma 			

"Pipe" Parts

The classification of "Pipe" parts includes tubes as well as true fabricated pipes and elbows.

Note

Unless otherwise specified, these properties apply to tubing and piping only, not to electrical components.

Design Table Parameters

All Design Library pipe parts require design tables to store the required information:

- `$PRP@Pipe Identifier`
- `InnerDiameter@PipeSketch` or `@FilterSketch`
- `OuterDiameter@PipeSketch` or `@FilterSketch`
- `NominalDiameter@FilterSketch`

Certain other components may use these parameters to size themselves accordingly when dropped into an assembly. These include pipe elbows and pipe and electrical hangers/clips such as `pclip2` (Nominal, Inner and Outer Diameters).

Electrical wire and cable use only the `OuterDiameter@PipeSketch` parameter. Data for wire and cable parts are stored in a .XML file; There is no Design Library part.

Configuration Name

The **Configuration Name** is used in the **Route Properties** dialog as the **Base Configuration**.

Configuration Specific Properties

The **Pipe Identifier** property is required for the component to be recognized as a tube or pipe. The property is entered as a **Configuration Specific Property** through the **File, Properties** dialog as `$PRP@Pipe Identifier`. The value of this property is used to name all instances of the tube or pipe in the route.

Other properties can also be added as descriptions or additions to a BOM table. Properties such as `Description`, `Weightperfoot`, `Wall Thickness` and `Swbompartno` are common additions.

Dimensions

The `InnerDiameter`, `OuterDiameter` dimensions are taken from the `PipeSketch` sketch of the `Extrude` or `Sweep` feature. The `NominalDiameter` is taken from the `FilterSketch` sketch.

Additional dimensions can be added for use in equations or descriptions.

Note

The length of each tube or pipe segment is extracted for use in the BOM table.

Lesson 5
Design Library Review

Upon successful completion of this lesson, you will:

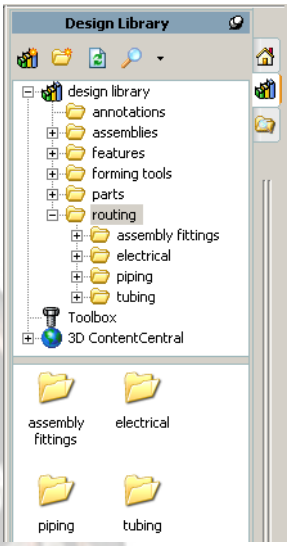
- Understand the function of the Design Library: Features, Parts and Assemblies.
- Be able to organize Design Library components.
- Understand the options available for Routing.

Design Library Task Pane

The **Design Library** pane is used to access and store commonly used library features, sheet metal forming tools and parts. **Design Library** parts can be added to an assembly by simply dragging and dropping them into that assembly. Mate Relations, SmartMates and Route Points are used to position the part.

Introducing: Design Library

The **Design Library** Task Pane tab is a menu set devoted to the storage and easy retrieval of library features, forming tools, assemblies, and piece parts. It is designed to resembly Windows Explorer in apprearance and function.



Navigation through the directory structure is by single (select and view)and double (open) click.

Reload refreshes the window with changes that have been made to the folder since it was opened.

Where to Find It

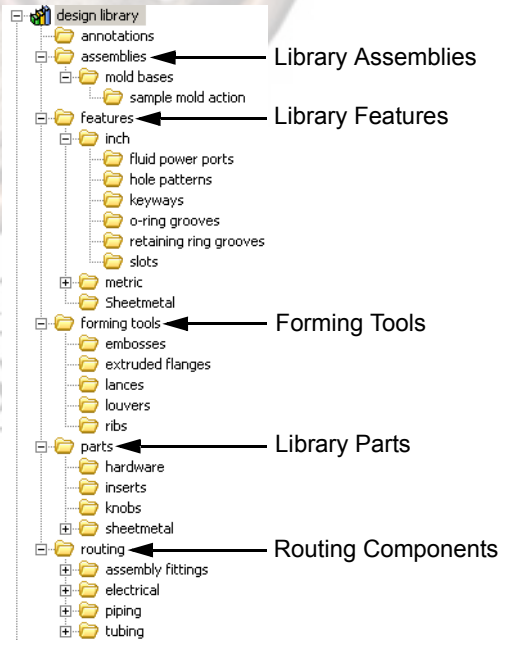
- On the Task Pane, select the **Design Library** .

Essentials of Using the Design Library Task Pane

Taking full advantage of the Design Library pane requires an understanding of the file structure it uses. Although some library features and parts are supplied with the SolidWorks software, the real power of the Design Library is in creating and using your own folders and libraries.

Directory Structure

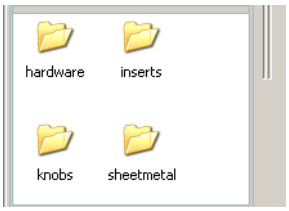
These files can be located on your hard drive using Windows Explorer by browsing to the `Solidworks` install folder and opening the `data\design library` folder. This folder contains five main folders for the Design Library pane: `assemblies`, `features`, `forming tools`, `parts`, and `routing`.



Note: The view in the Design Library pane is virtually identical to that in Windows Explorer.

Library Parts

Parts shipped with the Design Library are found in the subdirectories located under the folder `parts`. Each folder creates a corresponding folder in the Design Library pane. All of these must be `*.sldprt` files.



Library Assemblies

Assemblies shipped with the Design Library are found in the subdirectories located under the folder `assemblies`. These must be `*.sldasm` files.

Adding to the Folders

You can add any library feature, part or assembly to these folders, either using the Design Library panes or Windows Explorer, and they will appear in the Design Library lower pane. They will appear as icons showing their preview pictures.

You can drag and drop files onto the Design Library pane from the open file or from Windows Explorer. Press the Delete key to remove them from the Design Library.

Sub-folders and Panes

Each sub-folder, such as `threaded fittings (npt)`, holds the appropriate type of files (`*.sldprt` in this case). The folder name appears in the Design Library upper pane. The folders and files themselves appear as icons in the lower pane (similar to the left-right relationship in Windows Explorer).

You can also add your own sub-folders in any of the directories. These will appear as folders in the Design Library pane.

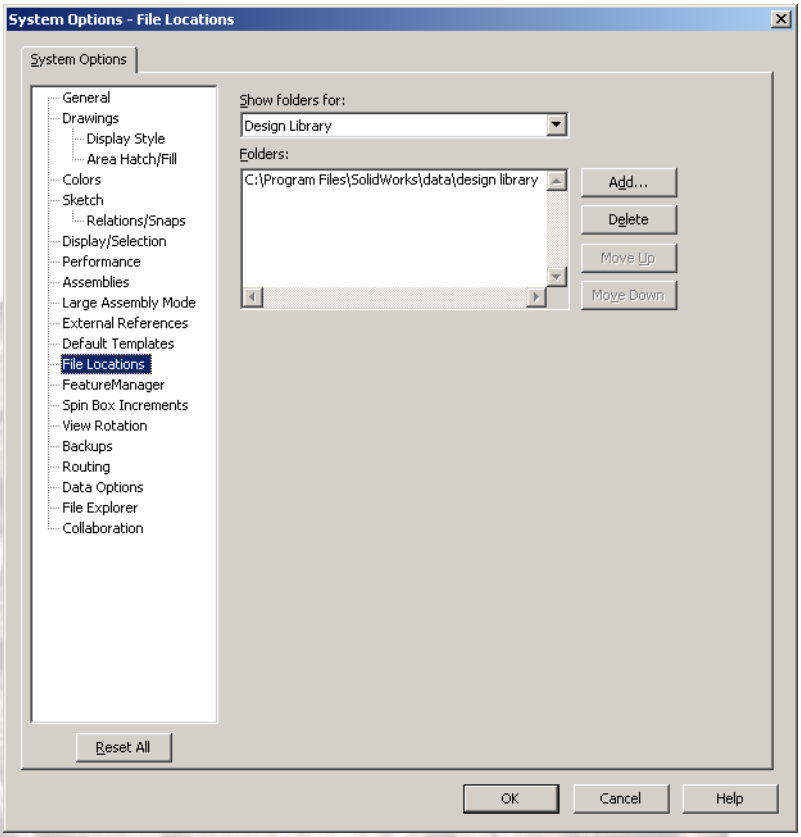
Icons

The graphics of the icon are taken automatically from the last saved image of the library feature or part. They can be shaded or wireframe images, but you should zoom in on them for the best results.

The name of the icon comes from the name of the library feature, part or assembly as it appears within the folder. It can be changed by clicking on it.

Organizing Your Libraries

You can control where the SolidWorks software looks for your libraries by setting a search path in **Tools, Options, System Options, File Locations, Design Library**.



Note

See "**Before You Begin...**" at the beginning of **Lesson 7**, regarding adding the Training Design Library location.

Folders

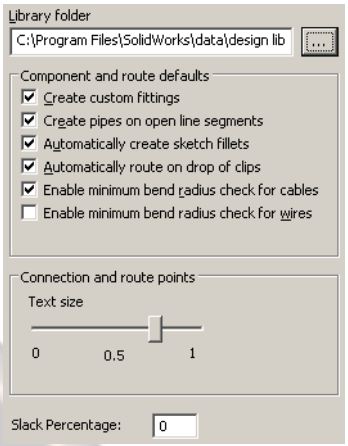
In the **Folders** list you can set search paths for many files that SolidWorks references. A partial list is given below. For a complete list, refer to the online help.

- Library parts, assemblies, features, and forming tools .
- Color swatches, material databases and textures.
- Document templates, referenced documents, sheet formats.
- Blocks, BOM templates, hole callout format files, and dimension favorites.
- Macros, journal files and macro feature files.

Several paths can be set for each category. If you have more than one path, the system searches them in the order they are listed.

Routing Options

Options that are specific to Routing can be found in the **Tools, Options, System Options, Routing.**



- **Library folder path** - Similar to the Design Library location in **File Locations**, this is the default routing component location. It may be the same as a Design Library location, or can be a different folder. This location will appear in the Route Properties dialog and may be edited there as well.

Note It is important to set this path to the Training Design Library before beginning the case studies in Lessons 7 and 8.

- **Create custom fittings** - Automatically creates custom configurations of the default elbow fitting when needed.
- **Create pipes on open line segments** - Generates a pipe for 3D sketch segments that are connected to a fitting at only one end.
- **Automatically create sketch fillets** - Automatically add fillets at intersections as you sketch.
- **Automatically route on drop of clips** - Will auto-route a flexible tube or cable through a properly configured route clip when it is dropped from the Design Library.
- **Enable minimum bend radius check** - For electrical cables only. Reports an error if the bend radius of an arc or spline in the route is less than the minimum specified for the cable in the cable library.
- **Enable minimum bend radius check for wires** - For electrical cables / wires / harnesses only. Reports an error if the bend radius of an arc or spline in the route is less than the minimum specified for the individual wire or cable core in the cable library. If there are many wires in the assembly, this option might result in slower performance.
- **Connection and route points text size** - Scales text for connection and route points to a fraction of the document's Note font.
- **Slack Percentage** - For electrical routes only. Calculated cable lengths are increased by the stated percentage to compensate for sag in flexible routes caused by gravity.

Lesson 6
3D Sketching Review

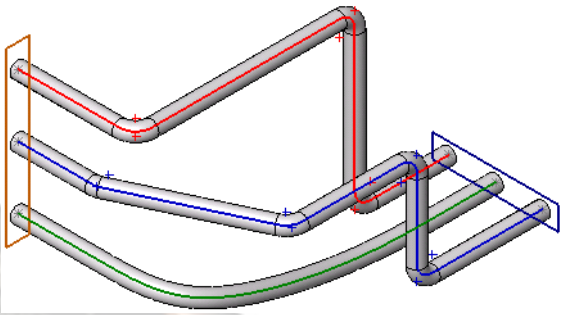
Upon successful completion of this lesson, you will be able to:

- Create a 3D Sketch.
- Use the orthogonal method to sketch using standard planes.
- Use the selected plane method to sketch using user defined planes and planar faces.
- Use the spline tool to create a freeform route line.

3D Sketching

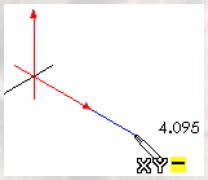
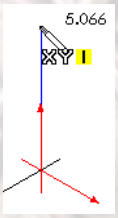
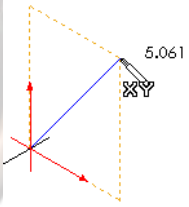
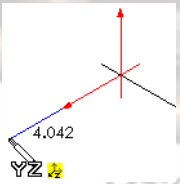
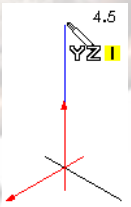
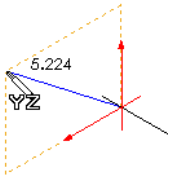
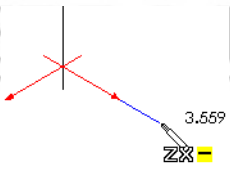
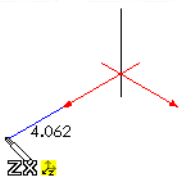
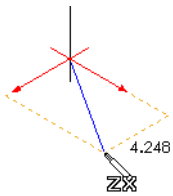
A **3D Sketch** is a sketch that allows you to create geometry in any plane or planar face. In routing, it is used to define the path followed by the tube, pipe or cable.

Orientation changes during sketching are provided by the **Space Handles**, a set of red axes that show the current directions as you sketch. The **Tab** key is used to toggle from the default `Front` axes to those of the `Right` and `Top` planes.



Space Handles

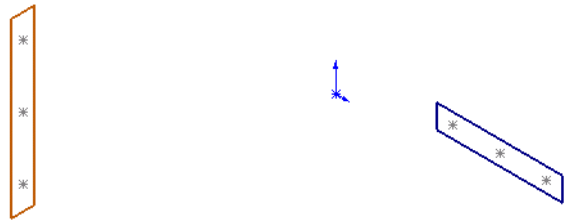
The **Space Handles** are a set of axes that can be changed on the fly while sketching. When the **Line** tool is selected, a set of axes appear with the line as it is sketched. These axes are parallel to those of the `Front` reference plane and allow sketching along those axes or within the plane. Two other planes can be accessed through the **Tab** key.

	Local Horizontal Axis	Local Vertical Axis	Local Plane
`Front` **(Initial plane)**			
`Right` **(Tab once)**			
`Top` **(Tab twice)**			

Note

Regardless of the plane used, the axis labels refer to the axes of the `Front` plane (**X**, **Y** and **Z**).

1 Open part.
Open the part `3D Sketch Example`. The part contains two visible sketches that will be used as connection points.



Orthogonal 3D Sketching

Using the *standard* three planes for sketching allows you to sketch along the **X**, **Y** and **Z** axes of the `Front` plane. You can also sketch on any plane formed by a pair of the axes.

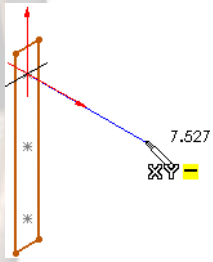
Where to Find It

- Click  from the Sketch toolbar.
- Click **Insert, 3D Sketch**.

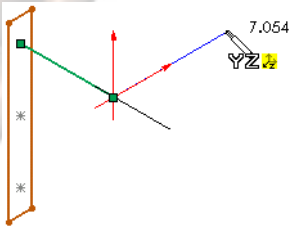
2 New 3D Sketch.

Click **3D Sketch**  to open a new 3D Sketch.

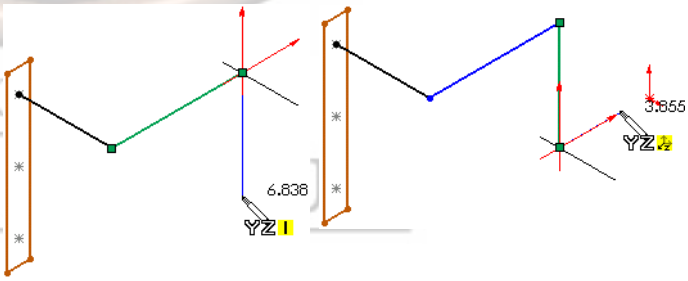
3 Along X Axis.
Clicking the **Line** tool and start a new line at the upper point and drag the other end of the line. Position along the axis until the `X` label appears indicating that the line is on the `X` axis. Extend the line as shown.



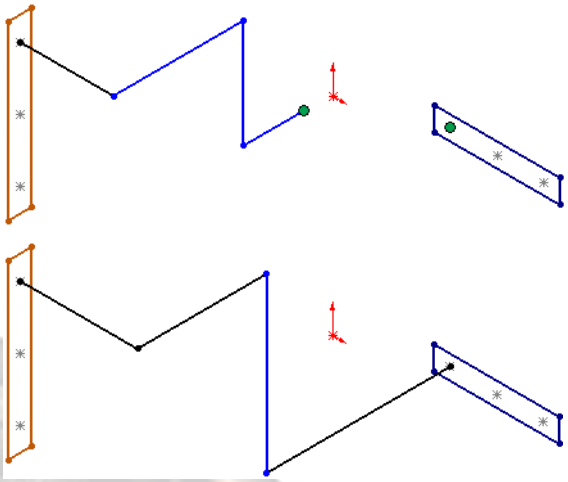
4 Using the Space Handles.
Begin the next line at the open endpoint and drag the line. Press the **Tab** key to orient the **Space Handles** to the orientation shown. Drag the line along the `Y` axis. Locate the end of the line roughly as shown.



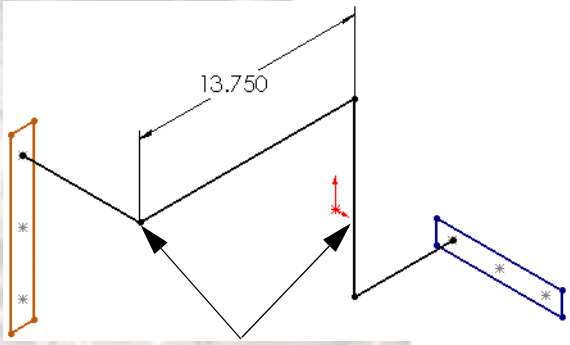
5 Along Y Axis.
Using the same set of axes, sketch a line along the `Y` axis followed by the `Z` axis.



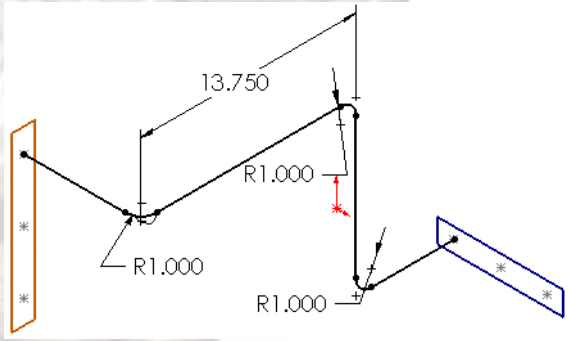
6 Coincident.
Add a **Coincident** relation between the open endpoint and the left most point in the sketch.



7 Dimension.
Add a dimension between the vertical line and endpoint as shown. Set the value to **13.75”**.



8 Fillets.
Add fillets of radius **1”** to the corner endpoints.



Tip

3D Sketches offer a few more relations than a standard sketch. They are **ParallelYZ**, **ParallelZX** and **AlongZ** and refer to the planes and axes of the sketch.

9 Exit the sketch.

Sketching on Selected Planes

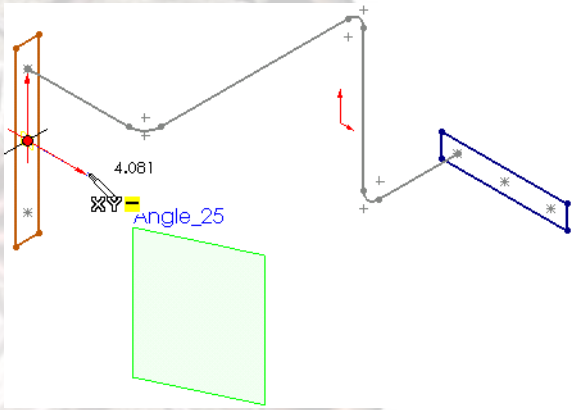
3D Sketching can use planes other than the standard `Top`, `Front` and `Right`. An alternative plane or planar face can be selected by **Control**-clicking it while sketching. This process activates a set of Space Handles with the `X` and `Y` axes parallel to the selected plane. Two other sets of axes (rotations of the first) are also available through the **Tab** key.

Note

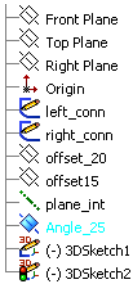
Performing this process during Routing (**Edit Route**) requires only a selection without the Control key.

1 **Show Plane.**
Right-click the reference plane `Angle_25` and select **Show**. Click **Select** to turn off highlighting of the plane.

2 **Along X axis.**
Create a new 3D Sketch. Sketch a line starting on the middle point and running along the `X` axis as shown.

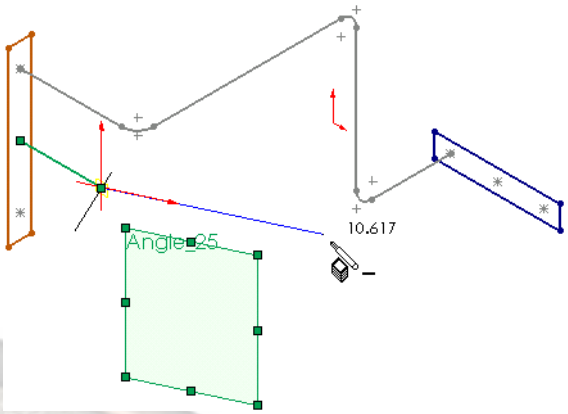


3 **Switch Plane Reference.**
Control-select the plane `Angle_25` from the Flyout FeatureManager design tree.



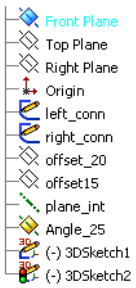
4 Along Plane.

Sketch along the X axis of the `Angle_25` plane as shown.



5 Return to Standard Planes.

Control-click the `Front` plane to return to the standard plane selections.

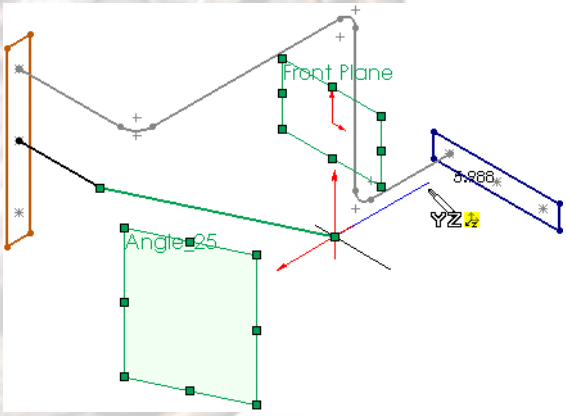


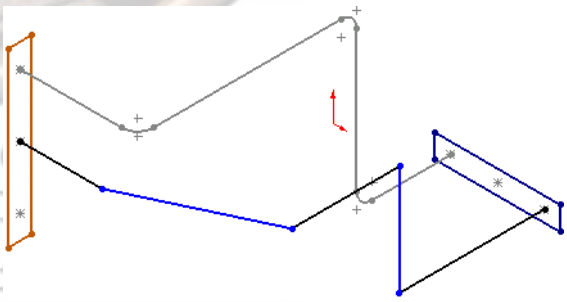
6 Along Z.

Use the **Tab** key to switch to the plane orientation shown. Sketch along the `Z` axis.

Continue along the `Y` and `Z` axes to complete the lines.

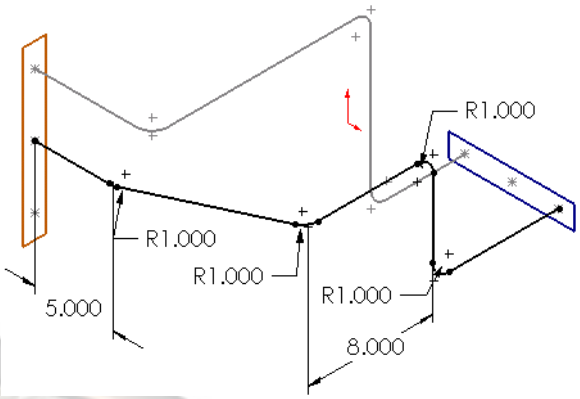
Add a **Coincident** relation to tie the endpoint to the sketch point.





7 Dimensions.
Add the dimensions **5”** and **8”** as shown. Also add radius **1”** fillets.

Close the sketch.



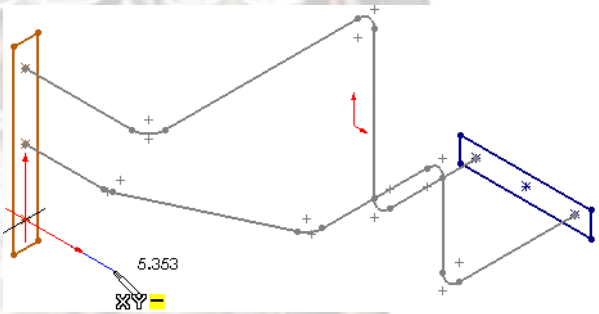
Spline

The 3D Sketch can be used to represent a “flexible” path between connection points. This method uses **Spline** to form smooth 3D curve between lines. The lines remain to help shape the spline.

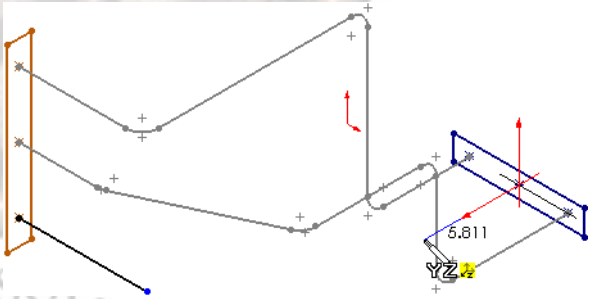
Note

This type of sketch can not be used with piping.

1 Starting Line.
Create a new 3D Sketch. Sketch a line starting at the lower left sketch point along the X axis.



2 Ending Line.
Create a second line starting at the center sketch point along the Z axis.



Spline

Spline is used to form a shape that is more free-form than orthogonal segments. The spline can be set to be tangent to straight segments.

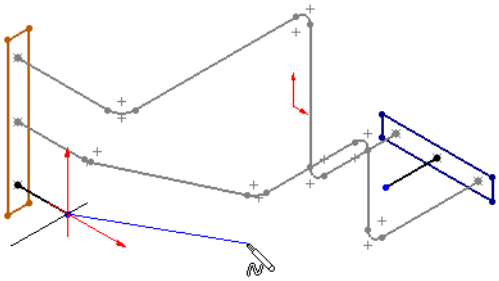
Where to Find It

- Click  from the **Sketch** toolbar.
- Click **Tools, Sketch Entities, Spline**.

3 Spline.

Click **Spline**, then click one of the straight segment endpoints. Drag the shape toward the other endpoint.

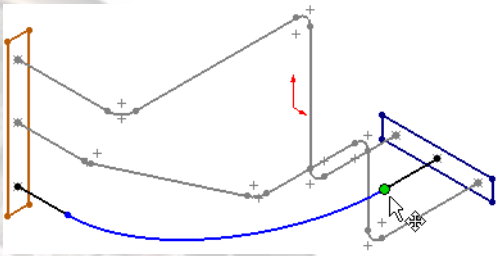
You can use either click-click or click-drag method.



Note You can add intermediate spline points, by clicking the left mouse button, to help form the desired shape.

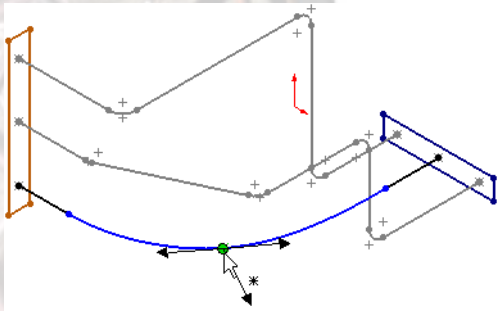
4 End the Spline.

Click on the endpoint of the other line segment to finish the spline. Right-click and choose **End spline**.



Add **Tangent** relations between the spline curve and each of the straight segments.

The spline can also be shaped by dragging the endpoints, or adding intermediate spline points after creation.



Note When using a spline in routing, the tangent relations are added automatically.

3D STUDIOS
P.L.R.
Profesor:
PEDRO LUIS RUIZ

Lesson 7
Tubing Routes

Upon successful completion of this lesson, you will be able to:

- Route flexible tubing using the spline tool.
- Route rigid tubing using 3D orthogonal sketching.
- Using Auto Route for both rigid and flexible tubing.

Tubing Routes

Tubing Routes use tube parts shaped by 3D Sketches to form sub-assemblies that contain tubes and fittings. The tube can be shaped orthogonally (rigid tube) or flexibly (hose or malleable tube).

Before You Begin...

Supplied with the training files for this and other case studies in this manual are simplified versions of the Part and Assembly Design Library directories. The purpose of this is to avoid conflict between your Design Library, installed with the SolidWorks software, and the Design Library-based parts that exist already in these training assemblies.

File Locations

Before starting these lessons, you should open the **Tools, Options** dialog, and on the **File Locations** tab, browse to and add these training paths to the Design Library.

```
C:\SolidWorks 2005 Training Files\Routing\training design library
```

Routing

Next, on the **Routing** tab, under **Library folder**, set to

```
C:\SolidWorks 2005 Training Files\Routing\training design library\routing
```

Also, click **Automatic route on drop of clips**.

Note

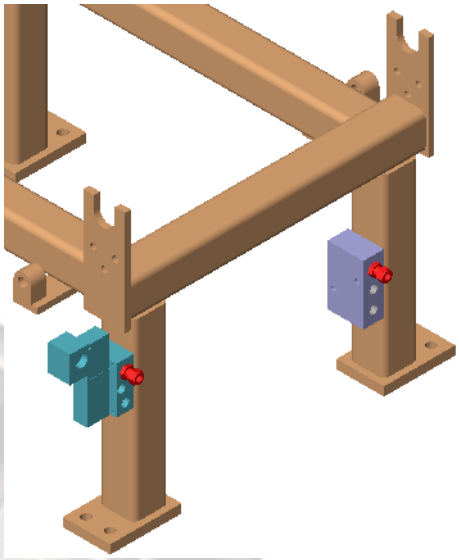
This is the default install location for the SolidWorks training files. If the files were installed in an alternate location, this path should be set accordingly.

These locations should be used for all component insertions in this manual. It is important to set this path first to avoid conflict with the files in the other (SolidWorks installed) Design Library location.

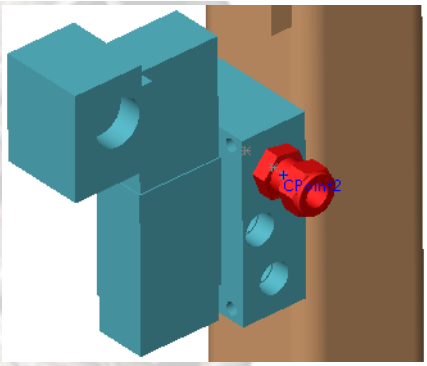
Note

Some of the paths displayed in illustrations of dialogs in the manual may not coincide with this guideline.

1 **Open assembly.**
Open the assembly `Tubing Assembly` in the `Tube Route` directory. The assembly includes two tubing fittings, `solidworks-lok male connector<1>` and `<2>`. Each fitting contains the routing points `CPoint2` and `RPoint1`.



2 **Route Points.**
Show the routing points using **View, Routing Points**.



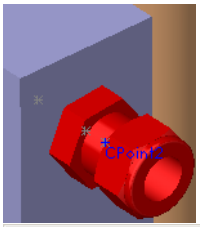
Route Properties

The **Route Properties** are settings that define the characteristics of a route. It is used to determine which tube part is used on the route 3D Sketch, whether a flexible or orthogonal route is used and whether bends or elbows are used at changes in direction.

The dialog is triggered when a new route is created.

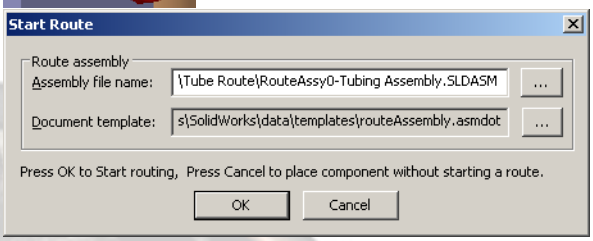
3 Start Route.
Zoom in on the `manifold` part.

Right-click the CPoint and select **Start Route**.

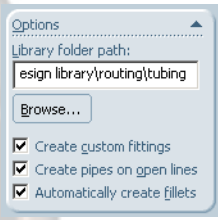


The **Start Route** dialog requests an **Assembly file name**. This name is used for the routing sub-assembly.

Leave the default name and click **OK**.



4 Route Properties Dialog.
Click **OK** and the **Route Properties** dialog appears.



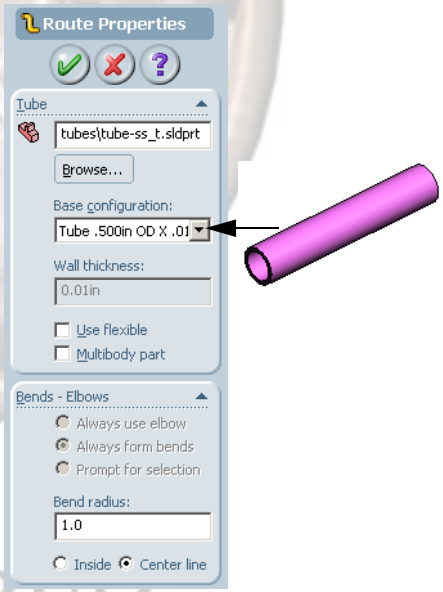
Under **Options**, choose `training design library\routing\tubing` as the **Library folder path**. This sets the proper folder for the fittings chosen to represent the tube.

For **Tube**, choose `tube-ss_t` in the `Tubes` directory and **Base configuration** `Tube .500in OD X .010in wall`.

Clear **Use flexible**.

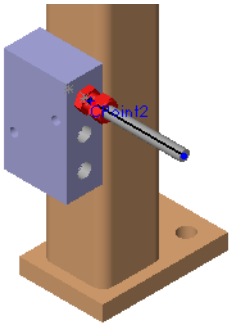
For **Bends - Elbows**, choose **Bend radius: 1”**and **Center line** position.

Click **OK** to start the route.

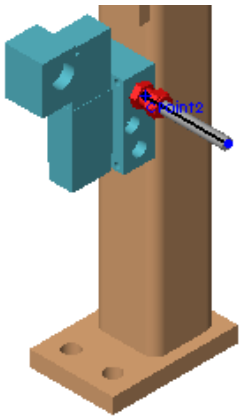


Note The **Base configurations** are restricted to those that match the nominal diameter set by the **Connection Point**.

5 Drag Route Line.
A short route line is started at the CPoint location. Select the endpoint of the line and drag it, increasing the length of the line. This is the start of the route.



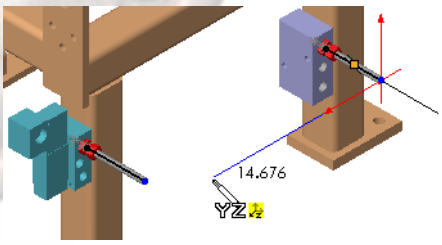
6 Add.
Right-click the CPoint on the opposite side of the assembly and select **Add to route**. This adds another small route line as part of the current route.



3D Sketch Route

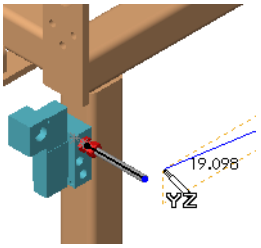
Routes can also be created from a manual 3D Sketch. The sketch is created in the **Edit Route** mode that occurs after Route Properties are completed.

7 Orthogonal route.
Complete the route by sketching an orthogonal line from the initial segment toward the end segment. **Tab** once to use the YZ plane.Drag **Along Z**, but don't release the endpoint yet.

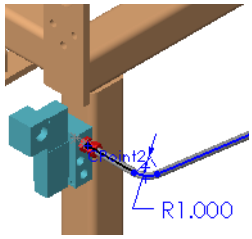


8 Snap.
As with 2D sketching, lines will snap to horizontal and vertical positions. However these two segments are not on the same Y plane, so a horizontal sketch will not close the loop.

Drag the endpoint up to create an angled tube segment. The cursor shows we are still on the YZ plane.



9 Finish the sketch.
Drop the endpoint on the open endpoint of the other segment. The bend is added automatically.



Note

This sketch could also be completed with a conventional X-Y-Z route, with 3 bends, using **Merge** to join open ends.

10 Close sketch and save.
Close the sketch. You are prompted to save the tubing part created. Click **Save**. Return to editing the top level assembly.

Deleting a Route

Routes are sub-assemblies that can be edited or deleted as components. Deleting a route deletes all the components within it including tubes, 3D Sketches and fittings.

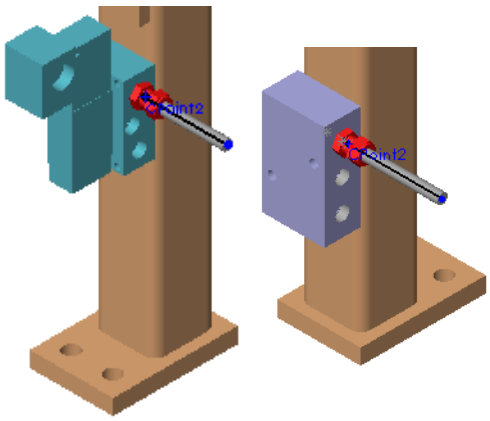
11 Delete the route.
Select the component `RouteAssy1-Tubing Assembly` and press the **Delete** key. Click **Yes** to confirm deletion.

Spline Sketch

Routes for flexible tubing can be created using a spline. The 3D sketch is created in **Edit Route** mode, but uses the spline tool instead of the line tool. 3D space is respected as the tool will form the spline as necessary through changes in X, Y and Z.

12 Start a new route.

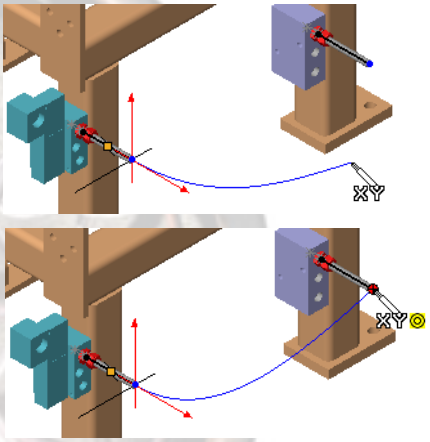
From the same CPoint, start a new route, with all the same parameters, *except* this time check **Use flexible** in the **Route Properties** dialog. This will allow for the use of a spline curve.



Add another route segment to the `.375 AIR CONTROL VALVE` as before.

13 Close off with a spline.

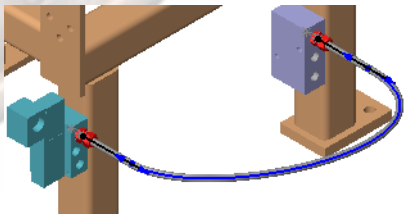
Select the spline tool. Click and drag from one endpoint to the other.



Note

It is not necessary to use **Tab** to change planes when dragging directly to an existing endpoint. The system will intuitively align itself in 3D space.

When released, the ends of the spline form tangent relations with the straight segments automatically.



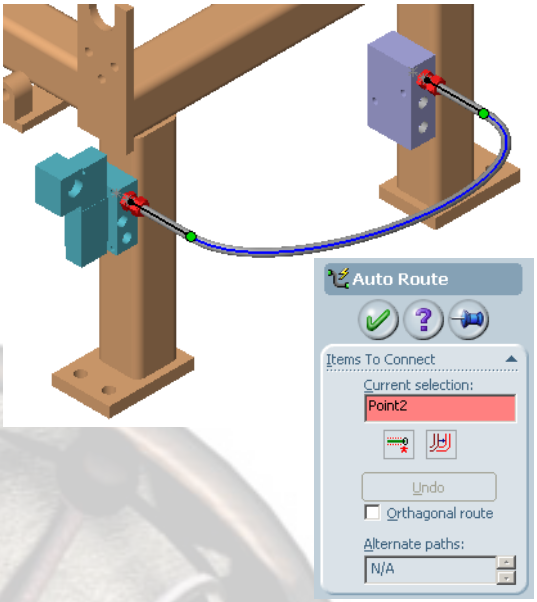
Auto Routing

Tubing routes, both flexible and orthogonal (or orthographic), can be automatically generated using the **Auto Route** option.

14 Delete spline.

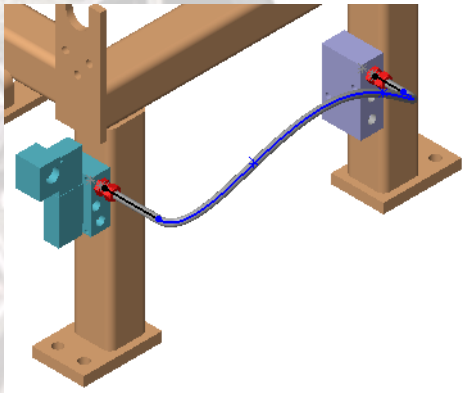
Delete the existing spline segment. Leave the start and end sketch line segments.

15 Auto Route Settings.
Right-click one of the open endpoints and choose **Auto Route**. Select the endpoint of the other segment. A spline is added connecting the endpoints. It is tangent to both line segments. Click **OK**.



16 Shape Spline.
Drag the line endpoints to shape the spline.

The spline shape can be further edited with the addition of spline points (right-click **Insert Spline Point**).

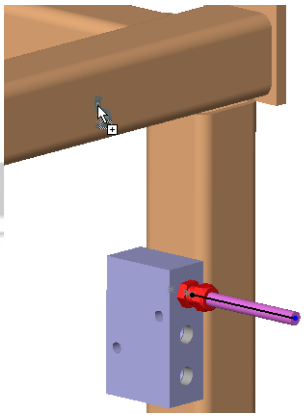


Tip For clearest manipulation of the added spline points, switch to side, front or top views of the design.

Clips Auto routing can also be achieved by the drop of a clip from the Design Library. The clip must have appropriate RPoints and a mate reference.

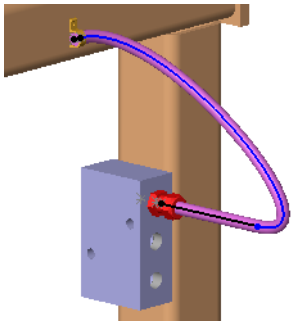
17 Delete spline.
While still editing the sketch, delete the spline segment just created. Leave the two straight segments intact.

18 Drag and drop a clip.
From the Training Design Library, in the `routing\tubing` directory, drag the `tubing-clip 0.5` into the assembly. Drop it on one of the small holes in the horizontal member above the tube.



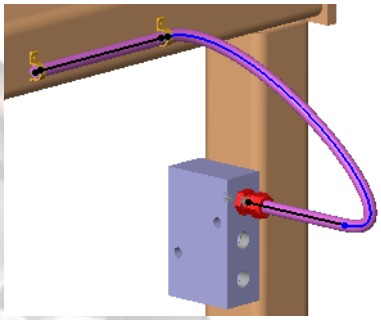
Note Routing by dropping clips works with *flexible* tubing, not rigid,or with electrical wiring only (**Use flexible** checked in route properties).

19 Result.
The tube is routed automatically through the clip. Two segments are added; a spline and a straight segment defined by the clip.



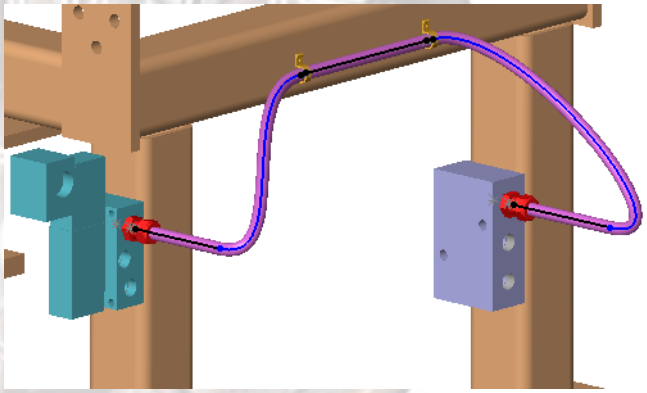
20 Repeat.
Add a second clip on the other hole. Again, the route is extended automatically.

Exit **Insert Component** mode.



21 Close the route.
Finish the route by using **Auto Route** between open endpoints.

If the minimum radius of curvature is exceeded, it can be adjusted afterwards by dragging the straight segments.

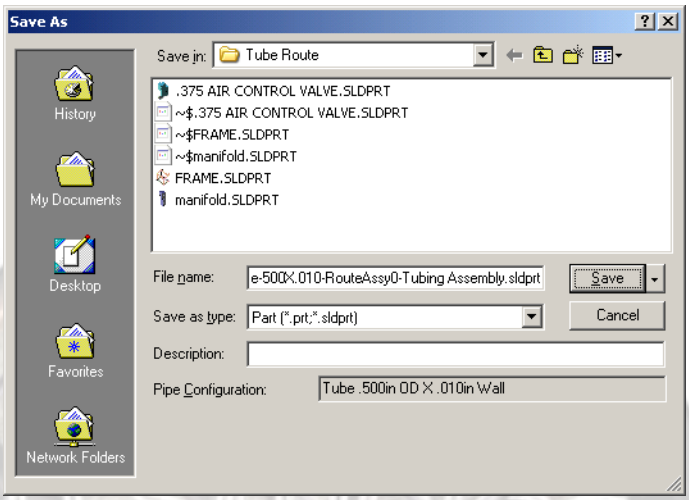


Note

Clips and other route components can be selected manually to establish a sequence. Click **View, Axes** and select the axis within the routing component.

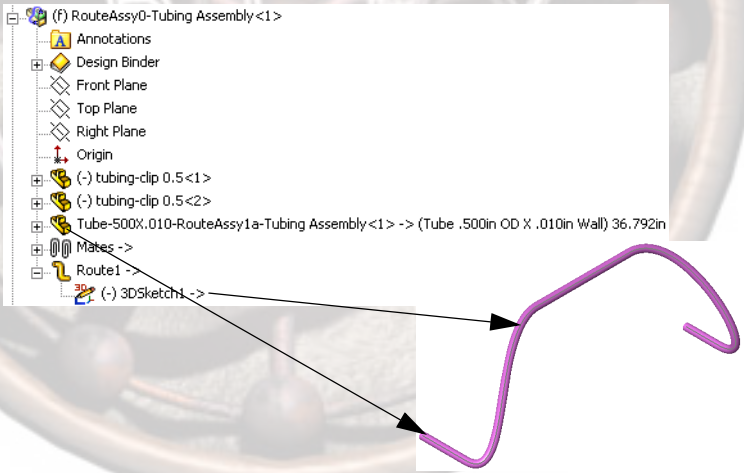
22 Save tube.

Exit the sketch. The tube part used in the route is saved with the route from the Part Design Library. The default name is generated from the name of the configuration used and the parent assemblies.



Click **Save**.

23 Completed Tubing Route.



The tube is `Tube-500X.010-RouteAssy0-Tubing Assembly` with the configuration `Tube .500in OD X .010in Wall`. The length of the tube is listed following the configuration.

The **Route1** component includes the route sketch, `3DSketch1`.

Both the tube and the route are created in-context as indicated by the **->** suffix.

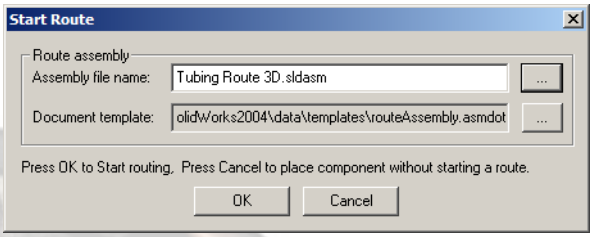
24 Delete route.

Select the component `RouteAssy0-Tubing Assembly` and press the **Delete** key. Click **Yes**.

Orthogonal Auto Route

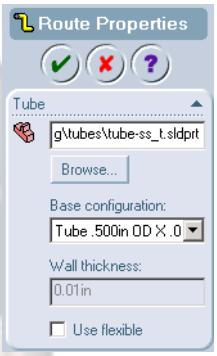
Routes can also be created using Auto Route in an orthogonal scenario, rather than flexible. The sketch is created in the **Edit Route** mode that occurs after Route Properties are completed and uses right-angle bends only.

25 New route.
Start a new route from a CPoint using the same procedure as before. Type the custom assembly name `Tubing Route 3D` and click **OK**.



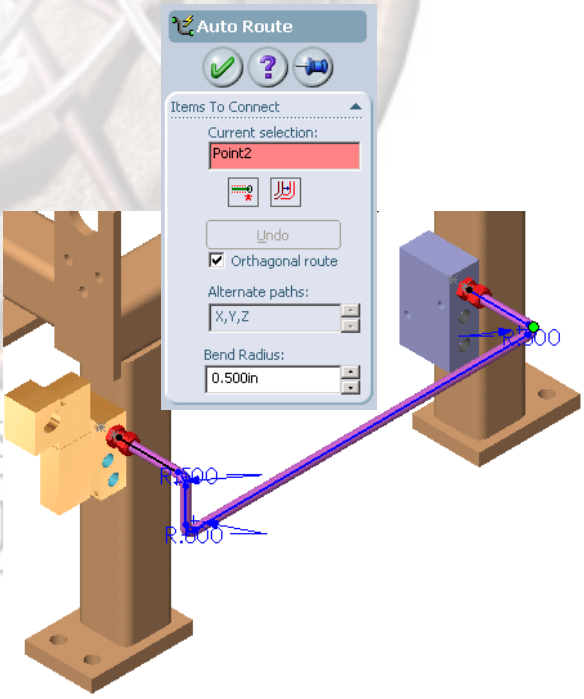
26 Route Properties.
Use the same settings in Route Properties but clear **Use flexible** and set **Bend radius** to **0.5”**.

Click **OK**.



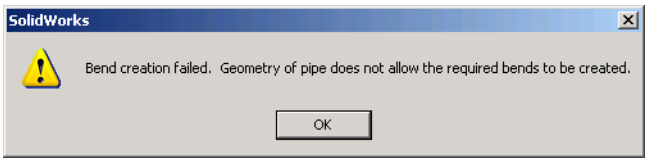
27 Auto Route.
Drag the open endpoints to about twice the length of the fitting.

Next select **Auto Route** and the **Orthogonal route** option. Use the up arrow in **Alternate paths** to move through each possible solution. Use the solution displayed at right.



Note

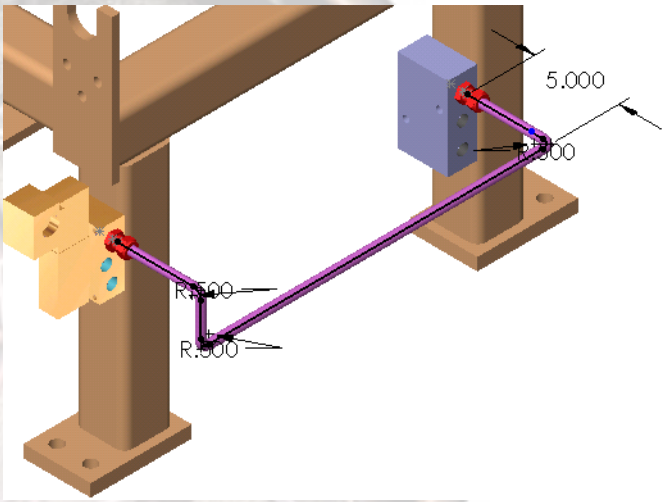
Some solutions can create invalid routes. When this happens, a warning dialog appears.



For example, when there is not enough space between two perpendicular segments to put in a full radius of the default size, the segments will show as a sharp corner. An adjustment to that particular radius or to the separation will be needed to show a proper radius.

Simply click **OK** to continue.

28 Fully define. Add the **5in** dimension as shown to fully define the sketch. Exit the sketch and save the files.



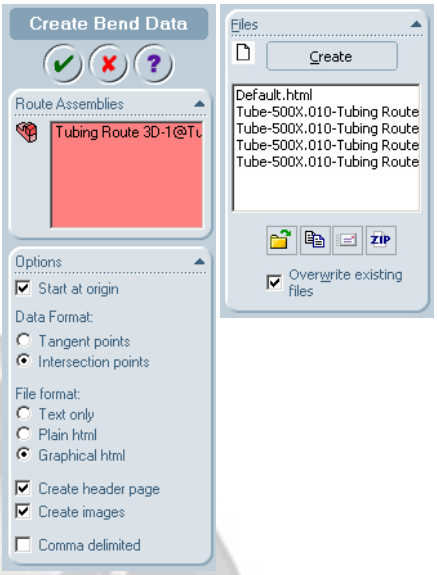
Create Bend Tables

Bend Tables can be used to extract detailed information from a tubing route for fabrication. The information is available in several file formats including text, html and eDrawings™ eprt file.

29 Create Bend Data.
Right-click `Tubing Route 3D` and select **Create Bend Tables**.

For **Options**, click those shown at right and click **Create**.

The files created are listed in the **Files** group box.



Note

You must have eDrawings™ installed to view the `*.eprt` file. The files are placed in a folder named `Tubing Assembly` under the current folder.

30 View file.
Select the `Tube-500X.010-Tubing Route 3D-Default.html` file and click **View Selected File(s)**  .

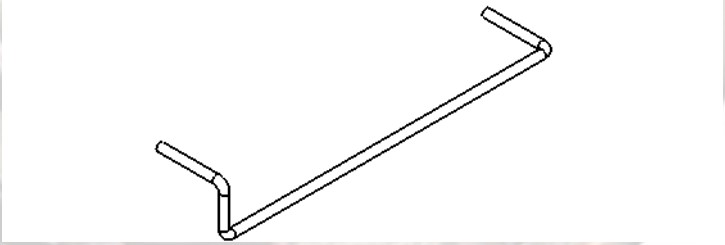
X	Y	Z	RADIUS
0.0000	0.0000	0.0000	0.0000
5.0000	0.0000	0.0000	0.500
5.0000	0.0000	20.86000	0.500
5.0000	3.0000	20.86000	0.500
4.500	3.0000	20.86000	0.0000

Other options with the files include **Copy All Files**  , **Send Email Containing All Files**  and **Create a Zip Archive Containing All Files**  .

31 Bill of Materials.
Create a drawing of the `Tubing Route 3D` sub-assembly with one isometric view.

Select the view, and insert the BOM (**Insert, Tables**, **Excel Based Bill of Materials...**) using the `Piping BOM Template` (located in <SolidWorks install directory>`\lang\english` folder). It includes the `CUT LENGTH` property.

ITEM NO.	QTY.	PART NO.	DESCRIPTION	CUT LENGTH
1	1	Tube .500in OD X .010in Wall		31.369in



32 Alternate BOM.
Alternatively, you can insert a standard (non-Excel) Bill of Materials and customize the cells or use the supplied `Tubing BOM Template.sldbomtbt` template with **Insert, Tables, Bill of Materials**.

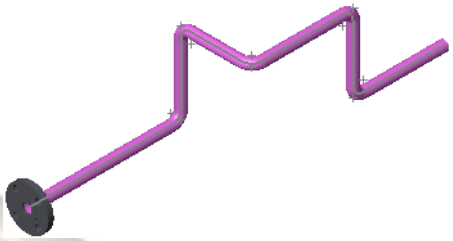
ITEM NO.	PART NO.	DESCRIPTION	CUT LENGTH	QTY.
1	Tube .500in OD X .010in Wall	Tube-500X.010_1-Tubing Route 3D	31.369in	1

The template is located in the `training design library` folder.

Editing a Route

Tubing and piping routes can be edited to change: route diameters, add fittings, remove tube/pipe sections and create pipe penetrations. Most of these operations are performed while in **Edit Route** mode.

1 **Open `Tubing Options`.**
Open the assembly `Tubing Options` in the directory of the same name. It contains a flange and tube route.



Change Route Diameter

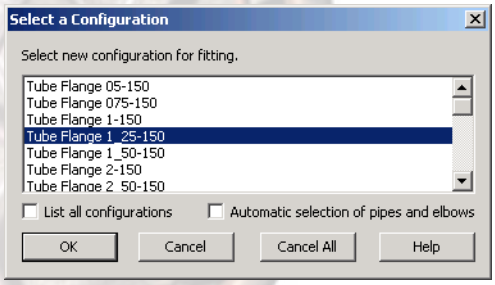
Change Route Diameter is used to change the diameter of the entire route including the tube and related fittings. In this example, the current nominal diameter of the tubing is **1 inch**. This option is available in **Edit route** mode.

Where to Find It

- Right-click a line of the route and select **Change Route Diameter**.

2 **Edit route.**
Edit the existing route.

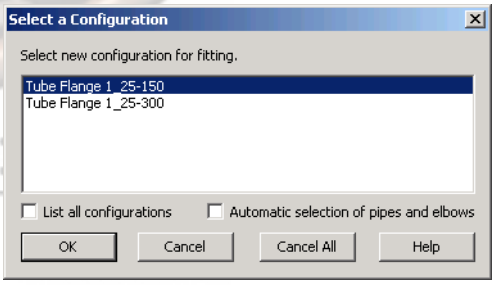
3 **Larger Diameter.**
Right-click the route and select **Change Route Diameter**.
Select the configuration `Tube Flange 1_25-150` from the dialog and click **OK**.



Note

Using the **Automatic selection of pipes and elbows** check box would prevent the second dialog from appearing.

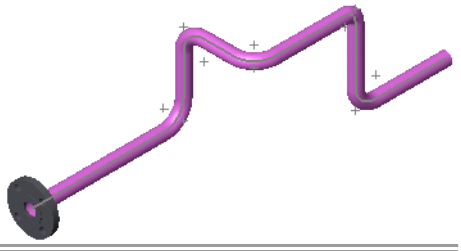
In the next dialog, select the same configuration name.



Note

If the Route Properties dialog appears. select the file `tube_ss_t` for the pipe.

4 Rebuild.
Both the flange and tubing have been changed to match **1.25”** diameter tubing. The *name* of the tubing part does *not* change with the diameter change.



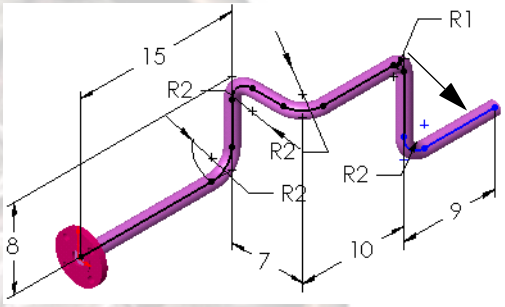
Add Fitting

Fittings such as flanges and tees can be added directly to the route without using the Design Library. This option is available in **Edit Route** mode.

Where to Find It

- Right-click a line of the route and select **Add Fitting**.

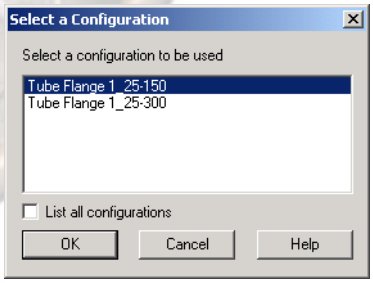
5 Edit the route.
Edit the route, then right-click the open endpoint of the sketch and select **Add Fitting**. Browse to the `Training Design Library` directory and select the `slip_on_tube_flange-ss_t` component. from the `Tubing` folder.

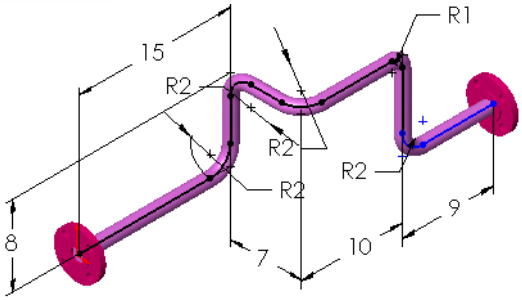


Note

It is important that you use the part from this directory because this is the same part that already exists in the assembly (a second instance).

Select the configuration `Tube_Flange 1_25-150` and click **OK**.





Constrain Sketch to Fitting

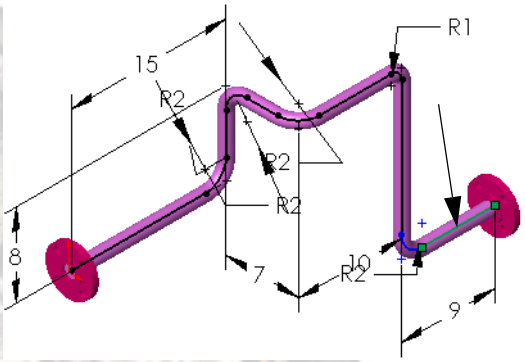
Fittings that are added to an existing route endpoint are controlled by the sketch, moving with changes to the attachment point. **Constrain Sketch to Fitting** can be used to reverse that by repositioning the fitting and driving the sketch. This option is available in **Edit Sub-assembly** mode and is usually used with piping.

Where to Find It

- Right-click a pipe of the route and select **Constrain Sketch to Fitting** or **Constrain Fitting to Sketch**.

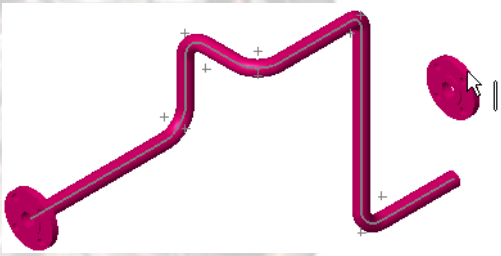
---

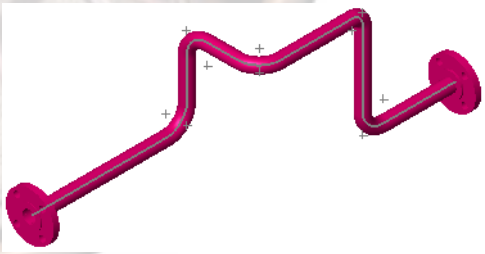
6 **Drag endpoint.**
Edit the route sketch and drag the line indicated to reshape the route. The component is attached to the line and moves with it.



7 **Drag fitting.**
Exit the 3D Sketch, remaining in **Edit Sub-assembly** mode. Right-click the `slip_on_tube_flange-ss_t<2>` component and select **Constrain Sketch to Fitting**. Drag the fitting component and drop it. The route will re-connect to it.

Select **Constrain Fitting to Sketch** to return to control by the sketch.





8 **Save and close.**

Lesson 8
Piping Routes

Upon successful completion of this lesson, you will be able to:

- Route fabricated piping using orthogonal 3D sketching and angled 3D sketching.
- Use an alternate elbow in a piping route.
- Use orthogonal auto routing with pipes.
- Edit a piping route and add fittings to a route.

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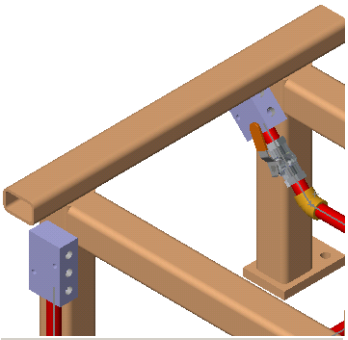
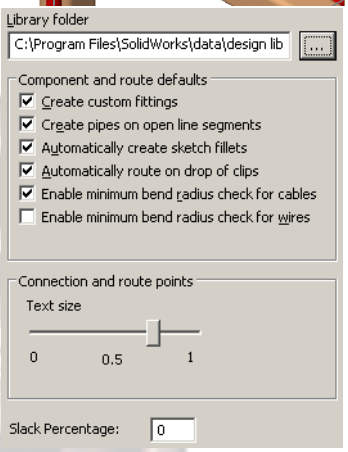
Profesor:
PEDRO LUIS RUIZ

Setup for Piping

As before, be sure **Tools, Options, System Options, Routing, Library folder path** is set to the training design library directory:

`training design library\routing\piping`



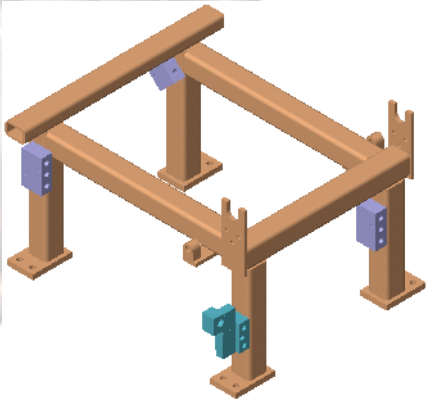


Sketching a Route

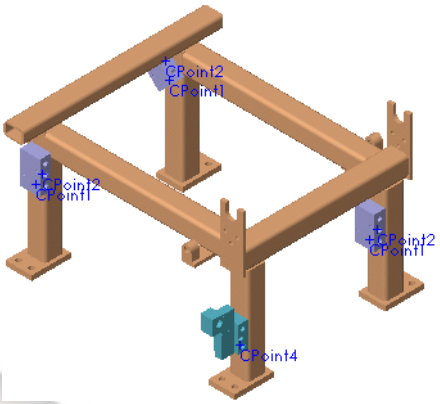
This route will be sketched in a similar manner to the 3D Sketch of the previous lesson.

1 **Open assembly.**
Open the assembly `Piping Assembly`.

2 **Set Editing Colors.**
Click **Tools, Options, System Options** and **Color**. *Clear* the option **Use specified colors when editing parts in assemblies**.



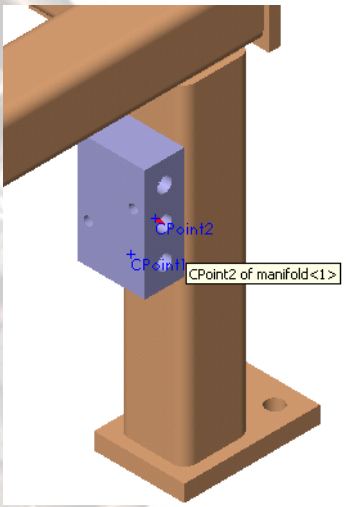
3 Routing components.
The components `.375 AIR CONTROL VALVE` and three instances of the `manifold` are routing components. Click **View, Routing Points** to see the connection and routing points of each one.



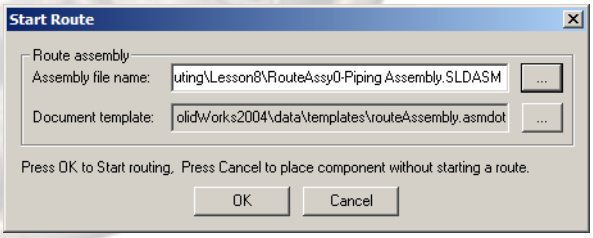
Starting the Route

Piping routes are started in the same way as tubing routes.

4 Start route at CPoint.
Start the route at a the connection point `CPoint2` of `manifold<1>`. Right-click the **CPoint** and select **Start Route**.



5 Start Route Settings.
In the **Start Route** dialog, use the default name of the route sub-assembly and use the default template.



6 Route Properties.
Click **OK** and the **Route Properties** dialog appears.

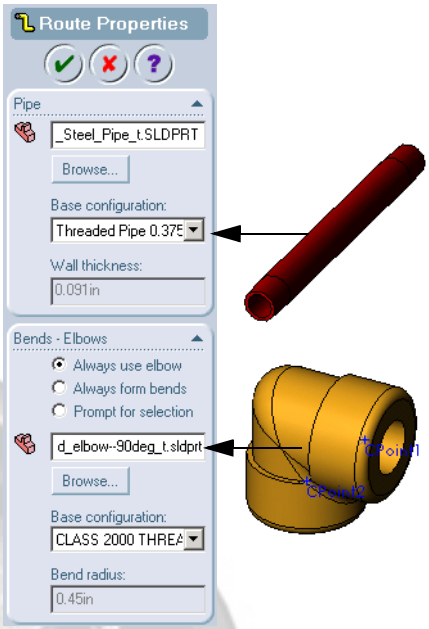
Under **Options**, choose `training design library\routing\piping\Threaded Fittings (NPT)` as the **Library folder path**. This sets the proper folder for the fittings chosen to represent the pipe and bends.

For **Pipe**, choose `Threaded_Steel_Pipe_t` and **Base configuration** `Threaded Pipe 0.375 in, Sch 40`.

For **Bends - Elbows**, choose `Threaded_Elbow--90deg_t` and **Base configuration** `CLASS 2000 THREADED ELBOW, .375IN`.

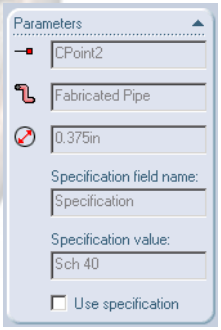
Note that the base configurations are restricted to those that match the nominal diameter set by the **Connection Point**.

Click **OK**.

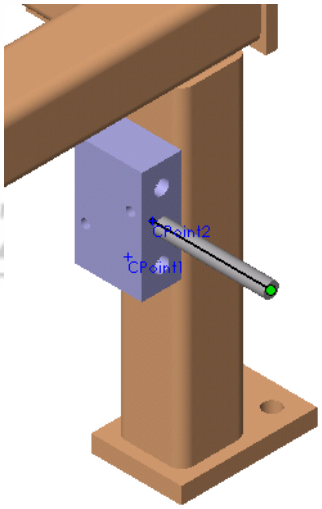


Note After adjusting the **Library folder path**, it is a good idea to reload the part files (tube, pipe and elbow) via the **Browse...** buttons to insure that the correct source file is displayed and loaded.

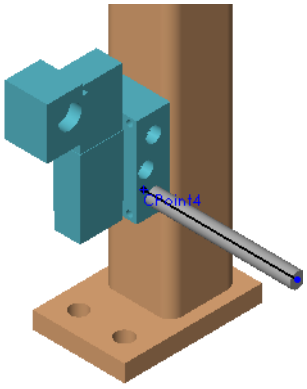
Note The **Parameters** option can be used to limit selections to a certain **Specification** such as a schedule or thickness value.



7 Route is started.
The route line is started as a short line in the direction indicated by the connection point. Drag the endpoint to make the line longer. This is the start of the route.



8 Add to route.
Right-click the `CPoint4` on `.375 AIR CONTROL VALVE<1>` and select **Add to route**. Drag the endpoint away from the `Frame`. This is the end of the route.

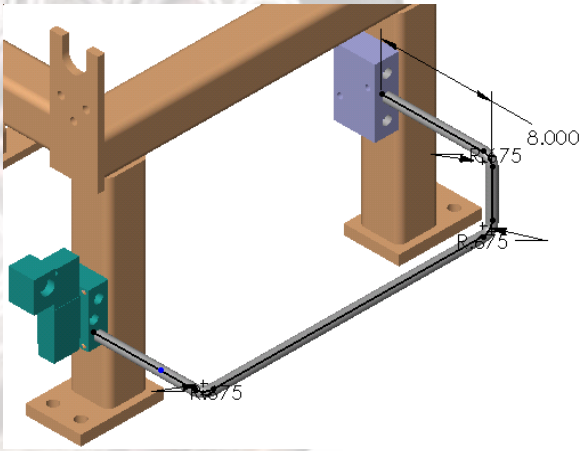


Piping Auto Route

Piping routes require lines and arcs in 3D space. In order to create these, a 3D Sketch is required. In this example, the sketch will be created automatically using and Auto Route.

9 Auto Route.
Select the open endpoints and use an **Orthogonal route**. Select the solution that looks like the one shown at right.

Add the **8in** dimension to the starting line.



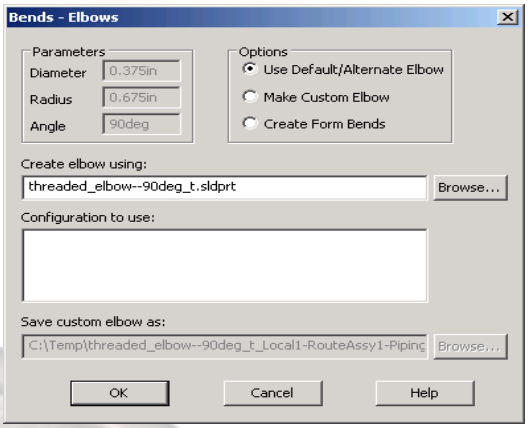
10 Exit the sketch.
Close the sketch and **Save** the pipe part with the default name.

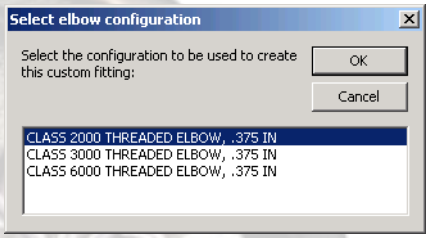
Elbow Selection

Piping routes require elbows at the common endpoints (bends) of lines. To complete the route, identify the proper elbow to use in the **Bends - Elbows** dialog.

11 Elbows.

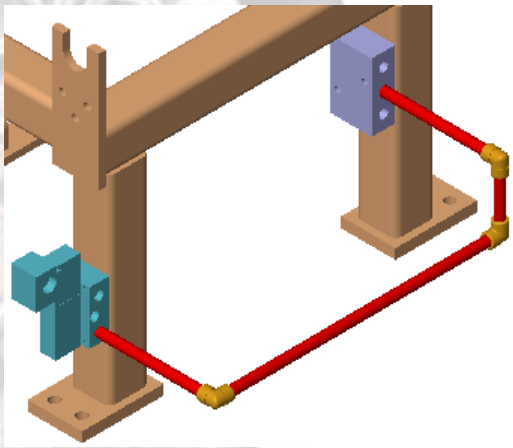
Select elbows for the route. In this example, all elbows use the `threaded_elbow--90deg_t` part with the `CLASS 2000 THREADED ELBOW, .375 in` configuration.



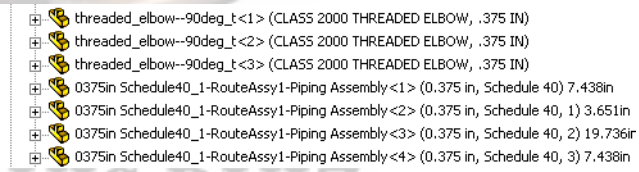


12 Completed route.

The route includes four pipe spools and three 90 degree elbows.

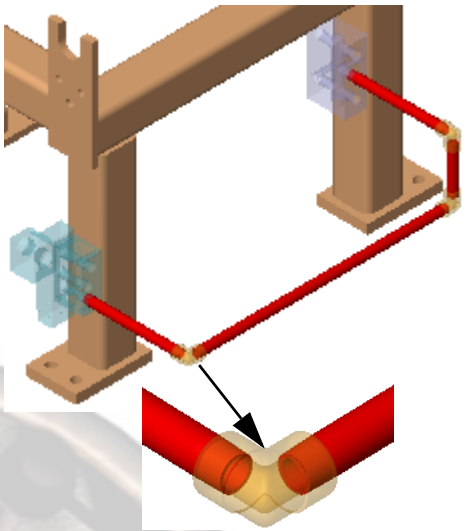


The pipe component instances include the individual pipe lengths, listed after the configuration name.



13 Pipe Lengths.
Changing the transparency of the elbows shows where the pipe segments end. The pipe is stopped at the CPoint locations in the fittings.

Hide the sub-assembly `RouteAssy0-Piping Assembly` for clarity in the next section.



Angled 3D Sketch Route

Routes that include angles other than 90° between lines induce some selection options when the route is completed. An example of this might be a drain pipe that requires a pitch other than horizontal.

In order to satisfy the route, you have options to:

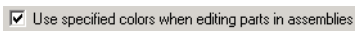
- **Use Default/Alternate Elbow**
  Select an alternative to the default elbow selected in the Route Properties. If the library contains an elbow of the proper angle (commonly a 45) it can be selected and used.
- **Make Custom Elbow**
  Create a custom elbow from a selected elbow such as the default 90° elbow (except threaded). The selected elbow is copied and modified to the angle formed. The component name is suffixed with `_Local#` and the configuration name includes the diameter, angle and radius of the elbow.

Note

Creation of a custom elbow from a standard threaded elbow, though possible within SolidWorks, is not practical or realistic.

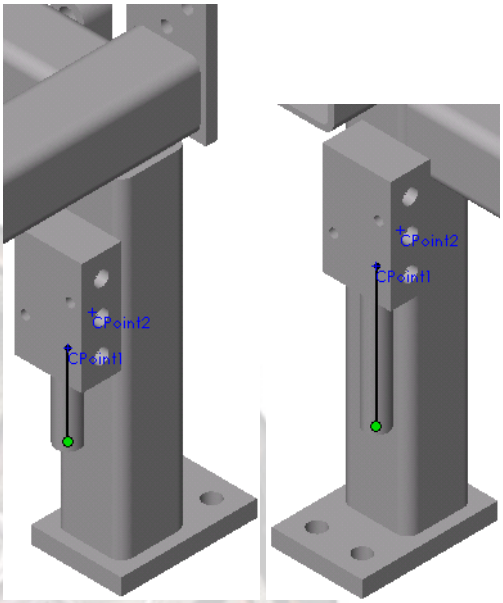
- **Create Form Bends**
  Form bends can be used where the piping is bent like tubing rather than connected with elbows.

1 Set Editing Colors.
Click **Tools, Options, System Options** and **Color**. Check the option **Use specified colors when editing parts in assemblies**.



This option changes the colors of component being edited to **Assembly, Edit Part** color. All other components are set to the **Assembly, Non-Edit Parts** color (default grey).

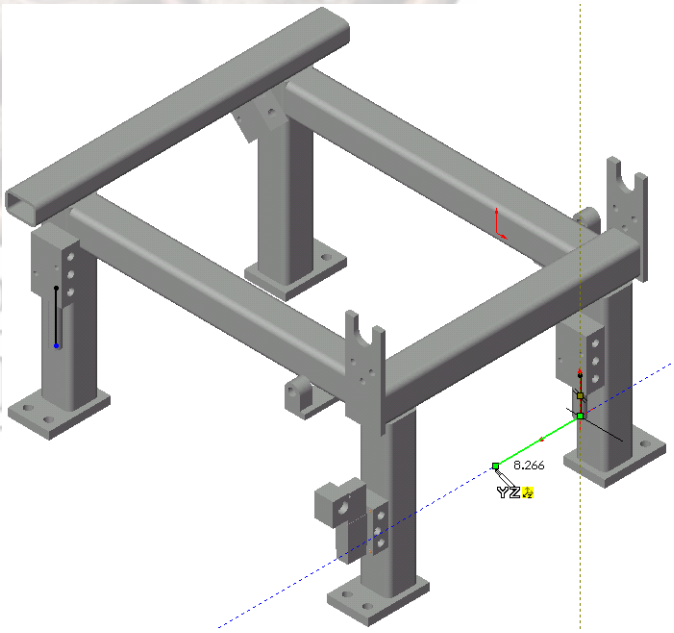
2 **New Route.**
Add a new route, using the default route name, starting at the connection point on the lower portion of the `manifold<1>` component. Drag the new route line down below the `Frame`.



3 **Add to the route.**
Right-click the CPoint on the `manifold<2>` and select **Add to route**. This starts another route line in the current route. Drag the line.

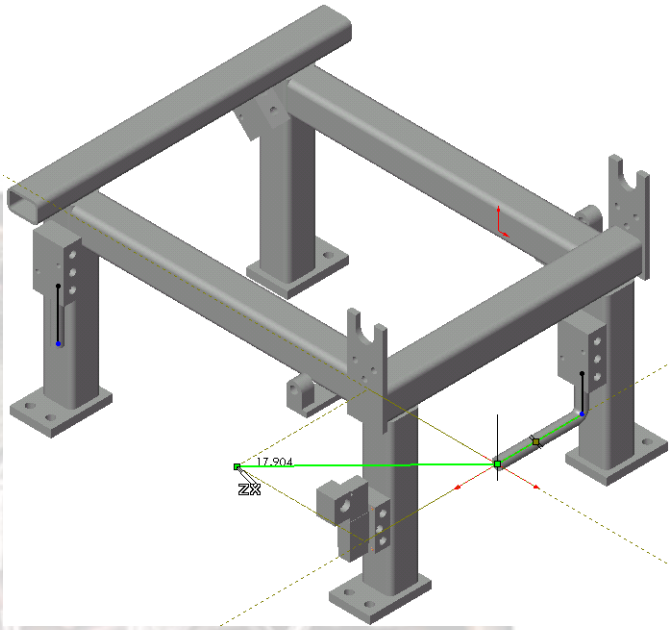
4 **Using the Space Handles.**
Continue the route by clicking the **Line** tool and starting a new line. Click the initial open endpoint and drag the other end of the line.

Press the **Tab** key to orient the **Space Handles**. When the axis orientation shown appears, stop clicking the **Tab** key and drag the line along the axis labelled (plane `YZ`). Locate the end of the line roughly as shown.



5 Sketch in plane.

Start the next line segment and press **Tab** to sketch an angled line in the ZX plane. Watch for light colored guides along the axes Z and X of the plane.

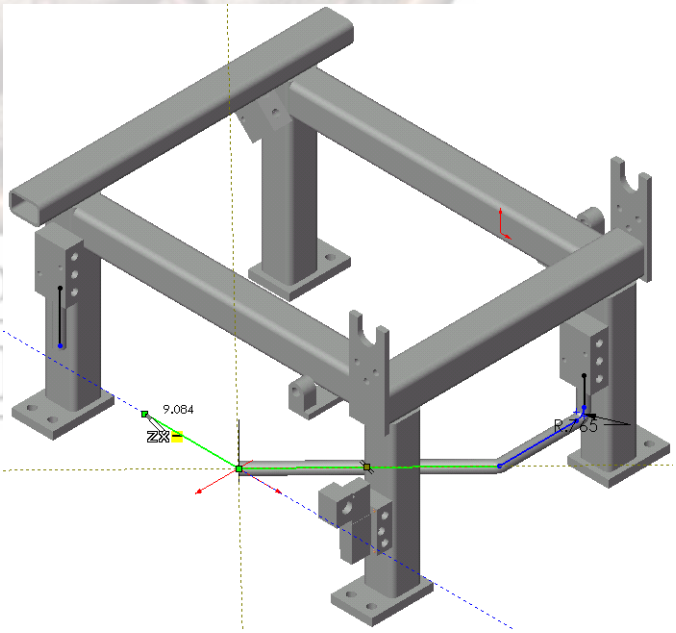


Tip

Due to the nature of a 2D graphical representation of 3D space, dragging 3D sketch entities after creation can give undesirable results, and is not recommended without first applying some constraints to the sketch.

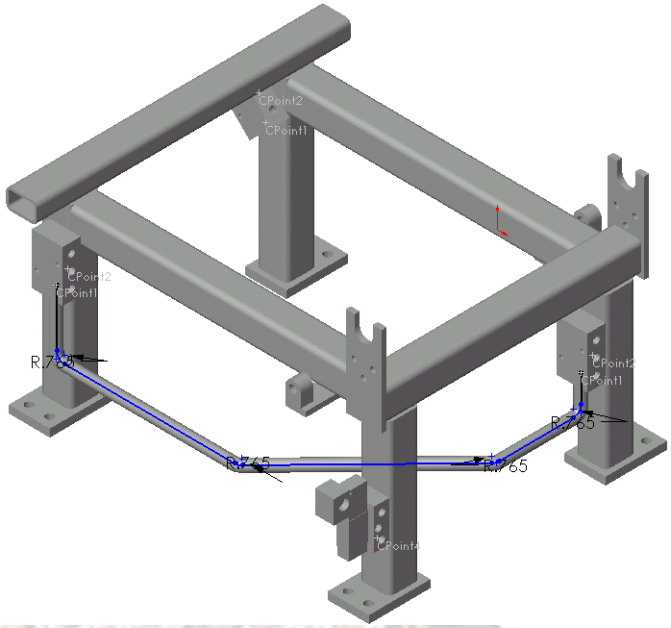
6 Sketch along X Axis.

Create the last line along the horizontal – axis (plane ZX) as shown.



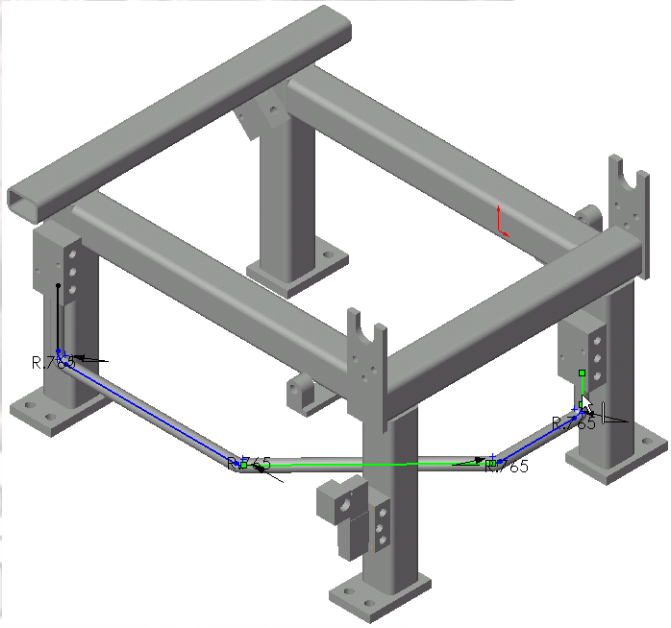
7 Merge Endpoints.

Select the open endpoints and add a **Merge** relation.

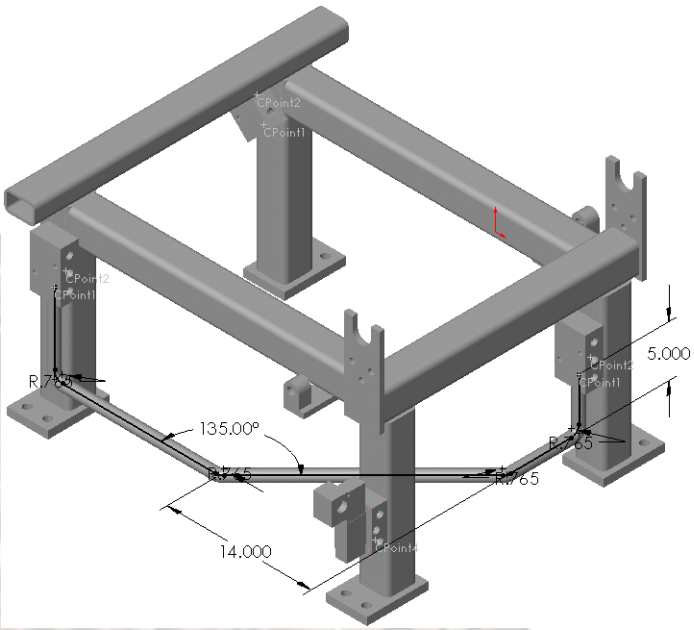


8 Perpendicular relation.

In order to keep the angled in the same plane as the adjacent lines, add a **Perpendicular** between the angled line and initial line.



9 Dimensions.
Add the angle and linear dimensions as shown. The angle dimension forces both corners to be 45° angles.



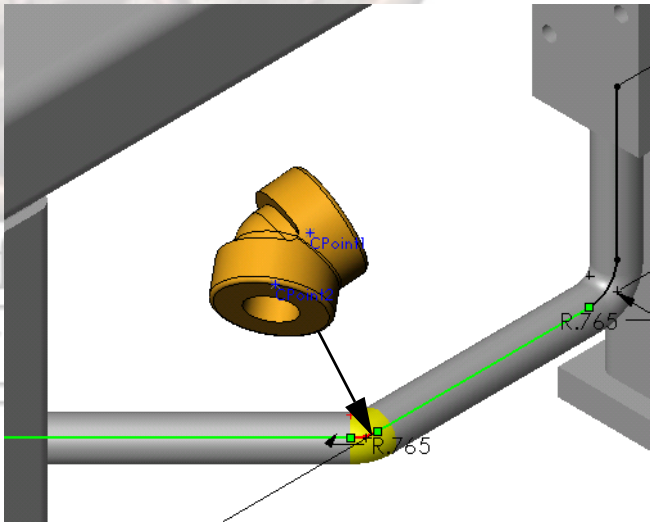
Alternate Elbows

All corners that are not equal to 90°, (equal to the default elbow), are flagged by the route as requiring a different type of elbow. The proper elbow must be selected for each bend.

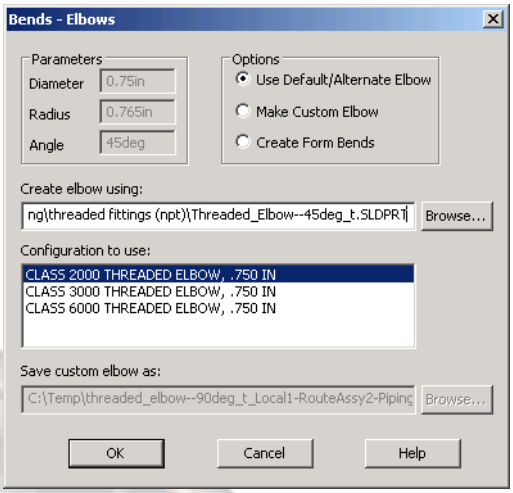
10 Exit the route.
Exit the route using **Exit Sketch** to access the **Bends - Elbow** dialog.

11 Choose Elbow.
Exit the sketch and the **Bends - Elbows** dialog appears. The first elbow in question is zoomed and highlighted.

The **Bends - Elbows** dialog appears.

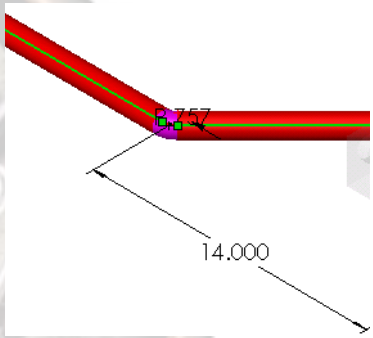


Use **Browse** to select the `threaded_elbow--45deg_t` component for use at this corner. Select the `CLASS 2000 THREADED ELBOW, .750IN` as the configuration to use and click **OK**.



Select the same replacement elbow for the next 45° corner using the same method.

Next save the pipe part file.



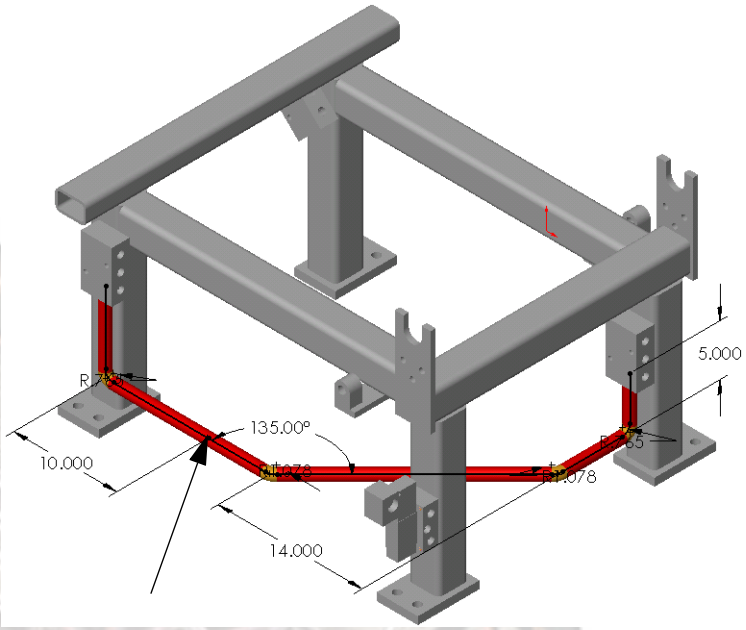
12 Completed route.
Edit the main assembly, closing the route sub-assembly. The pipes and associated elbows, both 90° and 45°, are added to the route `RouteAssy1-Piping Assembly`.

Editing a Route

Once created, the route and associated sub-assembly can be edited to change route sketch, add fittings or delete them.

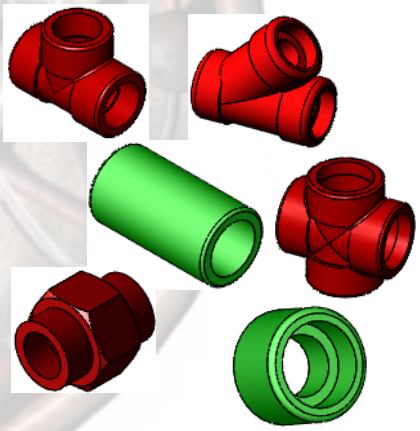
13 Split Entities.

Edit the route. Use **Split Entities** to break the line as shown. Add a dimension to locate the break.



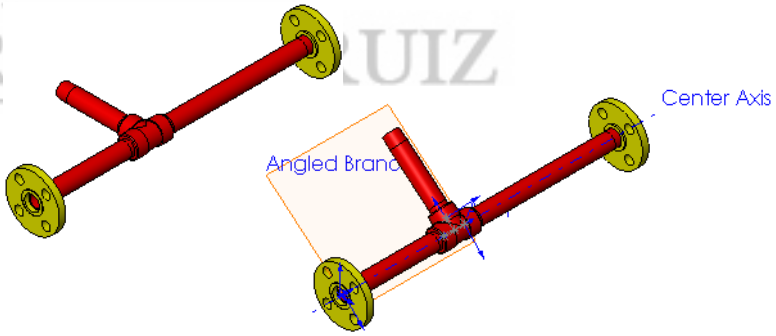
Adding Fittings

Pipe and elbow components are added to the route automatically based on the shape of the route's 3D Sketch. Other components like tees and eccentric reducers must be added manually at specific locations along the pipe route.



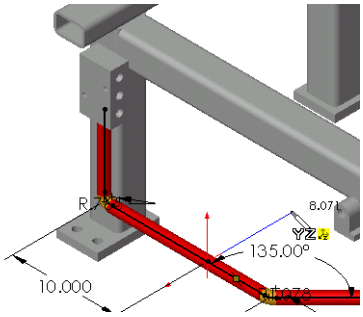
Fitting Orientation

The fitting orientation can be flipped using the **Tab** key provided an orientation line is not used. The orientation line is sketched from the end of the split line. An angled orientation line can be created using references planes and axes.



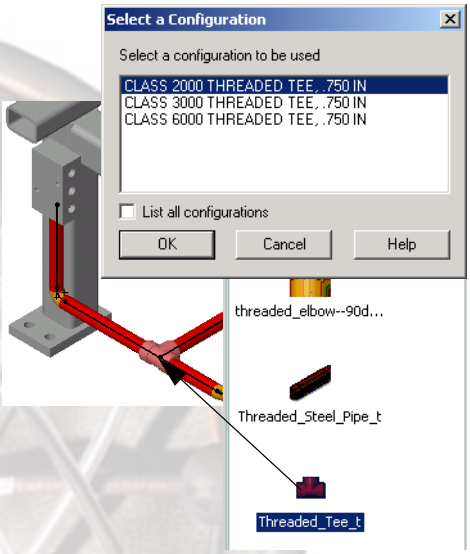
14 Orientation Line.

Add a line along the axis in the direction of the branch line as shown. This will specify the orientation of the tee for drag-drop from the Design Library.



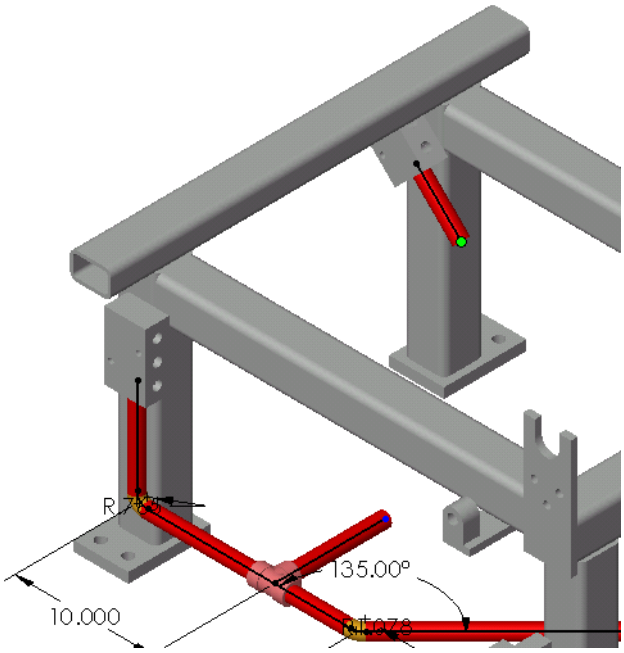
15 Add `threaded_tee_t`.

Drag and drop the fitting `threaded_tee_t` on the connection point, selecting the configuration as shown.



16 Add to route.

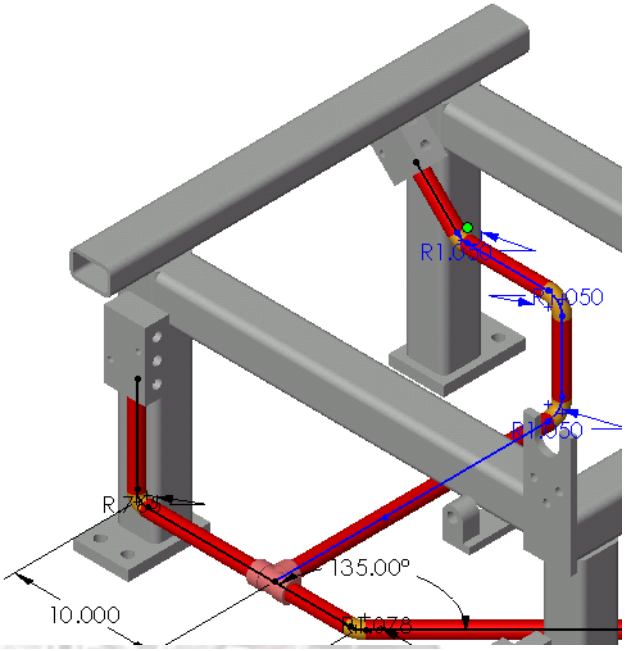
Add to the route from `CPoint2` of the `tee` and from `CPoint1` of `manifold<3>`.



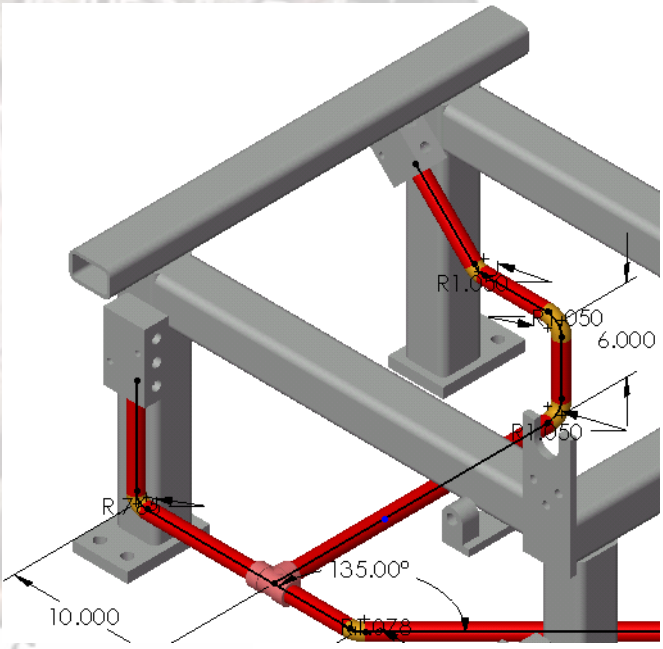
Tip

Do not use **Add to route** from the tee side of the line because it is already an active route line.

17 Auto route.
Use Auto route to complete the route. Choose the solution similar to the one shown at right.



18 Dimension.
Add a dimension (**6in**) to fully define the sketch.



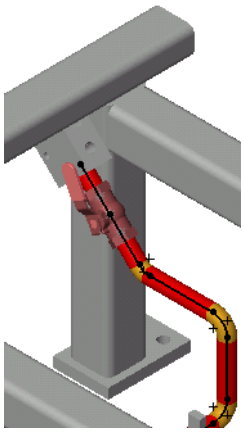
Using Assembly Fittings

Assembly based fittings include any fittings (valves and strainers for example) the are represented as multiple part components. They are added to the route using the same methods as part fittings.

19 Add Fitting.

Use **Split Entities** to break the angled line and add a **3in** dimension to the end line.

Right-click the point and select **Add Fitting** and select the assembly `Valve_t` (located in the `training design library\routing\assembly fittings\valves` folder) and list all configurations to select the 0.75in configuration.

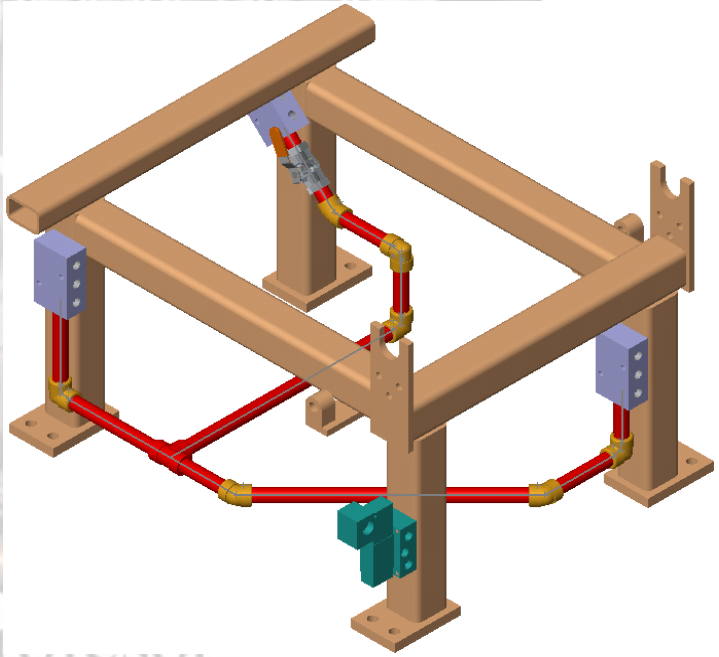


Tip

The fitting can also be dragged and dropped from the Design Library similar to the tee fitting.

20 Completed Route.

Exiting the sketch again calls the **Bends - Elbows** dialog to select another 45° elbow.



Remove Tube/Pipe

The **Remove Tube/Pipe** option is used to create a condition where bends or fittings contact each other directly. This option is available in **Edit Route** mode and is usually used with piping.

Where to Find It

- Right-click a pipe of the route and select **Remove Pipe** or **Remove Tube**.

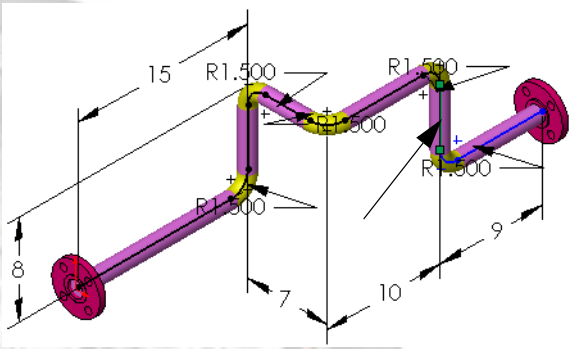
---

21 Open `Piping Options`.

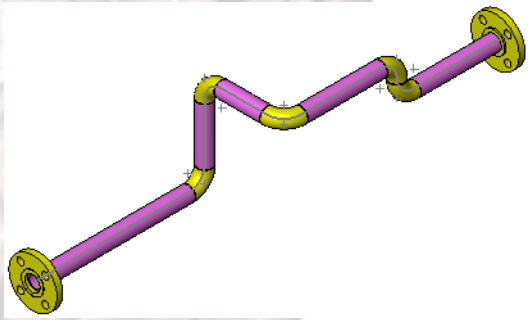
Open the assembly `Piping Options` and edit the route.

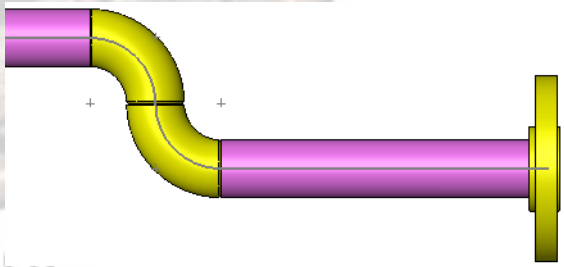
22 Select Line.

Select the line to be removed, indicated at right, and right-click **Remove Pipe**.



The line, and associated pipe, has been removed from the route.





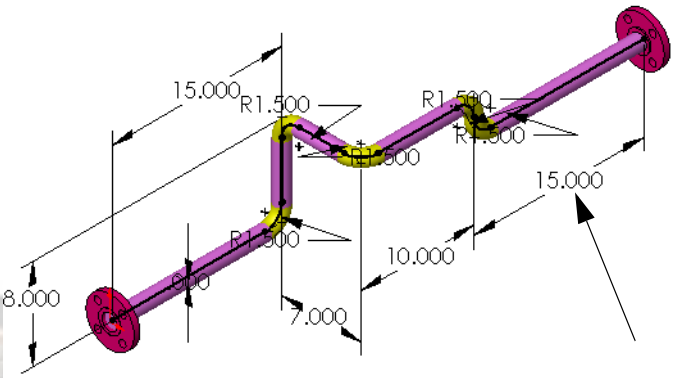
Custom Pipe/Tube Configurations

Custom Pipe/Tube Configurations can be created where pipe or tube lengths are equal but differ in geometry. This is useful when a penetration is used on one of two equal length pipes. This option is available in **Edit Sub-assembly** mode and is usually used with piping.

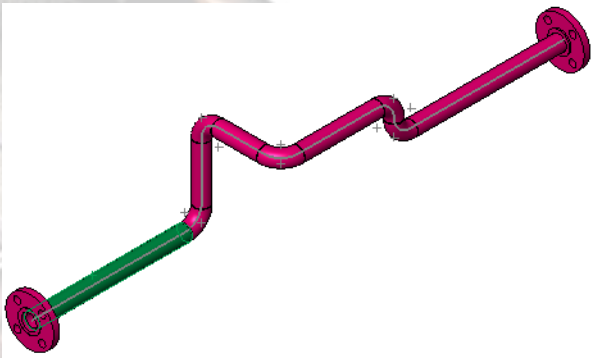
Where to Find It

Right-click a pipe of the route and select **Create Custom Pipe/Tube Configurations**.

23 Change Lengths.
Change the length of the last pipe in the route to match the first at **15in**.



24 Configuration.
Exit the sketch. Right-click the `1in Schedule40<1>` and choose **Create Custom Pipe/Tube Configuration**.



Pipe Penetrations

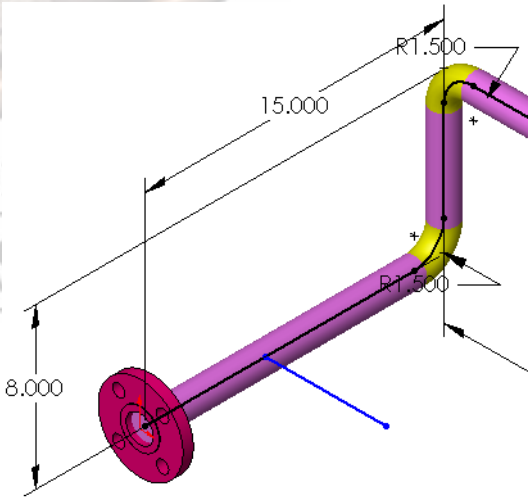
Pipe Penetrations can be created between intersection pipes. Both pipes are cut by the process. This option is available in **Edit Route** mode and is usually used with piping.

Where to Find It

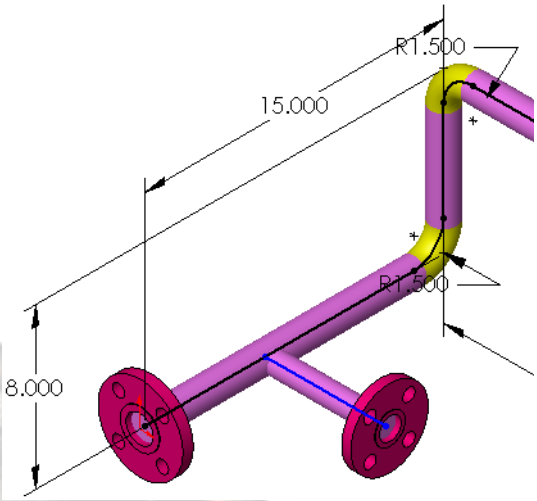
Right-click an endpoint of the connecting route and select **Penetrate**.

25 Add Line.
Edit the route sketch and add a line to the 3D Sketch as shown at right (blue). Use a **Coincident** relation between the endpoint and the route line.

Do *not* break the route line.



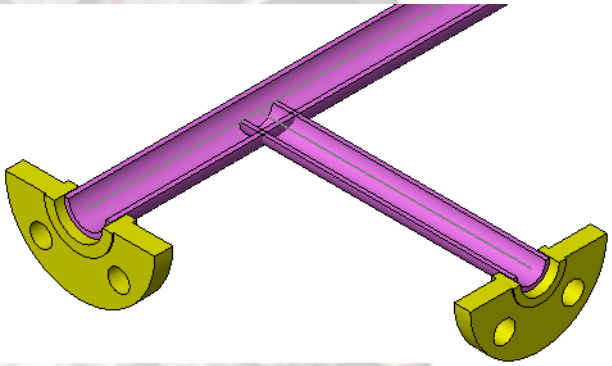
26 Flange.
Add a `Slip_On_Weld_Flange_t`, diameter `0.75`, to the open endpoint. The pipe is added along the line.



Note

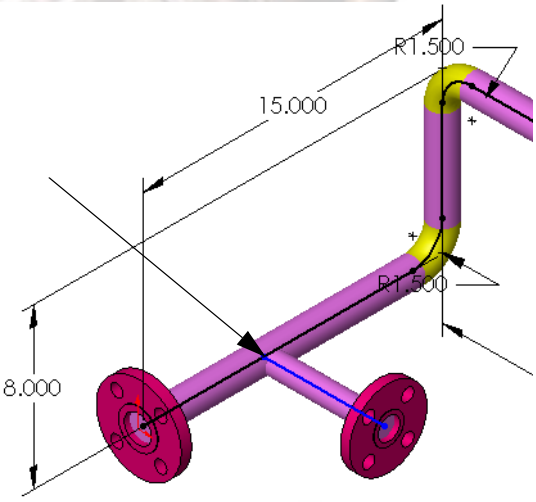
Use the pipe `pipe_t` and the elbow `LR_Inch_Elbow_t`.

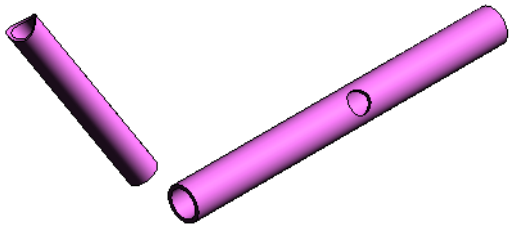
The pipes conflict.



27 Penetrate.
Right-click the junction point and select **Penetrate**.

Save the new pipe segment.





Lesson 9
Building Routing Library Parts

Upon successful completion of this lesson, you will be able to:

- Generate libraries of tubing and piping parts, including elbows and routing component assemblies, such as valves.

Fabricated Tube or Pipe Parts

Fabricated Tube or **Pipe Parts** can be created for use in a route. They are used along the length of the 3d Sketch created while routing. The specific naming of dimensions, sketches and features is important to ensure that the finished component is recognized as a routing component.

Note

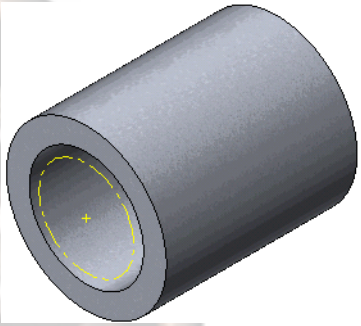
Design of electrical routing parts is covered in *Lesson 10: Electrical Routes*.

In this example, a PVC pipe will be created.

Note

The actual length of the pipe will be determined by the straight sections of the route.

The values used in this example are dimensions for a **1/2"** nominal diameter PVC pipe. Other sizes will be added through the design table.



Required naming information for the Tube/Pipe part:

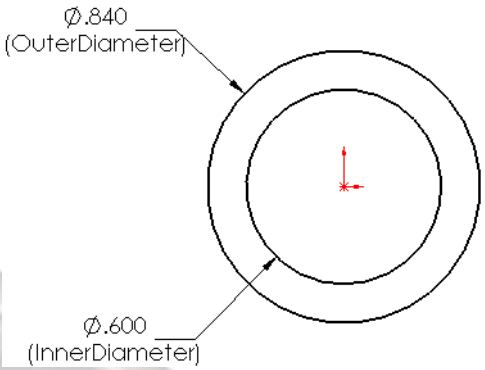
Base Feature Sketch	Name: `PipeSketch`	Dimensions: `InnerDiameter` and `OuterDiameter`
Base Feature:	Name: `Extrusion`	Dimension: `Length`
Sketch:	Name: `FilterSketch`	Dimension: `NominalDiameter`
Configuration Specific Properties	Name: `Pipe Identifier`	Value: <pipe description>

Note

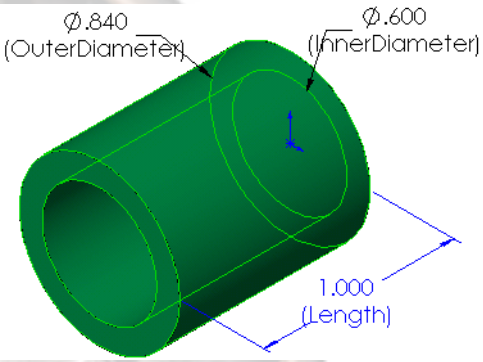
It is important to note that tubing and tubing components must use the term "`pipe`", not "`tube`", in these instances.

1 New Part.
Open a new inch part and edit the material to be **Plastics, PVC Rigid**.

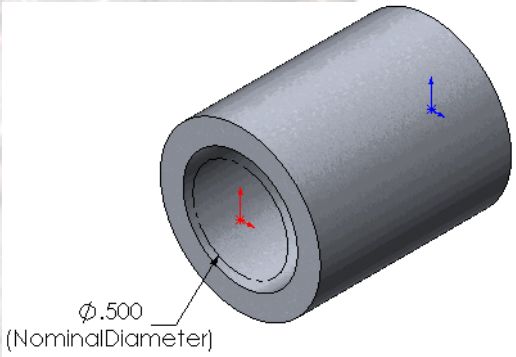
2 Sketch.
Create a new sketch on `Front Plane` (or `Plane1`), naming it `PipeSketch`. Create concentric circles as shown and name the dimensions `OuterDiameter` and `InnerDiameter` as shown.



3 Extrusion.
Create a boss feature named `Extrusion` using a depth of **1"**. Name the depth dimension `Length`.



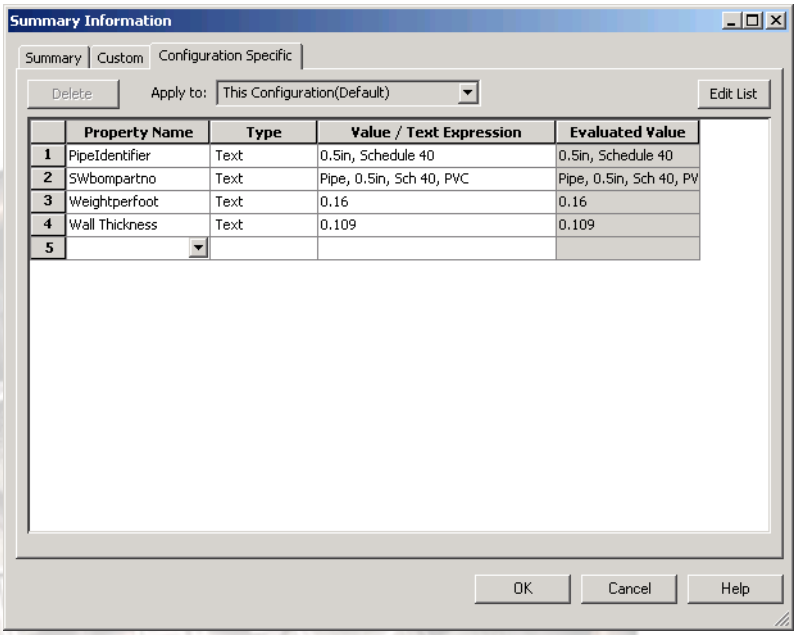
4 Nominal Diameter.
Add a new sketch on the front face, naming it `FilterSketch`. Add a construction circle and a dimension named `NominalDiameter`.



5 Configuration Specific Properties.

There are **Configuration Specific Properties** that may be assigned to the part through **File, Properties**. These properties will be transferred to the design table.

Add the following properties and values as shown below:



Note

In order to be recognized as a pipe component, only the `Pipe Identifier` property is required, but all descriptions should be carried with the library part.

Again, it is important to note that even a *Tube* part needs the property called `Pipe Identifier`.

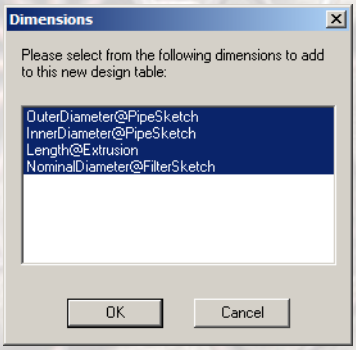
6 Auto-create Design Table.

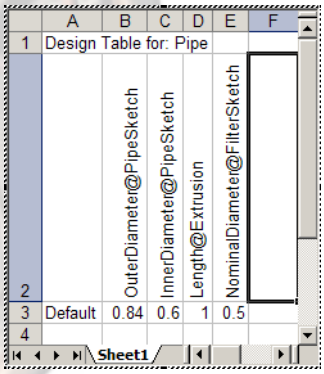
Using **Insert, Design Table...**, a design table will be automatically created in the part.

Use the settings shown for the new design table.

7 Add Dimensions.

Add the selected **Dimensions** to the design table using Shift-select and OK.



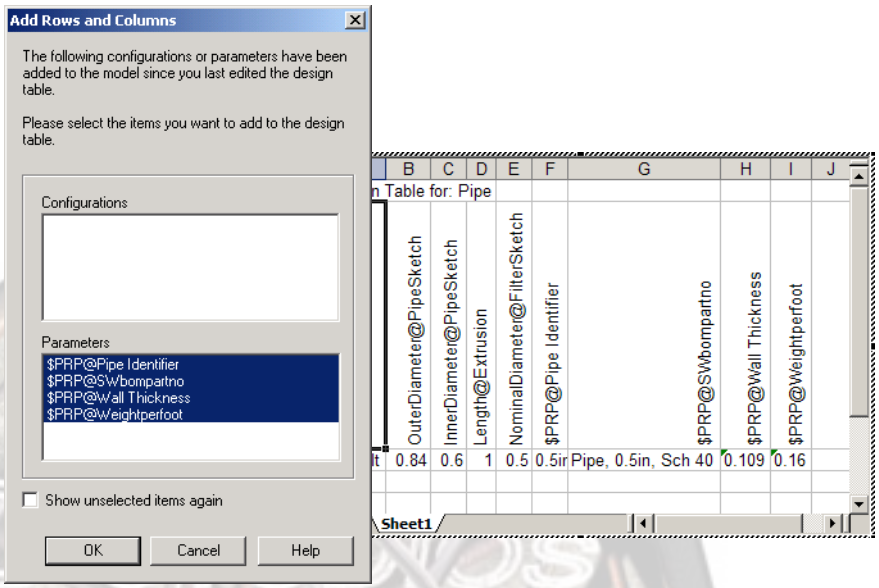


8 Close Design Table.

Click outside the design table to close it. The design table changes will be saved.

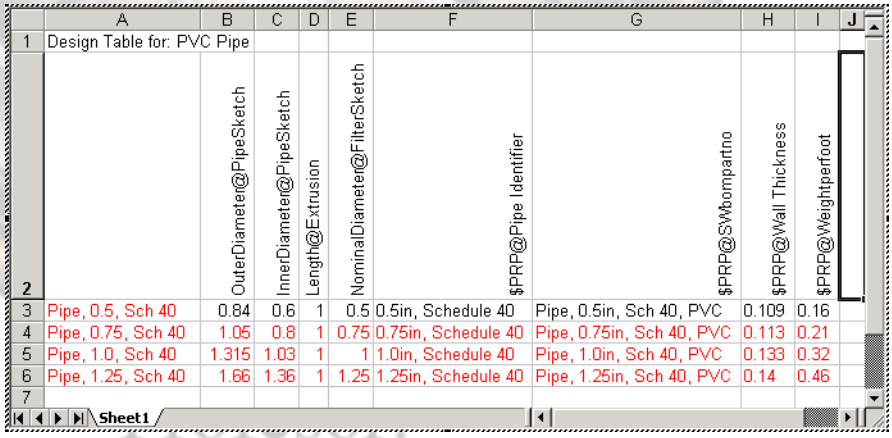
9 Edit Table.

Right-click the design table icon and **Edit Table**. Add the selected **Parameters** selected to the table.



10 Add new configurations.

While editing the table, replace the name `Default` on the first configuration and add three new ones as shown. Copy and paste where appropriate.



	A	B	C	D	E	F	G	H	I	J
1	Design Table for: PVC Pipe									
2		OuterDiameter@PipeSketch	InnerDiameter@PipeSketch	Length@Extrusion	NominalDiameter@FilterSketch	$PRP@Pipe Identifier	$PRP@SWbompartno	$PRP@Wall Thickness	$PRP@Weightperfoot	
3	Pipe, 0.5, Sch 40	0.84	0.6	1	0.5	0.5in, Schedule 40	Pipe, 0.5in, Sch 40, PVC	0.109	0.16	
4	Pipe, 0.75, Sch 40	1.05	0.8	1	0.75	0.75in, Schedule 40	Pipe, 0.75in, Sch 40, PVC	0.113	0.21	
5	Pipe, 1.0, Sch 40	1.315	1.03	1	1	1.0in, Schedule 40	Pipe, 1.0in, Sch 40, PVC	0.133	0.32	
6	Pipe, 1.25, Sch 40	1.66	1.36	1	1.25	1.25in, Schedule 40	Pipe, 1.25in, Sch 40, PVC	0.14	0.46	
7										

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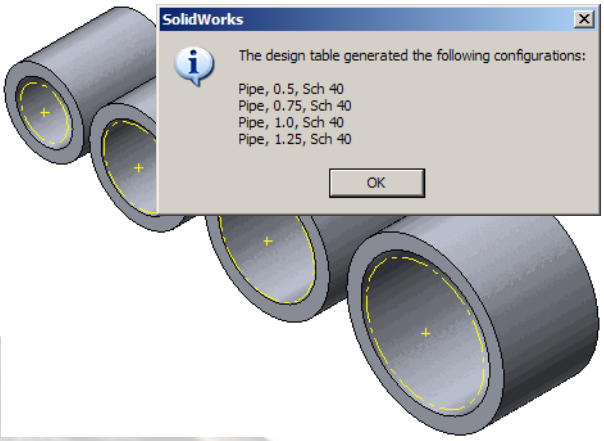
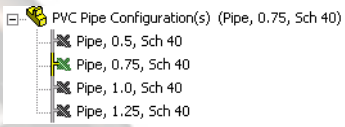
11 Configurations.
The configurations are added, successful ones being listed in the message box.



12 Remove `Default`.
The `Default` configuration is no longer needed and can be deleted from the ConfigurationManager.



13 Add to Design Library.
Add the part to the `training design library` folder by drag and drop. Another method is to add the folder where the file is located to the Design Library setting in **Options**, **File Locations**.

Tip

When adding parts to the Palette, it is best to create a new folder for a set of similar components.

Elbow Parts

Elbow Parts can be created for use in a route where the line changes direction. Generally, elbows are created for the conditions where pipes meet at right (90 degree) and 45 degree angles. Other angles are usually covered by using **Custom Elbows**, automatically converted from the standard ones.

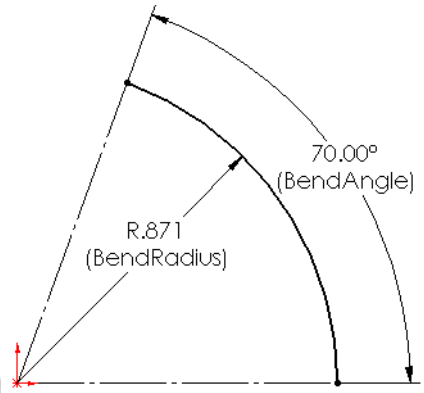
Like pipes, the specific naming of dimensions, sketches and features is important to ensure that the finished component is recognized as a routing component. Note that tube routes *do not* use elbows.

In this example, a PVC elbow will be created.

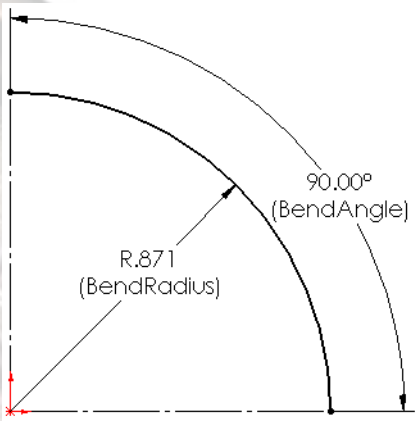
1 Elbow sketch.

Open a new part file with inch units.

Create a new sketch on `Front Plane`, naming it `ElbowArc`. Add construction lines and a centerpoint arc as shown. Fully define the sketch and name the dimensions `BendAngle` and `BendRadius`.

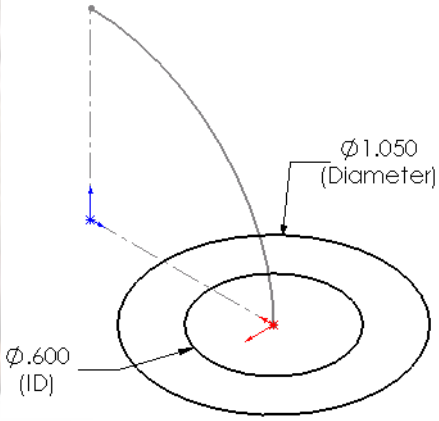


Set the angle to **90** degrees before closing the sketch.

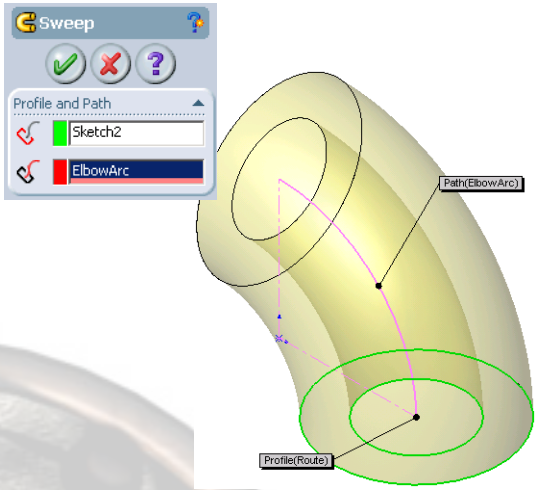


2 `Route` sketch.

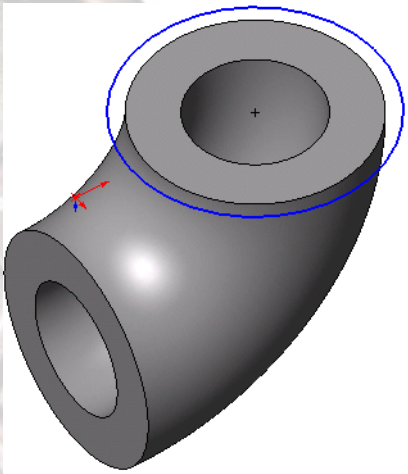
Add a new plane using **Normal to curve** at the end of the arc. Create a new sketch named `Route` for the cross section of the elbow. Add the `Diameter` and `ID` dimensions as shown.



3 Sweep.
Sweep the **Profile** `Route` along the **Path** `ElbowArc`.



4 Hub.
Create a new feature named `HubA` that includes a concentric circle of diameter **1.197in** that is extruded to a height (depth) of **0.247in**.

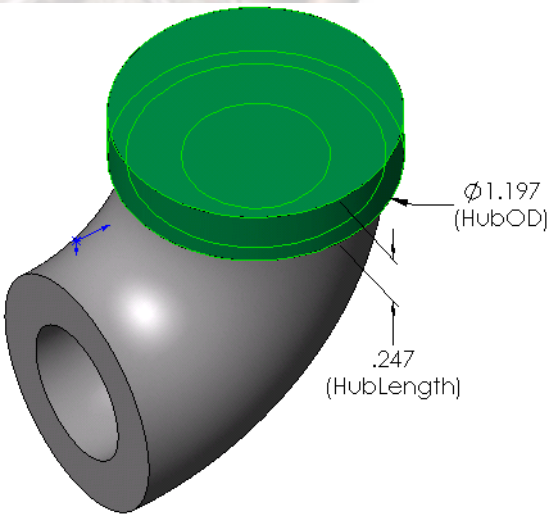


5 Link Values.
Use **Link Values** to add names to the extrusion diameter and depth values as shown:

Diameter = `HubOD`

Depth = `HubLength`

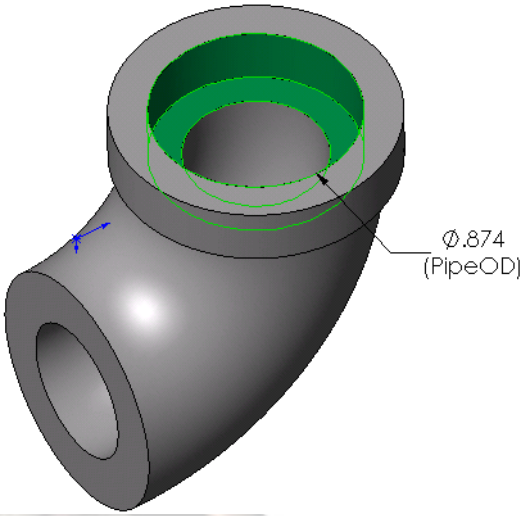
Adding a link value changes the name of the dimension. These will be linked to the dimensions of the other flange.



6 Cut hub.

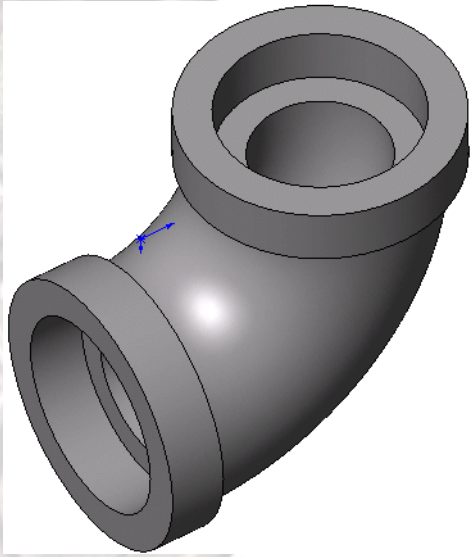
Create a circle, and add the **Link Value** `PipeOD` to the diameter dimension.

Cut with **Up To Surface**, using the rear face of the `HubA`.



7 Copy and Paste.

Copy and paste the boss and cut created above, repairing any errors. The link values will be included with the geometry.

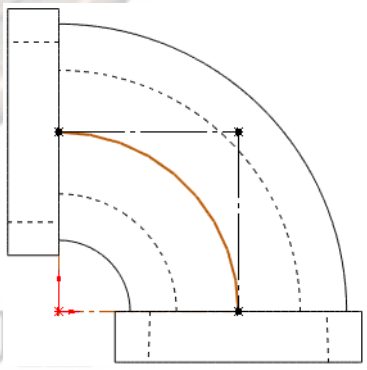


8 Points.

On the same plane that you created `ElbowArc` (`Front Plane`), create a new sketch named `ConnectPoints`. **Show** the `ElbowArc` sketch and add points as shown.

The third point is added at the theoretical intersection of the lines meeting at the elbow.

Hide the `ElbowArc` sketch again.



Tip Add the points first, then connect them with vertical and horizontal construction lines to fully define the location of the third point.

Routing Points

Routing points, which include Connection Points (**CPoints**) and Route Points (**RPoints**), are used to determine several details of the route.

- **CPoints** determine where the routing ends, the direction that it enters or exits the fitting, and the nominal diameter of the pipe, tube, or cable.
- **RPoints** are used for placement of fittings on a route line sketch endpoint.

Tip CPoints and RPoints can be renamed to suit your design.

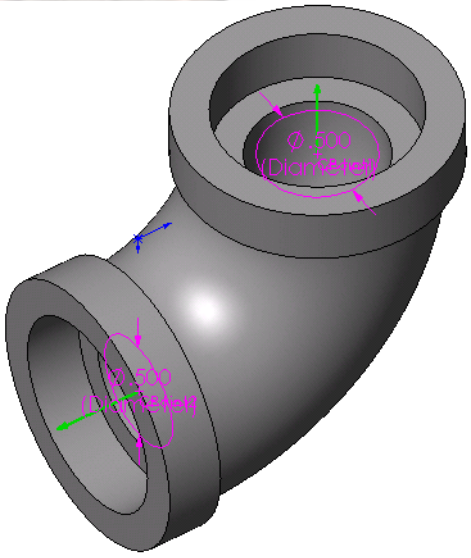
9 Add Connection Points.
Add a **CPoint** by selecting the face and sketch point.

Set Route type to **Fabricated Pipe**.

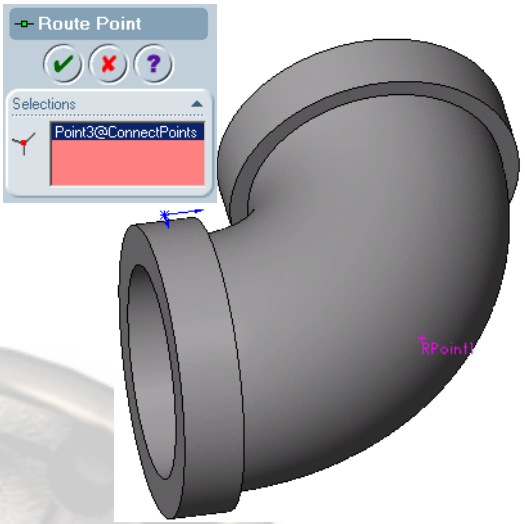
Set the **Nominal diameter** to 0.5”.

Add another CPoint at the lower connection point. Double-clicking the feature displays the nominal diameter at the connection point.

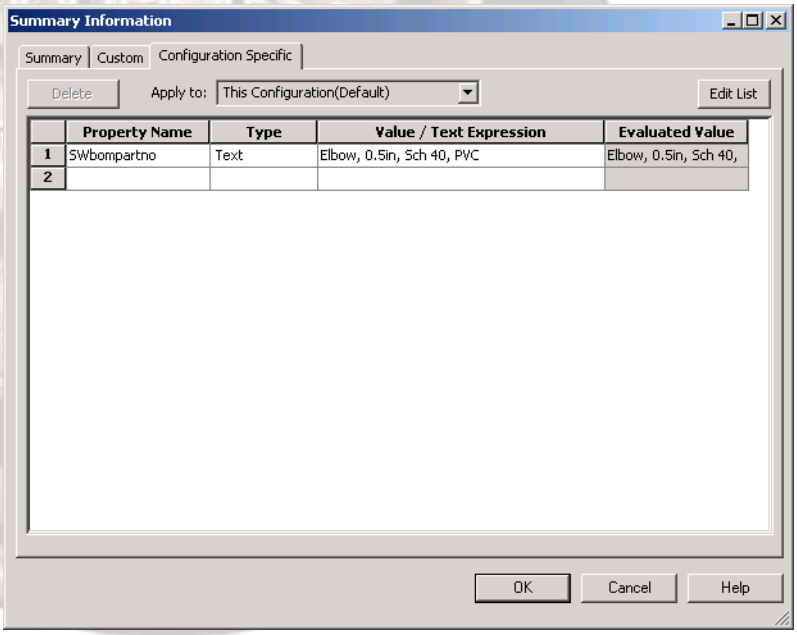
Click **View, Routing Points** to see the CPoint and Rpoint symbols.



10 Route Point.
Add a **Route Point** at the third sketch point location.

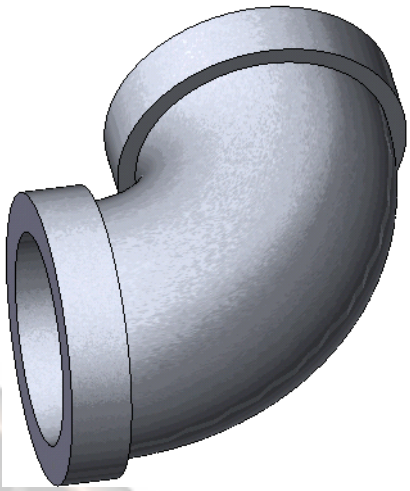


11 Configuration Specific Properties.
Add the property as shown.



12 Material.

Edit the material to **PVC rigid**.



13 Design Table.

Create a **Design Table** and add data as shown.

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: PVC Elbow-90											
2		BendAngle@ElbowArc	BendRadius@ElbowArc	Diameter@Route	ID@Route	HubOD@Sketch3	HubLength@HubA	PipeOD@Sketch4	Diameter@CPoint1	Diameter@CPoint2	$PRP@SWbompartno	
3	0.5in, Sch 40, PVC	90	0.871	1.05	0.6	1.197	0.247	0.874	0.5	0.5	Elbow, 0.5in, Sch 40, PVC	
4	0.75in, Sch 40, PVC	90	1.037	1.29	0.8	1.458	0.275	1.05	0.75	0.75	Elbow, 0.75in, Sch 40, PVC	
5	1.0in, Sch 40, PVC	90	1.198	1.583	1.03	1.771	0.3	1.315	1	1	Elbow, 1.0in, Sch 40, PVC	
6	1.25in, Sch 40, PVC	90	1.409	1.95	1.36	2.135	0.3125	1.66	1.25	1.25	Elbow, 1.25in, Sch 40, PVC	
7												

Sheet1

14 Add the part to the Design Library.


PVC_Elbow


PVC_Pipe


routing

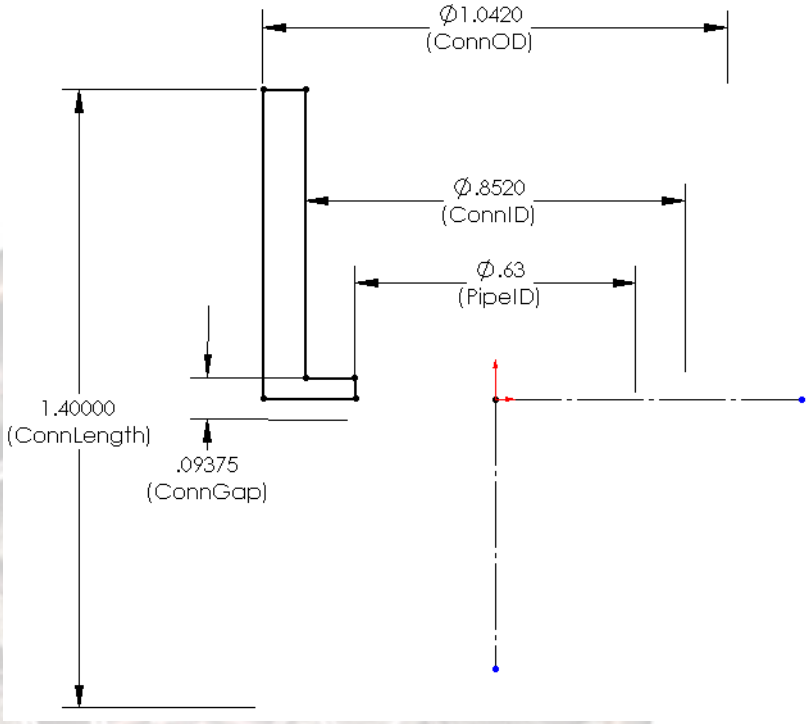
Fitting Parts

Fitting Parts are more generic and can be created for use in a route. They are anything that does not fit into the standard categories of pipe/tube, elbow, flange or reducer.

In this example, a PVC coupling will be created.

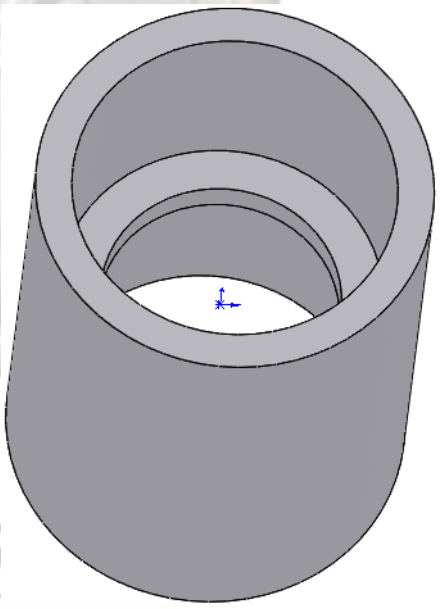
1 New part and Sketch.

Create a new part file with units inches and sketch using centerlines for symmetry. Use the dimension names as shown.



2 Revolve and Mirror.

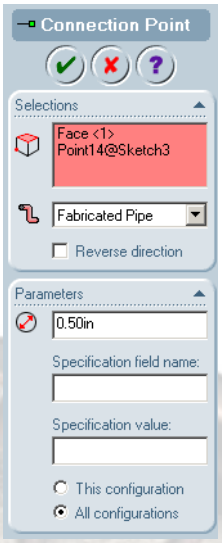
Create a revolved feature, creating the upper half of the part. use Mirror to complete the part.

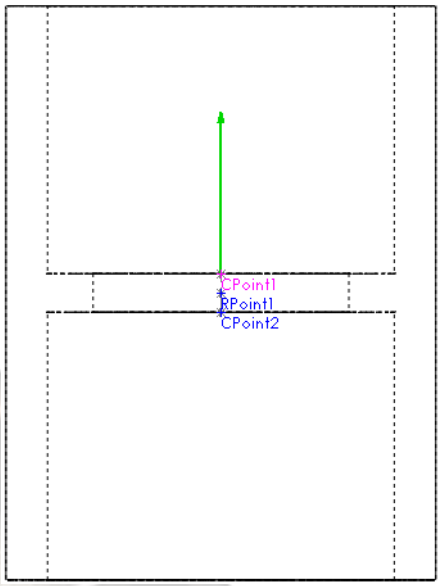


3 Route Points.
Create points in a sketch and add CPoints and an RPoint as shown.

Set type to **Fabricated Pipe**.

Set the **Nominal diameter** to 0.5 for each connection point.

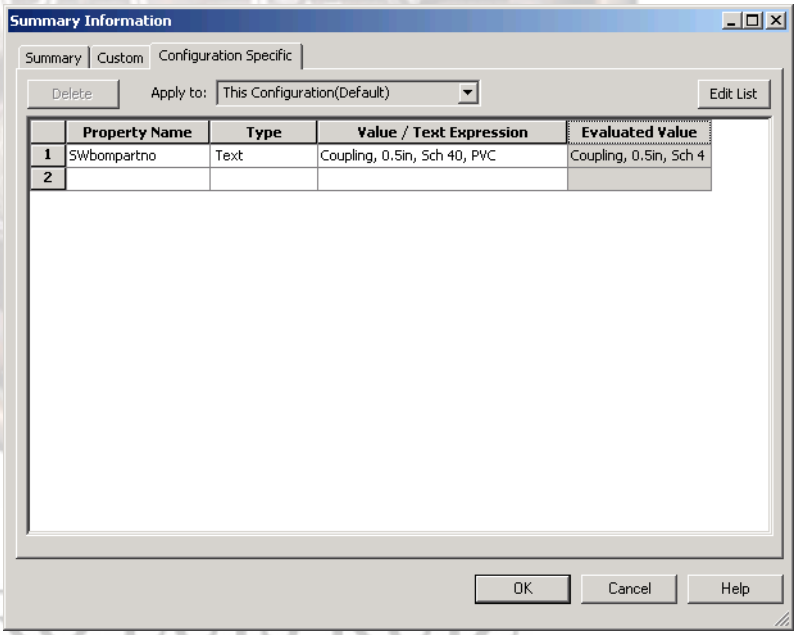




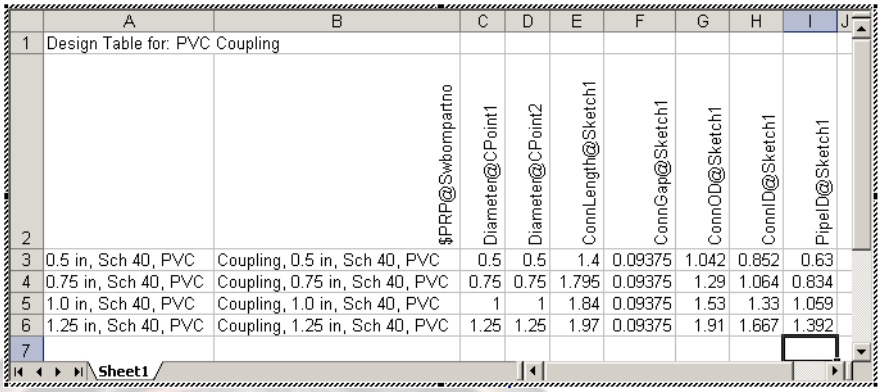
Note

If this were a tubing connector, the CPoints would be designated as type **Tubing**. For electrical, the designation would be **Electrical**.

4 Add properties.
Add the configuration specific properties shown below.



5 Populate the Design Table.



	A	B	C	D	E	F	G	H	I	J
1	Design Table for: PVC Coupling									
2		$PRP@Swbompartno	Diameter@CPoint1	Diameter@CPoint2	ConnLength@Sketch1	ConnGap@Sketch1	ConnOD@Sketch1	ConnID@Sketch1	PipeID@Sketch1	
3	0.5 in, Sch 40, PVC	Coupling, 0.5 in, Sch 40, PVC	0.5	0.5	1.4	0.09375	1.042	0.852	0.63	
4	0.75 in, Sch 40, PVC	Coupling, 0.75 in, Sch 40, PVC	0.75	0.75	1.795	0.09375	1.29	1.064	0.834	
5	1.0 in, Sch 40, PVC	Coupling, 1.0 in, Sch 40, PVC	1	1	1.84	0.09375	1.53	1.33	1.059	
6	1.25 in, Sch 40, PVC	Coupling, 1.25 in, Sch 40, PVC	1.25	1.25	1.97	0.09375	1.91	1.667	1.392	
7										

Sheet1

Note

You may see other parameters entered into the table by default. These can be added, removed or customized:

- `D1@Revolve1`
  This is the revolve angle of the base feature (360 degrees) and does not need to be configured.
- `MinStraightLength@CPoint1`
  This is the minimum length of pipe/tube used to start a route, in order to avoid excessive strain (default varies with configuration and size).
- `EndLengthAdjustment@CPoint1`
  This is an additional length that can be added to the pipe/tube to allow for end treatments such as swaging (default is zero).

Flange Parts

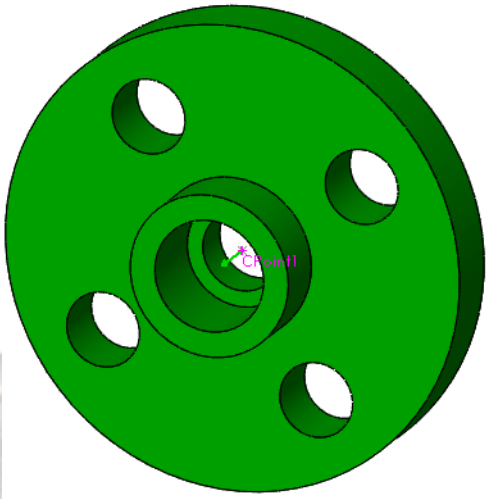
Flange Parts are used at the start or end of the route and are generally used for connections to equipment. Flange parts contain only one routing point, a CPoint.

In this example, a PVC flange will be created from an existing part.

1 Open `Flange.sldprt`.
Open the existing part `Flange`.

2 Connection Point.
Add a point and **Fabricated Pipe** CPoint inside the bore, diameter **0.5in**.

For a tubing flange, you would add a **Tubing** type CPoint.



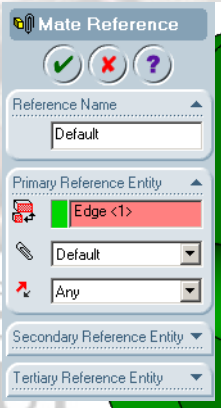
3 Set properties.
Add the configuration specific property `SWbompartno Flange, 0.5in, Sch 40, PVC` to the default configuration.

4 Design Table.
Insert a design table, as shown below.

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Flange											
2		$PRP@SWbompartno	BaseOD@Sketch1	BaseTh@Extrude1	ConnOD@Sketch3	ConnHgt@Extrude2	BoltPattDiam@Sketch4	HoleDiam@Sketch4	PipeOD@Sketch6	PipeID@Sketch7	Diameter@Cpoint1	
3	0.5in, Sch 40, PVC	Flange, 0.5in, Sch 40, PVC	3.5	0.5	1.22	0.97	2.375	0.56	0.84	0.6	0.5	
4	0.75in, Sch 40, PVC	Flange, 0.75in, Sch 40, PVC	3.875	0.53	1.44	1.06	2.75	0.56	1.05	0.8	0.75	
5	1.0in, Sch 40, PVC	Flange, 1.0in, Sch 40, PVC	4.25	0.66	1.75	1.19	3.125	0.59	1.315	1.03	1	
6	1.25in, Sch 40, PVC	Flange, 1.25in, Sch 40, PVC	4.625	0.66	2.1	1.69	3.5	0.59	1.66	1.36	1.25	
7												

Sheet1

5 Mate Reference.
Add a **Mate Reference** on the rear edge of the part as shown.



6 Save.
Save as `PVC Flange` and add it to the Design Library.

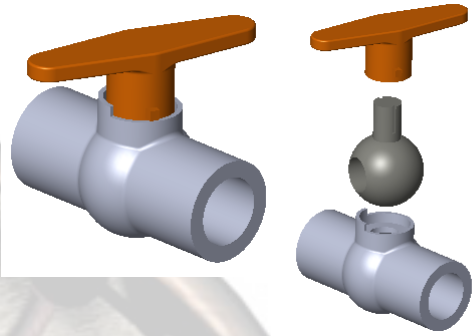
Assembly Fittings

Routing "part" fittings that are actually assemblies can be created for use through the Assembly Palette. These assemblies must contain routing points at the assembly level based on routing points in the component parts.

---

1 Open valve assembly.

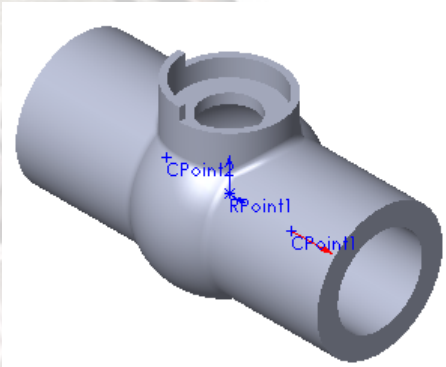
Open the `PVC Ball Valve` assembly. It consists of three components: `PVC Valve Body`, `PVC Valve Stem`, and `PVC Valve Handle`.



2 Routing Parts.

One part in the assembly is a routing part. The `PVC Valve Body` contains the required Routing Points: **Fabricated pipe CPoints** and **RPoint**.

The remaining parts are components in the assembly but contain no routing information.



Note

Two parts in the assembly contain multiple configurations: the `PVC Valve Body` and `PVC Valve Stem`. These component contain compatible configurations for both a 0.75" route and a 0.50" route.

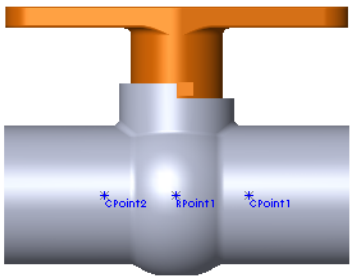
Assembly-Level Routing Points

Using the routing points embedded in the component parts, you can add assembly-level routing points. **ACPoints** are used to locate the ends of the route line, in this example where the pipe part is cut by the assembly component. **ARPoints** are used to locate the assembly on the route sketch.

---

3 View points.

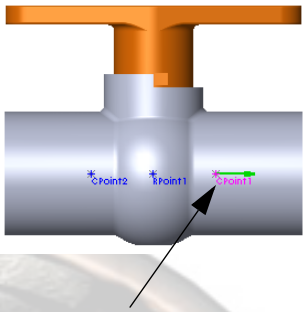
Click **View, Routing Points** to see the embedded CPoints and RPoint.

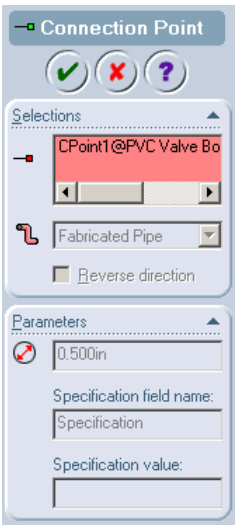


4 Assembly Connection Point.

Click the **Connection Point** icon and, on the model, select `CPoint1` of `PVC Valve Body<1>` and click **OK**.

The feature `ACPoint1` is added to the assembly using the settings taken from `CPoint1`.

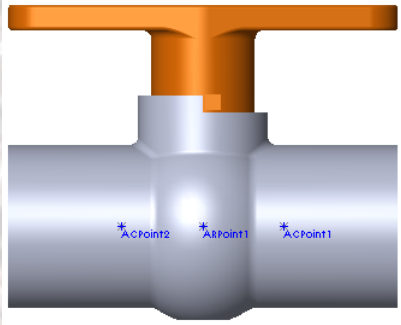




Repeat the procedure for the other CPoint of the `PVC Valve Body`, this time selecting the CPoint from the FeatureManager.

5 Assembly Route Point.

Add an assembly-based route point using the route point of the `Valve Body` component. It appears as the feature `ARPoint1`.



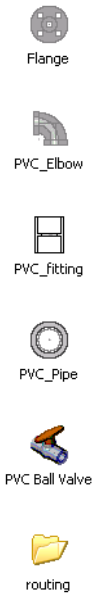
6 Configurations.

If the assembly is being used for multiple route diameters, multiple configurations must be used. The proper part configurations for each assembly configuration should be selected through a design table.

	A	B	C	D
1	Design Table for: PVC Ball Valve			
2		$CONFIGURATION@PVC Valve Body<1>	$CONFIGURATION@PVC Valve Stem<1>	
3	Ball Valve, 0.5in, Sch40, PVC	Valve Body, PVC, 0.5in, Sch40	Valve Stem, PVC, 0.5in, Sch40	
4	Ball Valve, 0.75in, Sch40, PVC	Valve Body, PVC, 0.75in, Sch40	Valve Stem, PVC, 0.75in, Sch40	
5				

Sheet1

7 Add to Design Library.
Drag and drop the top level assembly icon into the Design Library.



Note

Saving an assembly this way does not save the reference parts in the same location. To be safe, the part files should be copied to the Palette Assembly folder using **File, Find References...**, or Windows Explorer. In this way, should the original part files be moved or deleted, the assembly will find the parts in the local folder.

If not saved onto the Palette, the assembly can still be added easily to a route assembly using **Add Fitting**.

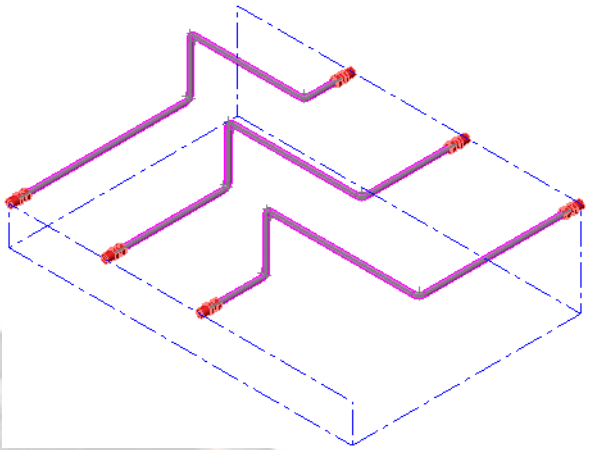
Exercise 1: Auto routing with Tubes

Create multiple route assembles using `tubing` library parts.

This lab uses the following skills:

- Setting route options and route properties.
- Auto routing.

Units: **inches**



Setup

Setup SolidWorks Routing to use a different library, in this example the included `tubing` library.

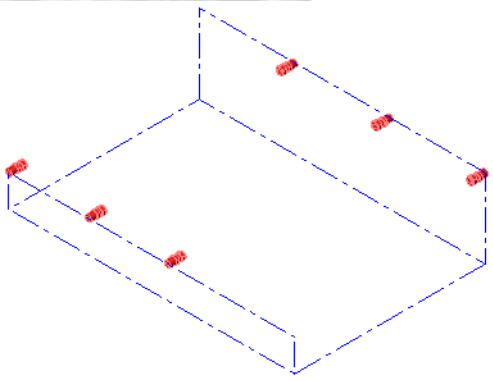
1 **Routing options.**
Click **Tools, Options, System Options, Routing** and set the **Library folder** to `training design library\routing\tubing`.

2 **Design library settings.**
Click **Tools, Options, System Options, File Locations** and add the `training design library` folder.

Creating the Routes

The route is added to an existing assembly that contains some components to start and stop it.

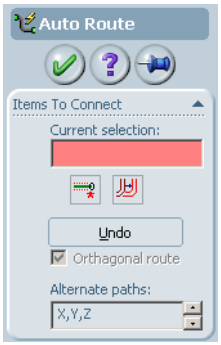
3 **Open `Tubing Lab`.**
Open the existing assembly `Tubing Lab`. It contains sketches and 6 `solidworks-lok_male_connector_t` fittings.



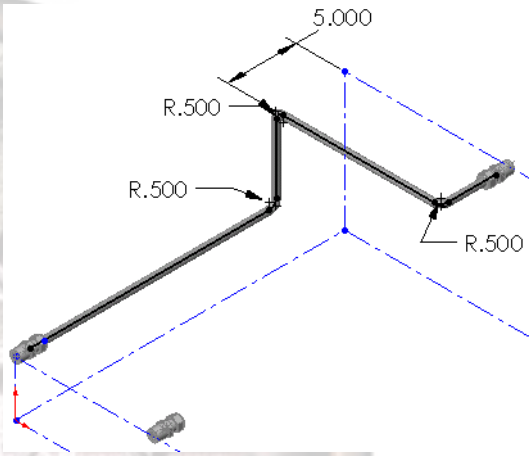
4 **Routing Points.**
Click **View, Routing Points** to display the routing points.

5 **Initial tube route.**
Create `Tube Route A` using a **0.5in** `tube-ss_t` tube with radius **0.5in**.

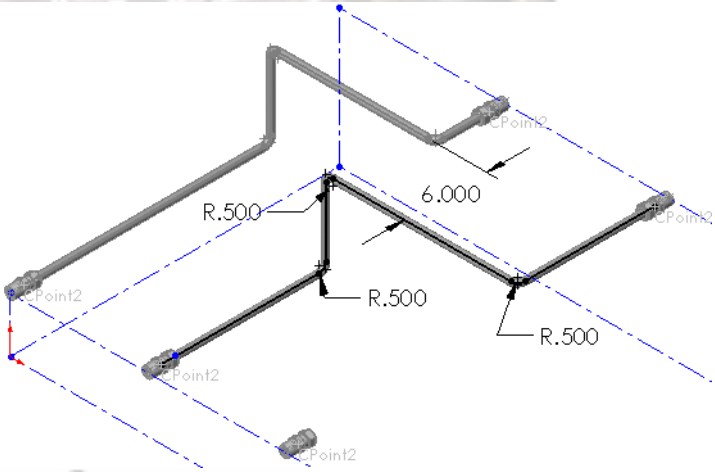
Use an **Orthogonal Auto route** to create the route. Choose a solution similar to the one shown at right.



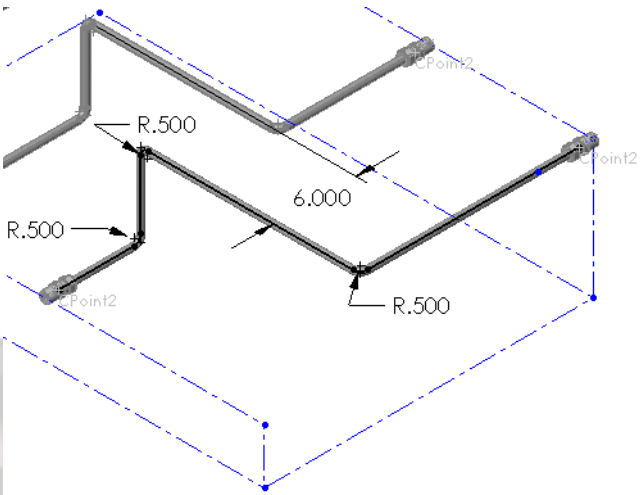
Add a dimension to fully define the sketch.



6 **Second tube route.**
Add a second route, `Tube Route B`, to the assembly. Use the same settings and dimension from the first route.



7 **Final tube route.** Add a third route, `Tube Route C`, to the assembly. Use the same settings and dimension from the second route.



8 **Save and close.**

Exercise 2: Multiple Piping Routes

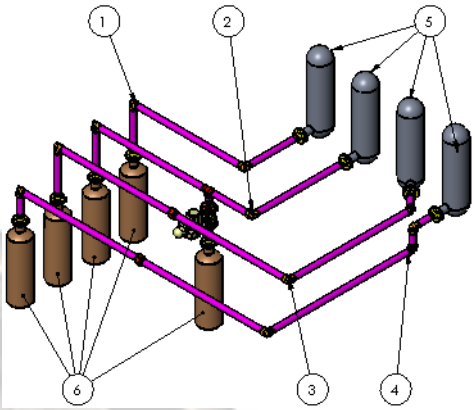
Add to this assembly to create multiple piping routes and associated pipe fittings.

This lab uses the following skills:

- Auto routing for piping.
- Creating multiple piping routes.
- Add branch lines and tee fittings.
- Add inline fittings like valves and unions.
- Create piping drawings.

Units: **inches**

ITEM NO.	PART NUMBER	QTY.
1	Route #1	1
2	Route #2	1
3	Route #3	1
4	Route #4	1
5	Feeder Tank	4
6	Holding Tank	5



Procedure

Open an existing assembly.

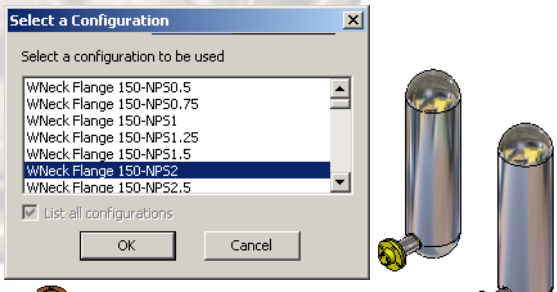
1 Settings.
Under **Tools, Options, System Options, Routing** use the Library folder `\\training design library\routing\piping`.

2 Open `Multiple Routes`.
Open the `Multiple Routes` assembly. It contains tank components placed to start and end piping routes.

Adding the first route

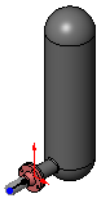
Add the first piping route.

3 Add Flange.
Drag and drop a `welding neck flange` fitting onto the flange of `Feeder Tank <1>`. Use the configuration `WNeck Flange 150-NPS2` for the 2” version.



4 Route name.
Name the route `Route# 1` and use the default route assembly template.

5 Route properties.



Use the following settings for the route properties. The source library files are in `piping/threaded fittings (npt).`

- **Pipe**

  Select `threaded steel pipe` with the `2in Sch 40` configuration.

- **Elbow**

  Click **Always use elbow** and select `threaded elbow--90deg` with the `CLASS 2000 THREADED ELBOW, 2.00 IN` configuration.

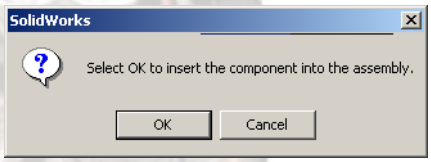
6 Pipe name.

Exit the route sketch using right-click **Exit Sketch**. When saving the pipe configuration, use the default naming convention `2in Schedule40-Route# 1`.

Ending Route Line

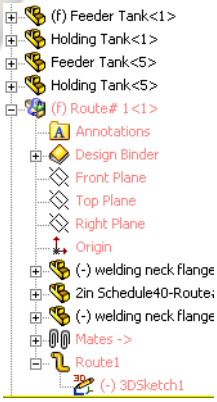
Creating multiple route lines in the same route assembly can be used to represent the start and end of a route. Having the start and end lines makes it possible to use the **Auto Route** function.

7 Add fitting to existing route.



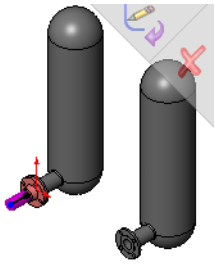
Drag and drop another `welding neck flange` onto the flange of the `Holding Tank<1>` component. At the **Start Route** dialog, click **Cancel**. Click **OK** to insert the component into the assembly.

8 Edit Route.



Right-click on the `Route# 1` sub-assembly or the `Route1` feature in the active sub-assembly and choose **Edit Route**. This action puts edits the 3D Sketch of the route feature.

9 Add to route.
Right-click CPoint of the new flange and choose **Add To Route**. A route line is added from the Cpoint.



Using Auto Route

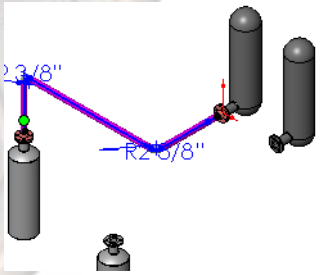
The **Auto Route** function can be used to connect open route endpoints with a series of automatically generated route lines and fillets. Generally several solutions are offered.

Tip

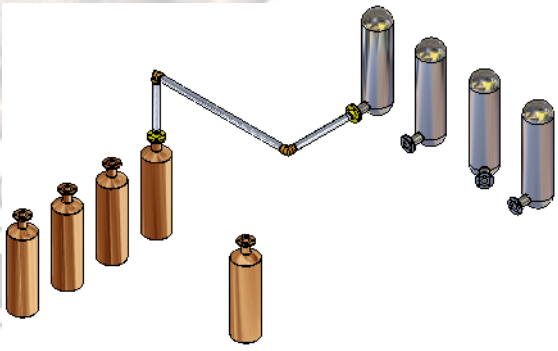
Pipe routes are limited to **Orthogonal routes** only.

10 Auto Route.
Select either of the open endpoints and right-click **Auto Route**. Select the other endpoint to see a solution route.

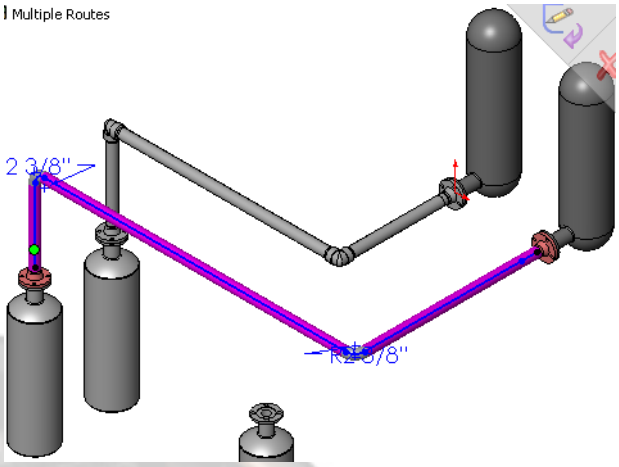
11 Select solution.
Go to the simplest solution (least elbows and pipe spools) and click **OK** to use it. Save the pipe using the default naming as `2in_Schedule40_Route# 1`.



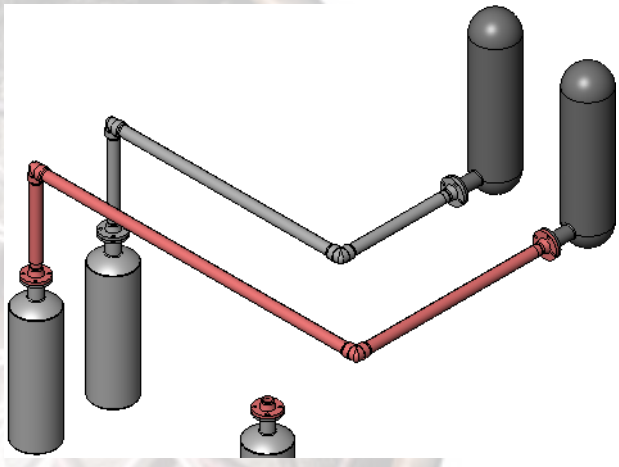
12 Exit the sub-assembly.
Click **Edit Component** to complete editing of the sub-assembly. Use `CLASS 2000` elbows.



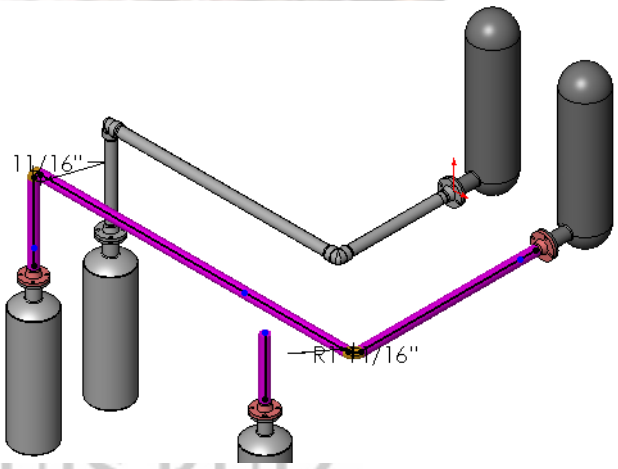
13 Add `Route #2`.
Using the same procedure, create `Route #2` with the same settings. Use Auto Route to connect the open endpoints in this solution.



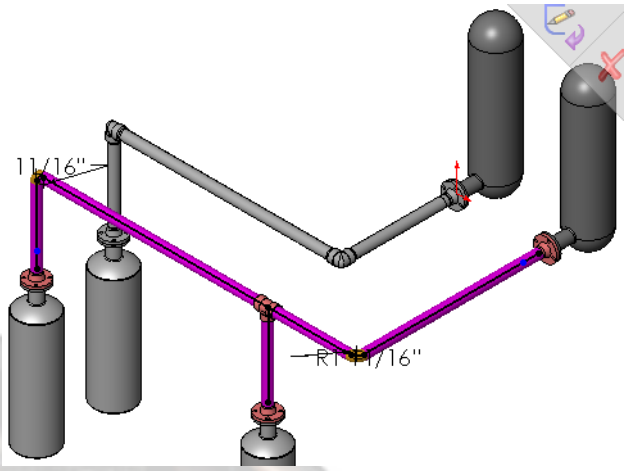
14 Add fitting.
Right-click **Exit Sketch** and add another `welding neck flange` to the route. Again, cancel the new route and place the fitting in the assembly.



15 Add to route.
Use **Add to route** to create a line from the new flange. Use **Split Entities** to break the line above it.



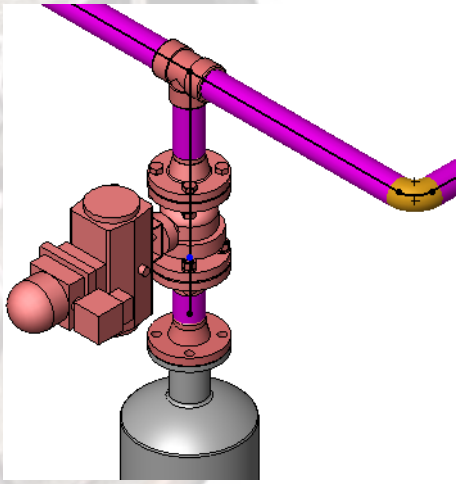
16 Add tee.
Add a `threaded tee (CLASS 2000)` to the junction between the lines.



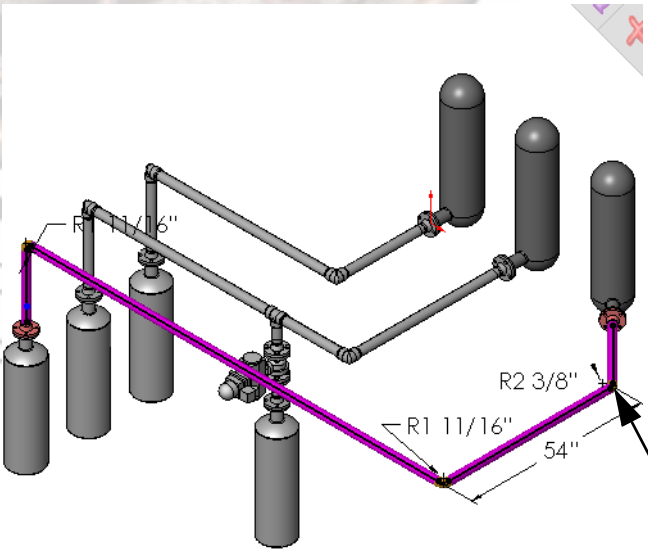
Adding Assembly Fittings

Add an assembly fitting (valve) to the route.

17 Add valve.
Use **Split Entities** to break the new riser near the midpoint. Drag the `2incontrolvalve` from the `assembly fittings/valves` folder and drop it.

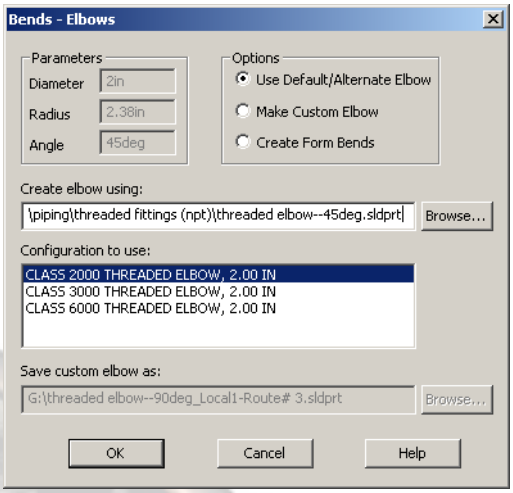


18 Editing the sketch.
Follow the same procedure to create `Route #3`. Use the auto route solution that includes a 45° angle as shown. Fully define and exit the sketch.

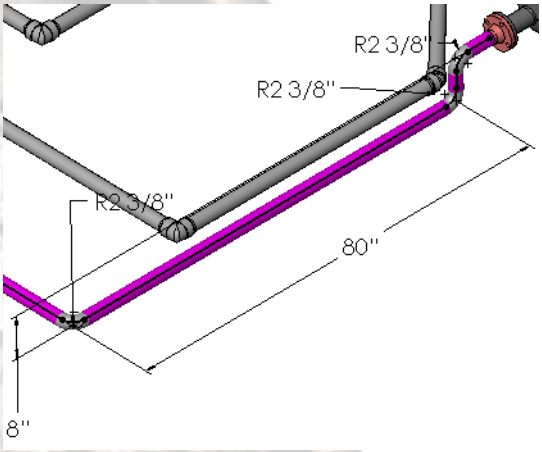


19 Elbows.
The default elbow setting is for 90° elbows. If a 45° elbow is needed, it must be selected explicitly in the **Bends - Elbows** dialog with the **Browse** button.

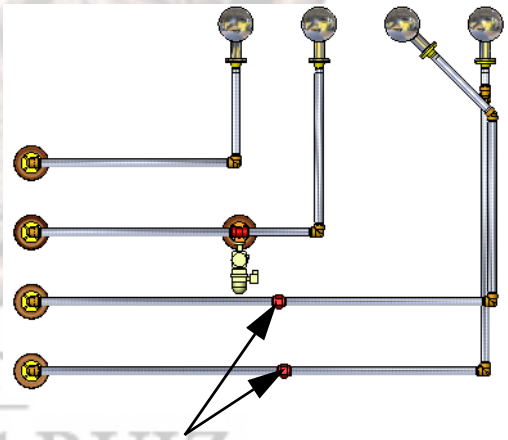
Select the `threaded elbow--45deg` fitting.



20 `Route #4`.
Add the fourth route, adding the **8”** and **80”** dimensions to fully define the sketch.



21 Add unions.
Edit routes and add `threaded union` fittings to `Route# 3` and `Route# 4` as shown.



Piping Drawings

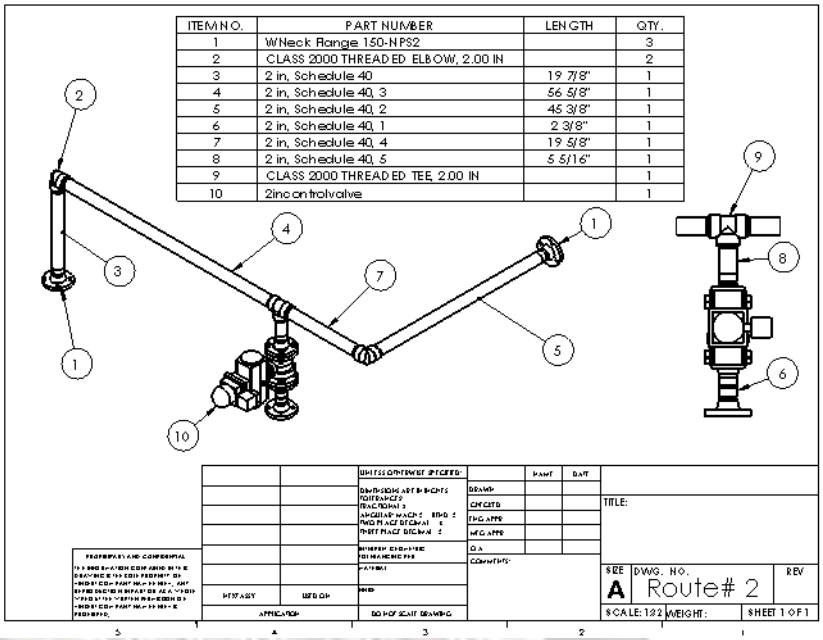
Create piping drawings.

22 Sub-assembly Drawing.
Create a drawing of the sub-assembly `Route# 2` using the `A-Scale1to2` template.

Tip

Use the `bom-standard` template for the BOM. Use **Column Properties** to change the `DESCRIPTION` column to one that uses the

custom property `swpipelength`.



ITEM NO.	PART NUMBER	LENGTH	QTY.
1	WNeck Flange 150-NPS2		3
2	CLASS 2000 THREADED ELBOW, 2.00 IN		2
3	2 in, Schedule 40	19 7/8"	1
4	2 in, Schedule 40, 3	56 5/8"	1
5	2 in, Schedule 40, 2	45 3/8"	1
6	2 in, Schedule 40, 1	2 3/8"	1
7	2 in, Schedule 40, 4	19 5/8"	1
8	2 in, Schedule 40, 5	5 5/16"	1
9	CLASS 2000 THREADED TEE, 2.00 IN		1
10	2inc ontrolvalve		1

23 Save and close the files.

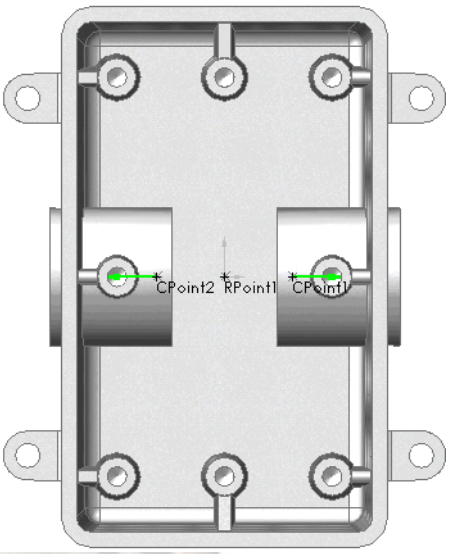
Exercise 3: Creating Library Parts

Edit this part and create a library part for use in another lab.

This lab uses the following skills:

- Adding reference geometry.
- Creating CPoints and RPoints.
- Adding parts to the Design Library.

Units: **inches**



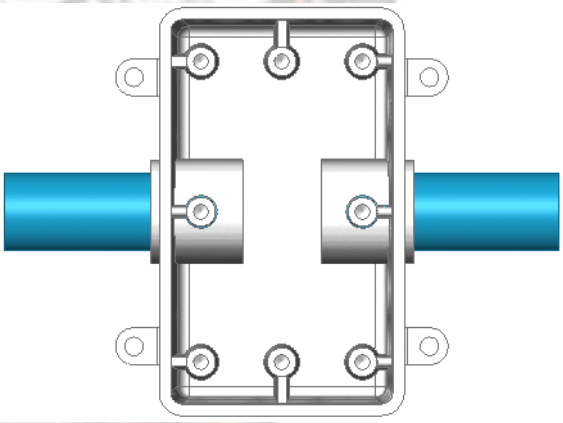
Note

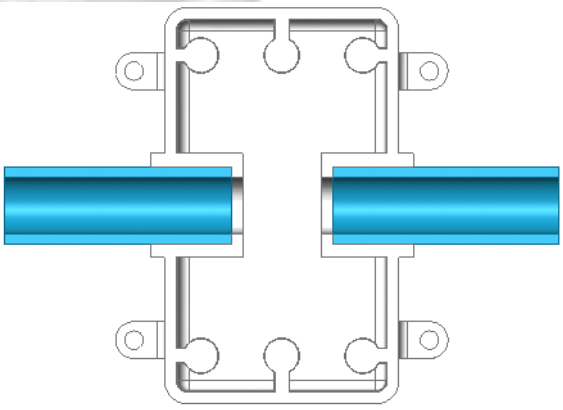
This Exercise illustrates the routing of electrical conduit as piping. Currently it is not possible to route electrically through conduit.

Function

The library part is an electrical box that is placed inline on the route. It is placed at a common endpoint on the route line and cuts the pipe back in two places.

The lower image shows the section view of the part and pipe line.





Procedure

Open an existing part.

1 Open `PVC Inline Box`.

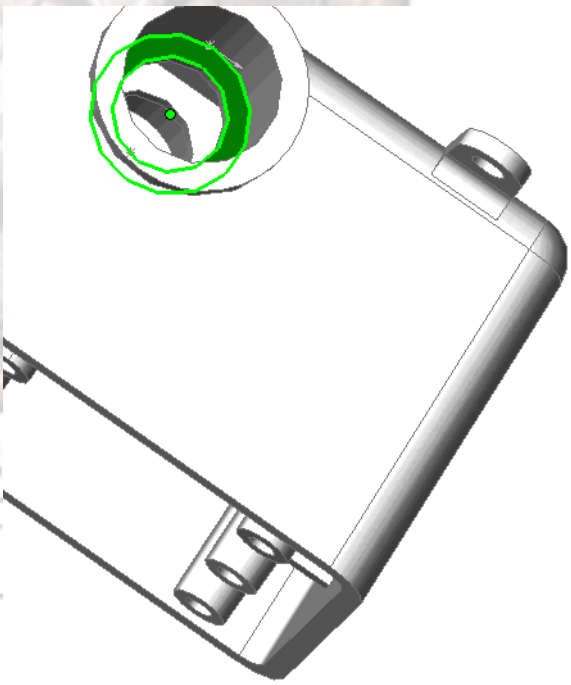
Open the `PVC Inline Box` part. It is an imported solid model.

2 Add point.

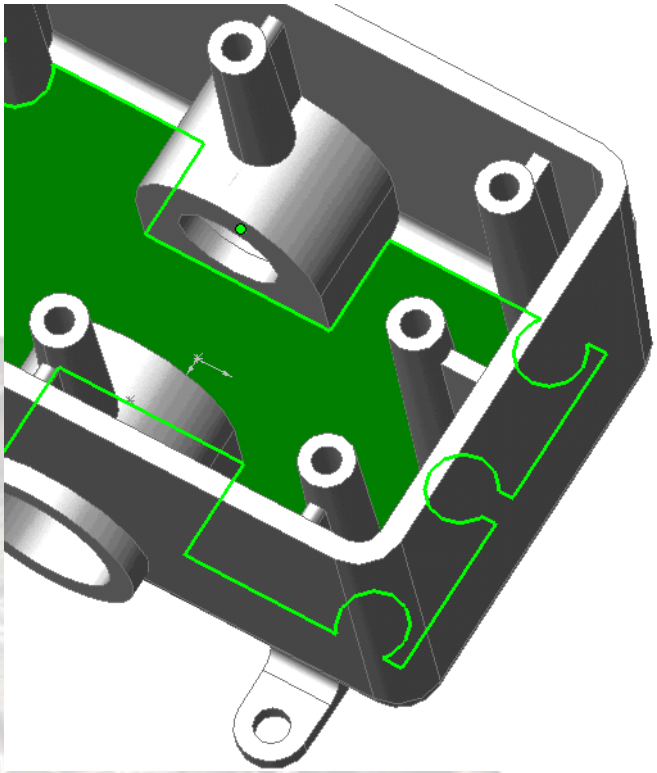
Sketch on the inner face as shown. Add a point that is Concentric to the circular edges.

3 Repeat.

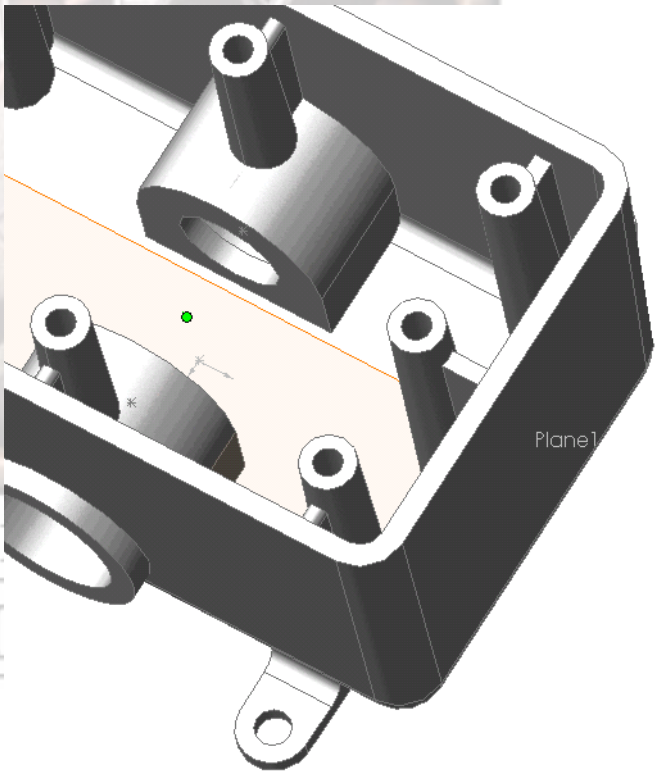
Repeat the procedure on the opposite side creating another concentric point.



4 **Plane.**
Create a new reference plane using the selected face and either of the points. Start a new sketch on this plane.

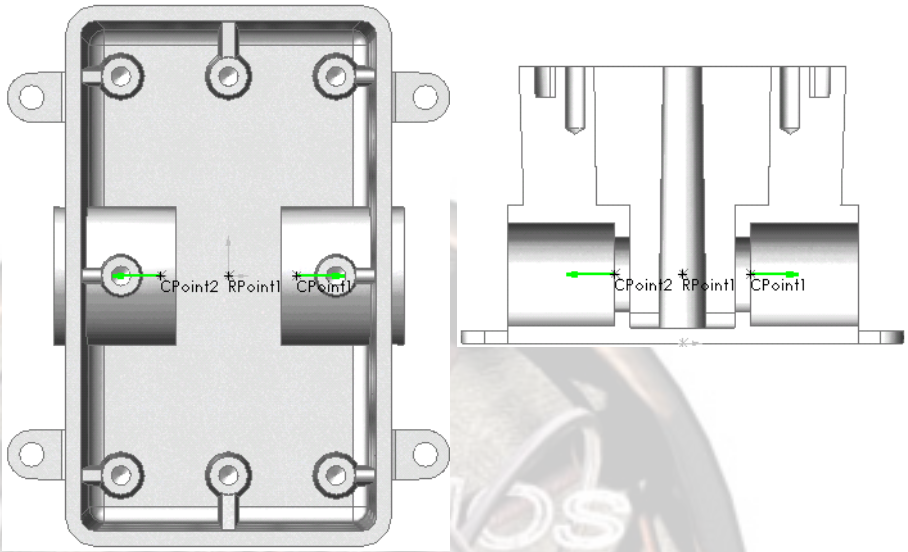


5 **Point between.**
Add a point at the origin as shown. Hide the plane.



6 Cpoints and Rpoint.

Create CPoints using the points and faces used to create the points. Use the settings for **Fabricated Pipe** and **Nominal Diameter 0.5”**. Create one Rpoint using the point between the CPoints.



7 Design table.

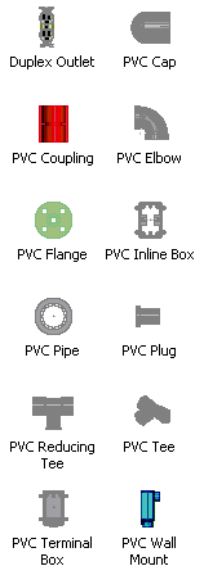
Auto-create a design table for the part. Add the configuration name `Inline Box, 0.5in, Sch 40, PVC` as a copy of `Default`.

	A	B	C	D
1	Design Table for: PVC Inline Box_&			
2		Diameter@CPoint1	Diameter@CPoint2	
3	Default	0.5	0.5	
4	Inline Box, 0.5in, Sch 40, PVC	0.5	0.5	
5				

Sheet1

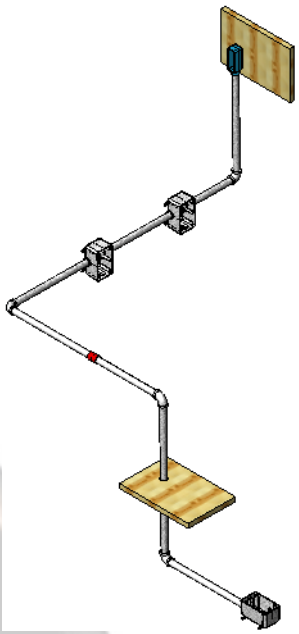
8 Add to library.

Drag and drop the part into the `PVC` folder. Keep the name `PVC Inline Box`.



Exercise 4: Using Library Parts

Create a route assembly using a set of PVC library parts, including one created in the previous lab, *Exercise 3: Creating Library Parts* on page 123.



This lab uses the following skills:

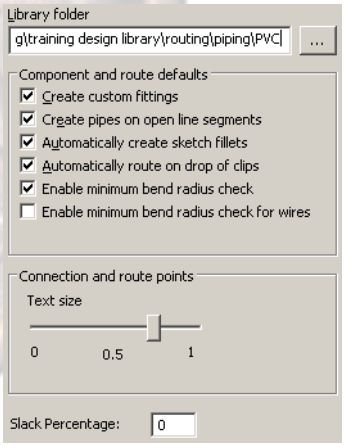
- Setting route options and route properties.
- 3D sketching.
- Adding library parts to the route.

Units: **inches**

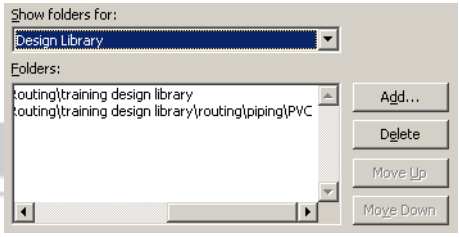
Setup

Setup SolidWorks Routing to use a different library, in this example the included `PVC` library. These settings point the software to the library fittings that you are currently using.

1 **Routing options.**
Click **Tools, Options, System Options, Routing** and set the **Library folder** to `PVC`.



2 **Design library settings.**
Click **Tools, Options, System Options, File Locations** and add the `PVC` folder.



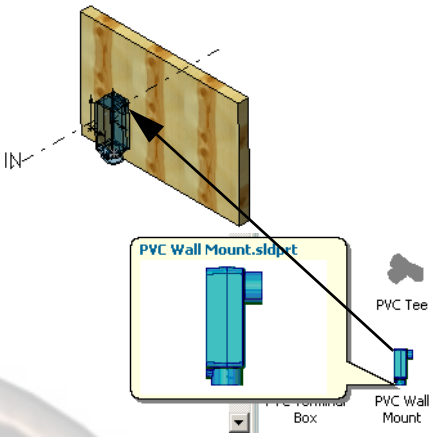
Starting the Route

The route is added to an existing assembly that contains some components to start and stop it.

3 **Open `PVC Assembly`.**
Open the existing assembly `PVC Assembly`.

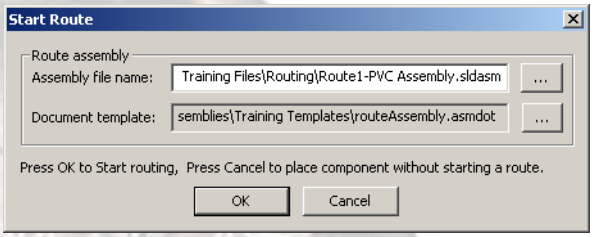
4 Insert fitting.

Drag the PVC Wall Mount fitting and drop it on the edge of the circular hole in the Mount<1> fitting. The fitting contains a mate reference and will snap to the hole.



5 Start Route.

The fitting also contains routing features to begin a new route. Name the assembly Route1-PVC Assembly and make sure that the routeAssembly template is used.

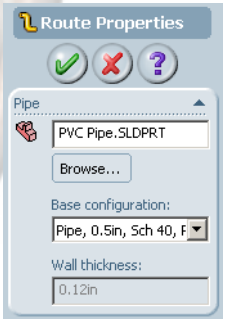


Route Properties

The route properties determine the characteristics of the route including the pipe diameter, elbow selection and library source.

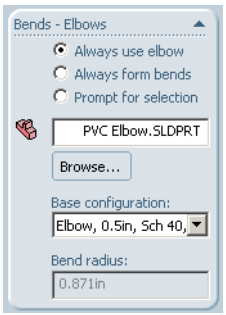
6 Pipe.

The **Pipe** setting determines what part is used to represent the straight pipe spools. Set it to PVC Pipe and the 0.5in base configuration.



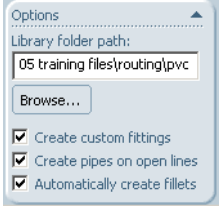
7 Bends - Elbows.

The **Bends - Elbows** setting determines what should be done where lines change direction. Click **Always use elbow**, the PVC Elbow and the 0.5in base configuration



8 Options.

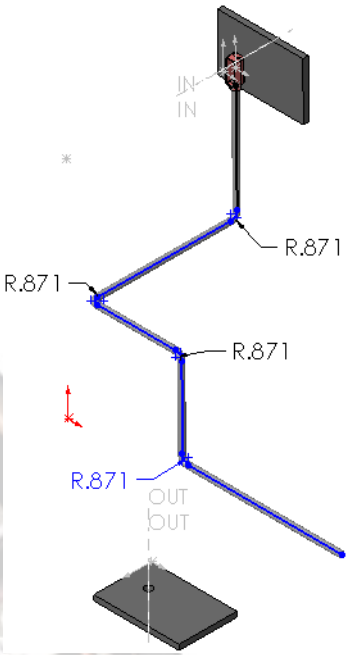
The **Options** setting sets the library folder for fittings and automatic creation options. Select the PVC library folder and click all three options.



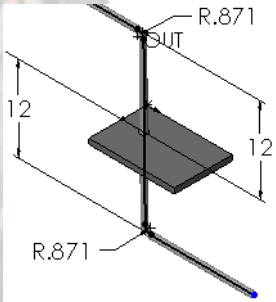
3D Sketching

Use 3D sketching to create the route.

9 Lines.
From the starting line, add the lines shown at right in the 3D sketch.

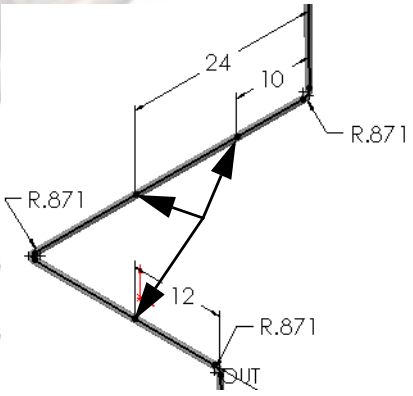


10 Relation and dimensions.
Add a coincident relation between the riser and the `OUT` axis. Add dimensions from the top face of the `Mount<2>` fitting as shown.



Note The sketch remains *under*defined.

11 Split Entities.
Use **Split Entities** to break the lines in three places as shown. Dimension the resulting endpoints as shown.



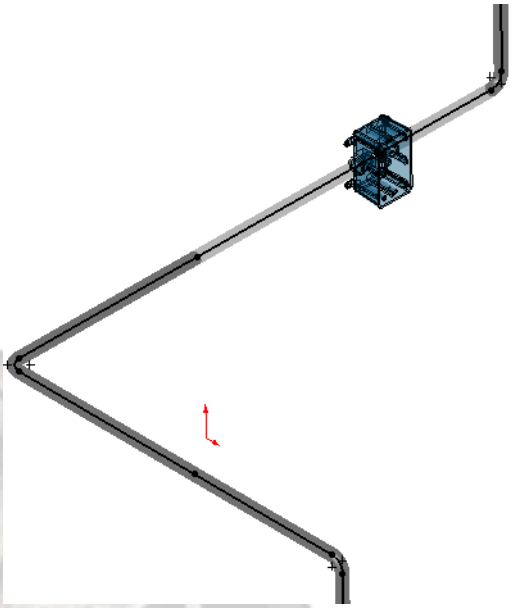
Adding fittings

Add inline and terminal fittings to the route.

All pipes and route fittings (library part components) are 0.5in, Schedule 40 configurations.

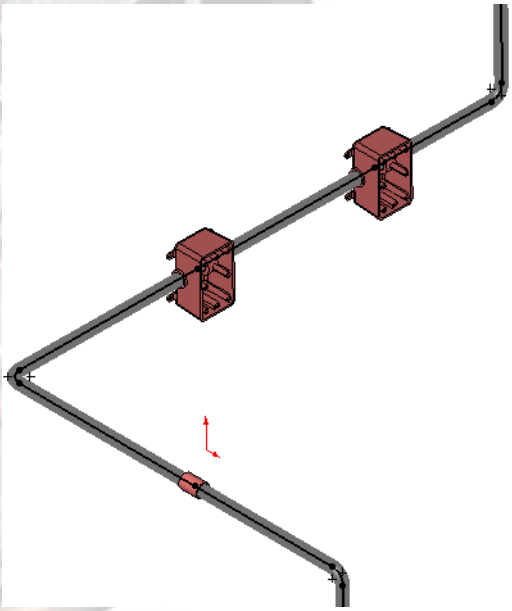
12 Add PVC Inline Box.

Drag the PVC Inline Box part into the assembly and drop it on the endpoint.



13 More inline fittings.

Add another instance of the PVC Inline Box. Also, drag and drop a PVC Coupling as shown.



Note

If you did not complete the previous lab *Exercise 3: Creating Library Parts* on page 123, use PVC Inline Box_&.

Lesson 10
Electrical Routes

Upon successful completion of this lesson, you will be able to:

- Route wires and cables using Auto Route and hardware.
- Add wire data to a route subassembly.

Electrical Routes

Electrical Routes use electrical components and an optional cable / wire library to form wire and harness route sub-assemblies. In their simplest form, electrical routes are much like flexible tubing routes. However, Electrical Harnessing and Cable Detailing adds the ability to specify individual pins, wire types and colors, wire coverings and other attributes. Harnessing will be covered in depth in *Lesson 11: Importing Electrical Data and Electrical Harnessing.*

Electrical routes also differ from tubing and piping routes in that there is no source SolidWorks part file for the wire, as there is for a tube or pipe.
Instead, electrical routes may use an optional cable / wire library file. The cable / wire library defines the attributes of wires and cables, and is in .xml format, but may be imported from Excel format. With no cable / wire library specified, the route uses a generic wire or cable part representation.

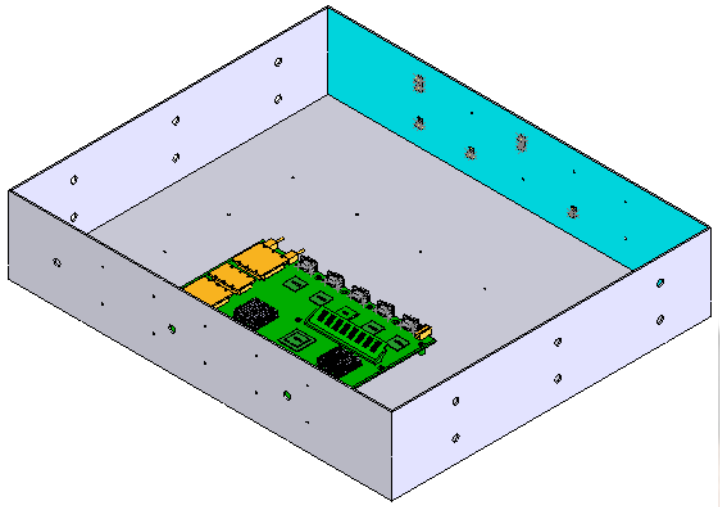
Other elements of electrical routing:

- **Routing Hardware**, which includes clips, clamps and brackets, as well as ring terminals and other simple connectors. These help to define the start and end points, the path of the route around other components, and are useful in auto routing.
- A **Component Library** file cross-references the components in a from-to list (see below) to the SolidWorks part documents for those components. The component library file is also in .xml format (importable from Excel). Component libraries will be covered in more depth in *Lesson 11*: *Importing Electrical Data and Electrical Harnessing.*
- A **From-To List** defines component and pin connection data for a route. You can generate a From-To list manually, or use a third-party schematic program that generates the electrical connection data. From-To lists are in Excel format. From-To lists will be covered in more depth in *Lesson 11: Importing Electrical Data and Electrical Harnessing.*

Basic Electrical Routing

Basic electrical routing is very similar to tube routing. Wires can be routed manually or automatically, orthogonally or flexibly. Clips can aid in route placement.

1 Open assembly `Signal control system`.
The assembly includes an enclosure, a circuit board assembly, and some pre-placed wire clips.



Electrical Connection Points

There are some unique properties of Connection Points (CPoint) for electrical route connectors:

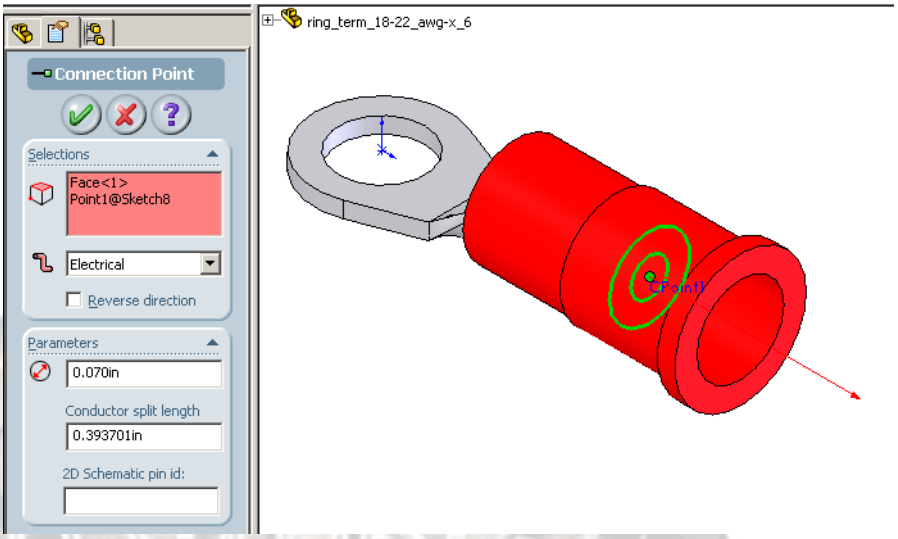
- The **Nominal diameter** of an electrical CPoint defines the maximum diameter of cable or wire bundle that the connector can take.
- The **Conductor split length** adds to the cut length of the cable to allow for strip back, termination, and so on.
- **2d Schematic pin id** assigns identification information for the pin if the connection point is for one pin in a multi-pin connector. If the connection point is for multiple pins, leave blank.

2 The ring terminal.
In the Design Library, under `routing\electrical`, locate `ring_term_18-22_awg-x_6`. Right click on it, and select **Open**.

3 Edit CPoint.
Right-click the CPoint, and select **Edit Feature**.
As with other CPoints, it references a point sketch and a planar face.

4 Electrical Parameters.
The **Nominal diameter** sets the maximum allowable wire diameter for this connector at **0.70”**

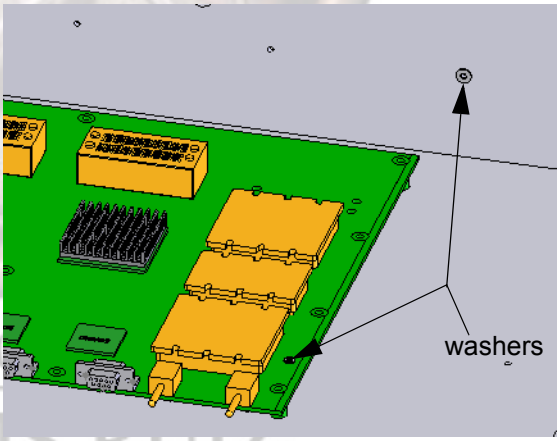
The **Conductor split length** is set at **0.3937”** (10 mm). A wire connected to this terminal will have this value added to its calculated length.



This is a single wire connector, so there is no reason for a Schematic pin id. For connectors with multiple contacts or pins, it may be desirable to add a CPoint for each and index them for individual wire assignments.
Exit CPoint properties and close this part file.

5 View Orientation.
Orient the assembly model to the view called `Wire route 1`.

6 Locate the 2 washers.
On the side panel closest to the circuit board, and on the edge of the circuit board itself, there are two washers mated to small holes. We will add a ground wire between these two contact points.



Drag one of these terminals onto the circuit board. When dragged over the washer, the pin-in-hole cursor appears, indicating a concentric mate reference. Drop the terminal onto the washer. The **Start Route** dialog appears.

7 Create and save the route.
Accept the default route sub-assembly name to start the route.

Electrical Route Properties

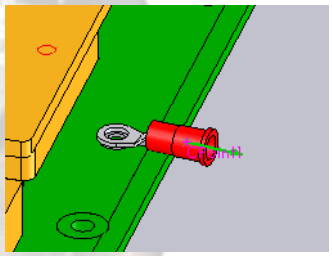
The Properties dialog for an electrical route differs from that of other route types:

- **Sub-type**
  Harness: A route with a bundle of wires or cables.
  Cable/Wire: A route with a single cable or wire.
- **OD**
  Set the route outside diameter. Can not exceed the **Nominal Diameter** assigned to the CPoint.
- **Fix diameter**
  Sets the route diameter to the value set in OD. Clear to enable diameter updates based on the diameters of the cables and wires that you route. Generally, this option should be left *unchecked*.

  See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

---

8 Properties.
In the Route Properties dialog, select sub-type **Cable/Wire**, and set the OD to **1.5mm.**



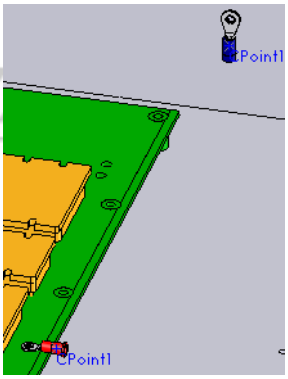
Make sure **Fix diameter** is unchecked.

While the Route Properties dialog is still displayed, the orientation of the ring terminal can be adjusted. Select it and drag the barrel so that it exits to the right, off the board. Click **OK**.

9 Auto Route dialog.
The Auto Route dialog launches; If we had a predetermined route for this wire, we could use existing clips or terminal connections, or drop clips from the Design Library, to Auto Route. Click **OK** to close this dialog.

10 Save the wire.
We will need another terminal connector. Exit the route sketch, and save the wire part as it currently exists.

11 Drop another connector.
While still editing the route sub-assembly, drop a different ring terminal on the other washer. This time use `ring_term_awg-14-16_awg-x 8` (the blue one).



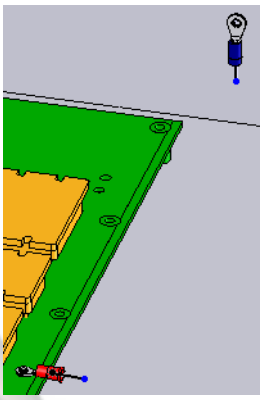
Cancel the Insert Component dialog.

Rotate the connector so that the wire exits down.

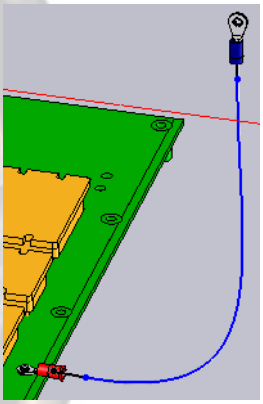
12 Edit the route.
Right click `Route1` in the tree and select **Edit Route**.

13 Add to route.
Right click the CPoint of the new connector and select **Add to route**.

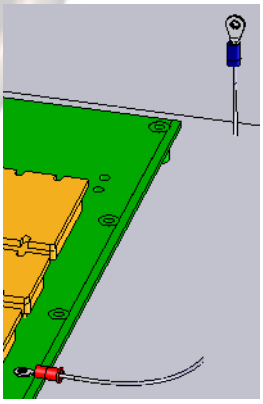
14 Auto Route.
Extend both wire segments to about the length of the strain relief. Right-click on one of the endpoints, and select **Auto Route**. Select the endpoint of the other wire segment to route to it.



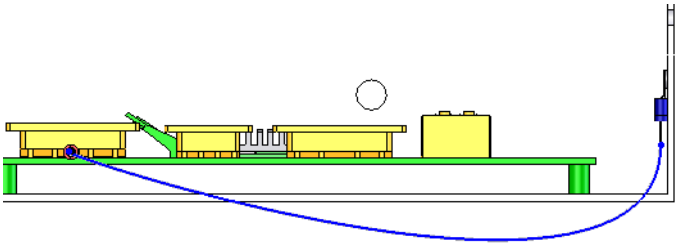
The system auto routes a spline between the terminals. **OK** the Auto Route dialog, and exit the sketch.



15 Modify sketch.
Our wire bows through the bottom of the box, so will need some modification. Edit the sketch again.

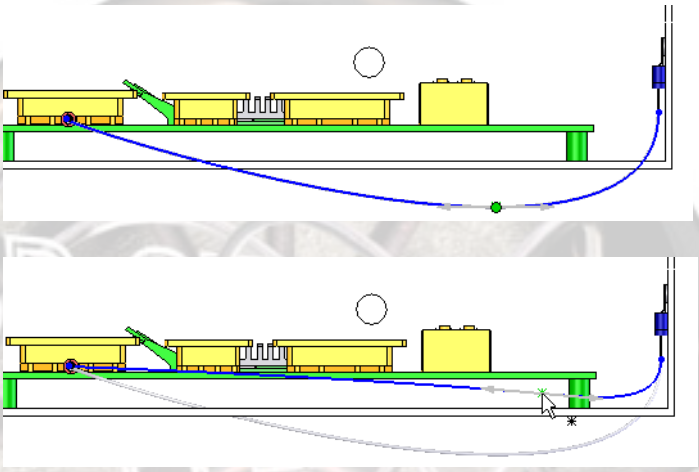


Tip Section view is very handy for the next step.



16 Spline point.

Add a spline point to manipulate the spline. Drag it back inside the box.



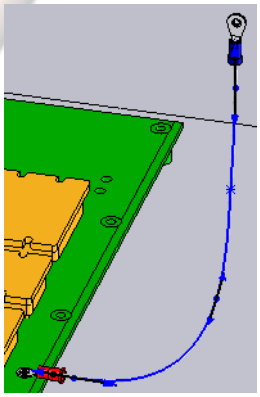
17 Branching.

Let's assume there is another component, to be added later, that also needs a ground connection.

While still editing the sketch, return to the `Wire route 1` view.

18 Add split point.

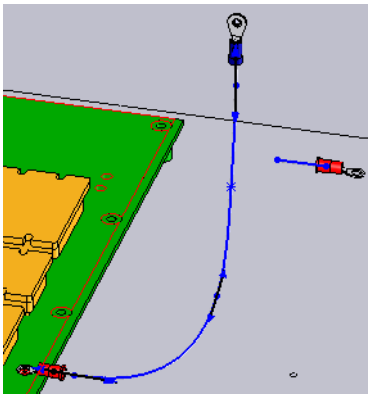
Right click on the route, and select **Split Entities**. Click on the spline near its midpoint. Right click, and return to **Select** mode.



19 Add connector.

From the Design Library, drag and drop another ring terminal `ring_term_18-22_awg-x_6` (red) into the assembly.

Answer **Yes** to the prompt.



Note When a routing connector is added while editing a routing sketch, it is added to the route automatically.

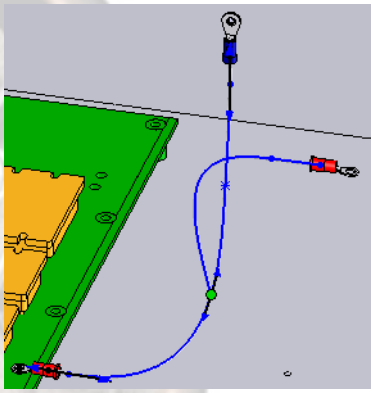
20 Make the connection.

Right-click on the new connector's route line endpoint, and select **Auto Route**.

Route to the split point on the spline. A connecting route wire segment is added.

Close the sketch and save the assembly.

When the component is later added, the route sub-assembly can be edited, this connector mated to the component, and the route wire will update.



Adding Wire Information

The route has been formed, but we know very little about the wire itself. What is its gage and color, is it stranded or solid? Does it have a protective covering of any kind?

Using the Wire Library, we can assign a standard wire to the route and add other information as well.

The Cable / Wire Library

The Cable / Wire Library file contains data on the available wires. It can easily be generated in Microsoft Excel format and imported into the assembly as a .xml file.

Note SolidWorks Routing has a default cable / wire library (`cable.xml`) located in the `design library\routing\electrical` folder. For this exercise we will use a comparable file located in the Training Design Library.

The cable / wire library file, including importation of a custom file, will be covered in more depth in *Lesson 11: Importing Electrical Data and Electrical Harnessing*.

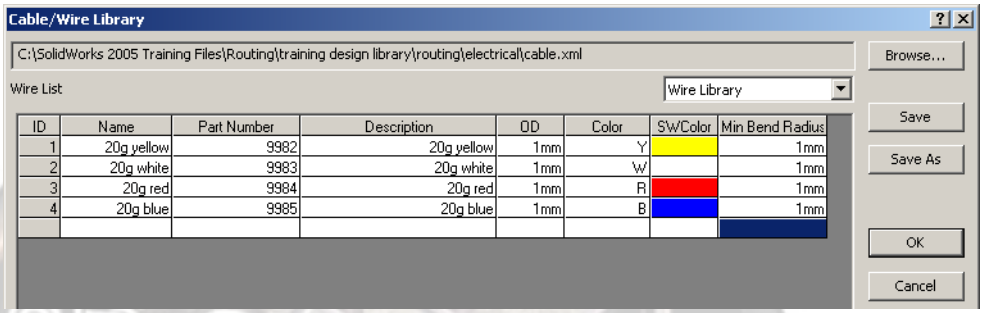
See **SolidWorks Routing Help Topics** in the **Help** menu for more information on the wire library and Excel file column format.

21 Open the library.

From the Electrical toolbar, click **Cable / Wire Library** .

Browse to `\\training design library\routing\electrical` and select `cable.xml` and **Open.**

In the menu, select **Wire Library** from the list. The available wires are displayed.



Click **OK**.

22 Edit the route.

In order to add actual wire data to this route, we need to first edit the route.

Edit Wires

After you define paths between connectors in the route sub-assembly, you can associate cable/wire data with the paths. The route diameter updates to reflect the diameter of the cables or wires you select for each path.

Where to find it:

While editing a route,

- From the Electrical toolbar, click **Edit Wires** ,
- From the main menu, select **Electrical, Edit Wires...**.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

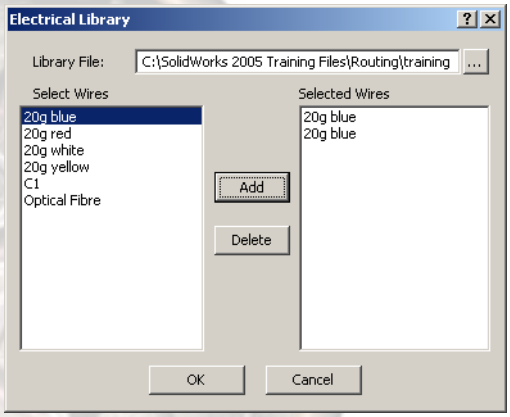
23 Edit Wires.

Select **Edit Wires** .

Here we can add or delete wires, or reorder segments.

24 Add a wire.

Click **Add Wire** . The Electrical Library dialog appears. Here we can select a wire specification to add. Let's choose the `20g blue` wire. Repeat to add a second blue wire. Click **OK**.

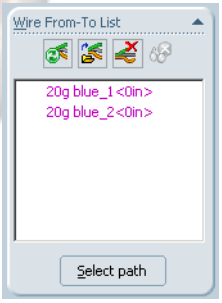


25 Wire added.

The new wires are shown in the list as magenta, indicating they are underdefined. We need to specify what portion of the route these wires are to follow.

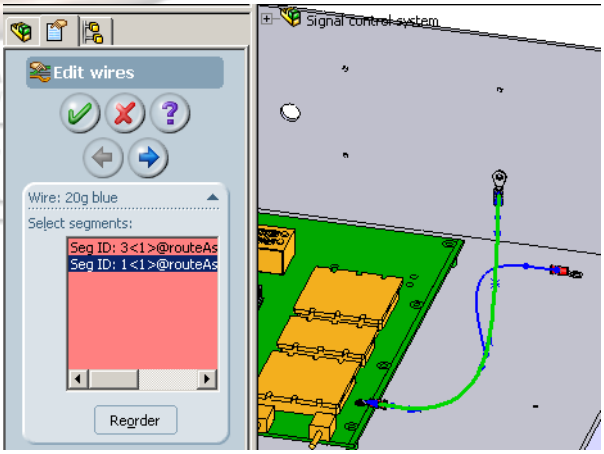
26 Select path.

Choose the first wire in the list, and click **Select path**.



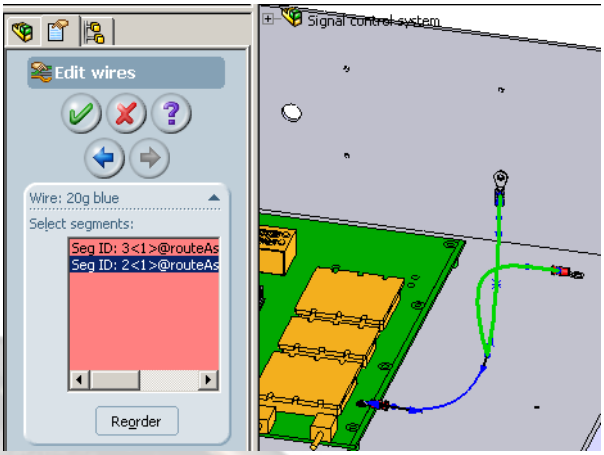
27 Pick the segments.

In the graphics view, pick the two indicated (highlighted green) segments, completing a path from the chassis to the circuit board. Then click **OK**.



28 The other wire.

Repeat for the second wire, this time choosing the path from the chassis to the "floating" ring terminal. Click **OK**.



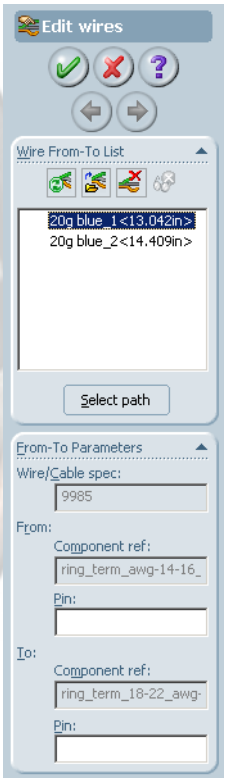
29 Edit wires data.

The Edit wires dialog now displays some valuable information:

- The lengths of each wire
- The wire specification (part number, as supplied in the cable / wire library)
- The From- and To- component names

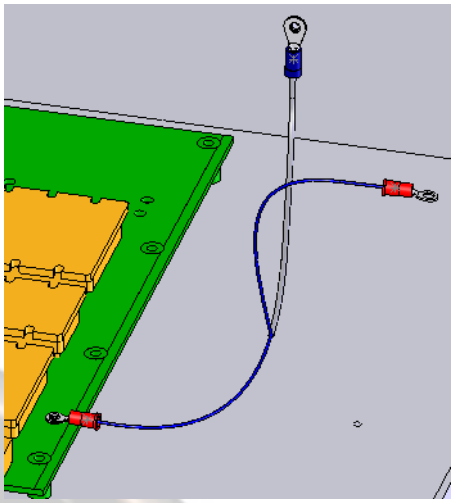
There are no "pins" in these connectors, but if there were we could also specify pins for each wire.

Click **OK**, then exit the sketch.



30 Updated display.

The display of our route has now updated with this new information.



The two single wires display as blue, and the proper size of 1mm diameter, per the specification.

The grouped, parallel run of the wires to the chassis is displayed as a generic gray color, and is noticeable larger in diameter.

3D STUDIOS
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Lesson 11
Importing Electrical Data and Electrical Harnessing

Upon successful completion of this lesson, you will be able to:

- Import wire and component libraries, and assign to routes
- Import a From - To list, and assign to wires to contacts
- Generate a route from available data, with assorted hardware.
- Create a Harness Drawing

Harnessing

Harnessing is the term given to the process of routing a bundle of wires or cables. A harness often involves branches to several different components and connection points. In the simplest terms, cable and wire routing refers to a single cable or wire, harnessing refers to more than one, and includes complete electrical data and a detailed Harness Drawing.

Importing Electrical Data

Electrical routes may use an optional cable / wire library, component library, and from - to list files, to specify vital electrical connection criteria. This electrical data would typically be supplied to the designer by the electrical engineer.

- The **Cable / Wire Library** file consists of a table listing the wires available for any design or even in an entire plant inventory. The data in the table includes information such as part number, wire name, description, diameter, size (gage), and color.

Note

SolidWorks supplies a default cable / wire library called `cable.xml`, located in the `electrical` directory.

- The **Component Library** file is a similar table of connector components required for the route assembly. It includes information such as the part number, the maximum cable/wire/segment diameter for the connector, the part file location (path), its usage configuration, and a general description.

Note

SolidWorks supplies a default component library called `components.xml`, located in the `electrical` directory.

- The **From - To List** file is yet another table, with the data that assigns wires in the bundle to their respective pins on the appropriate connectors. It consists of data such as wire name, wire spec (part number), and connector and pin “from” and “to” designations. See `from-to.xls` in the Lesson 11 Case Study folder.

Note

The Excel data files can contain many more columns of data, but they will not be used in routing.

Cable / Wire Library

The Cable / Wire Library file contains data on the available wires.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

1 Open `cables-wires.xls` in Microsoft Excel.
The cable / wire library file is located with the other files in the `Lesson 11` folder for this lesson.

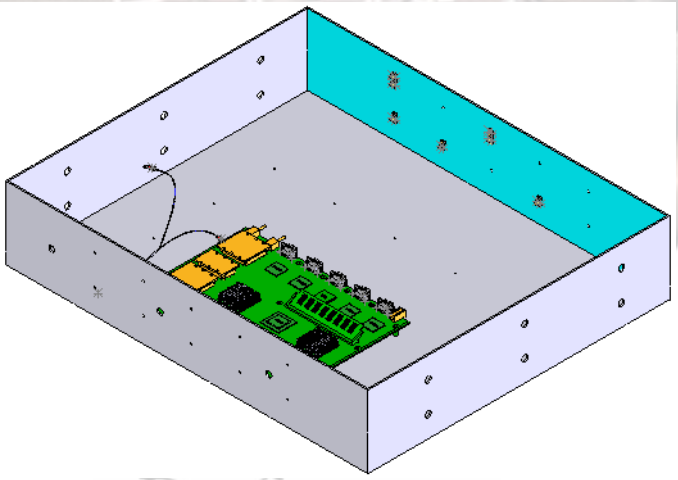
This Excel files contains information about the wires available to be used in our assembly.

For the sake of simplicity, this library has been limited just to little more than those wires that we will use in this example.

Partno	Wirename	Cablename	Description	Outer Diameter	Min Bend Radius	Size	NoOfCores	Color	DispColor	/cable/core/conductorOD/@value	/cable/core/conductorOD/@value/#agg
9978-20GST-BRN	AWM 1061-35		20g Stranded	0.001422	0	20g Stranded		brown	brown		
9978-20GST-R	AWM 1061-36		20g Stranded	0.001422	0	20g Stranded		red	red		
9978-20GST-ORNG	AWM 1061-37		20g Stranded	0.001422	0	20g Stranded		orange	orange		
9978-20GST-Y	AWM 1061-38		20g Stranded	0.001422	0	20g Stranded		yellow	yellow		
9978-20GST-GRN	AWM 1061-39		20g Stranded	0.001422	0	20g Stranded		green	green		
9444		UL AWM 2464 - 4	stranded PVC	0.005512	0		4		white		
	line 1	UL AWM 2464 - 4	core1	0.0016	0	20 AWG		black		0.0016	0.0016
	line 2	UL AWM 2464 - 4	core2	0.0016	0	20 AWG		white		0.0016	0.0016
	line 3	UL AWM 2464 - 4	core3	0.0016	0	20 AWG		red		0.0016	0.0016
	line 4	UL AWM 2464 - 4	core4	0.0016	0	20 AWG		green		0.0016	0.0016

Note the data. Close the Excel file.

2 Open `Signal control system.sldasm`.
It is in the completed state from the previous lesson.



Import Cable / Wire Library

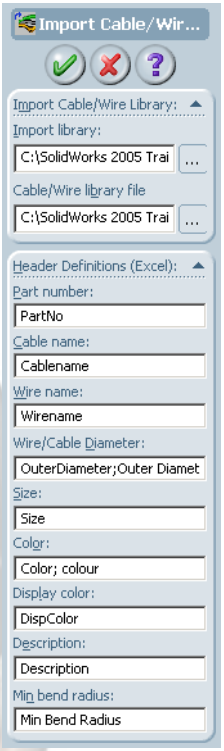
Where to find it:

From the Main Menu, select **Electrical, Import Data, Import Cable Library...**.

---

3 Import the wire library.
Select **Electrical, Import Data, Import Cable Library...**.

Under **Import Library**, select the **Browse** button to find the file `cables-wires.xls` in the `Lesson 11` folder.



4 Set the .xml file name and location.
Under **Cable/Wire library file**, browse to the folder `\\training design library\routing\electrical` to save the .xml file. Save it as `cables-wires.xml`.

5 Header Definitions.
These entries must agree with column headers in the Excel file for import to be successful. If more than one header is possible for the same item, separate with a semi-colon.

Tip

Please refer to **SolidWorks Routing Help Topics** in the **Help** menu for more information on Excel file columns construction.

Click **OK**.

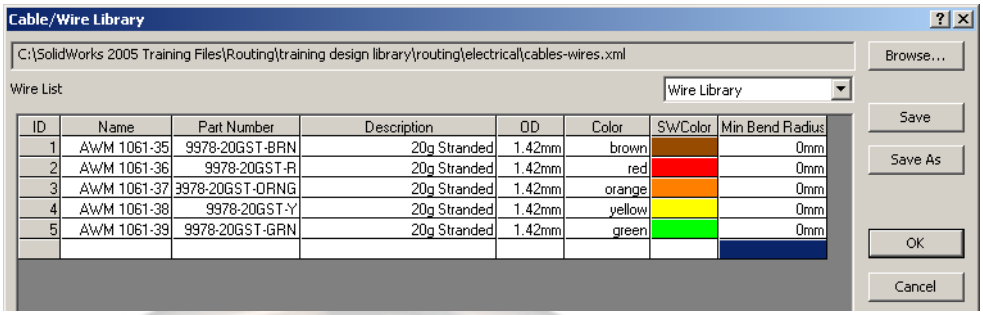
6 Confirmation.
The import is confirmed with a pop-up window. If there are import errors, they will be displayed here.

The library is now in .xml format.

7 Open the library.

From the Electrical toolbar, click **Cable / Wire Library**  . First we see the data designated as **Cable Library**.

In the menu, select **Wire Library** from the list. The available wires are displayed. Click **OK**.



ID	Name	Part Number	Description	OD	Color	SWColor	Min Bend Radius
1	AWM 1061-35	9978-20GST-BRN	20g Stranded	1.42mm	brown		0mm
2	AWM 1061-36	9978-20GST-R	20g Stranded	1.42mm	red		0mm
3	AWM 1061-37	9978-20GST-ORNG	20g Stranded	1.42mm	orange		0mm
4	AWM 1061-38	9978-20GST-Y	20g Stranded	1.42mm	yellow		0mm
5	AWM 1061-39	9978-20GST-GRN	20g Stranded	1.42mm	green		0mm

Component Library

The Component Library file contains information on the connectors to be used in the route assembly.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

8 Open `connectors.xls` in Microsoft Excel.
The connector library file is located with the other files in the `Lesson 11` folder for this lesson.

This Excel files contains information about the components to be used in our assembly. There are just two connectors to be used in this route assembly.

Partno	Max diameter	Configname	Description	Libname	Pins	Terminals	Gauge
Socket-6pinmindin	0.005	Default	connector	design library\routing\electrical\Socket-6pinmindin.SLDPRT	1,2,3,4,5,6		
db9 male	0.005	Default	connector	\training design library\routing\electrical\db9 male.SLDPRT	1,2,3,4,5,6,7,8,9		

Note

Important: If the training files were installed to a location other than `C:\SolidWorks 2005 Training Files`, the `Libname` path should be adjusted now.

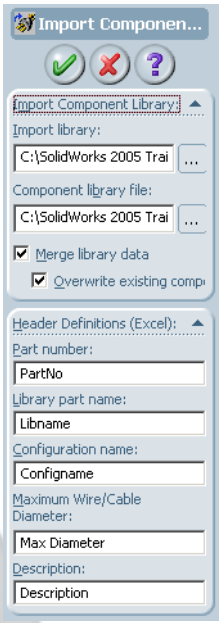
Close the Excel file.

Import Component Library

Where to find it:

From the Main Menu, select **Electrical, Import Data, Import Component Library...**.

9 Import the component library.
Select **Electrical, Import Data, Import Component Library...**.



Under **Import Library**, select the **Browse** button to find the file `connectors.xls` in the `Lesson 11` folder.

10 Set the .xml file name and location.
Under **Component library file**, browse to the folder `\\training design library\routing\electrical` to save the .xml file. Save it as `connectors.xml`.

11 Merge library data.
If an *existing* library .xml file is chosen, the option to merge the files together into one is given.

12 Header Definitions.
These entries must agree with column headers in the Excel file for import to be successful. If more than one header is possible for the same item, separate with a semi-colon.

Tip

Please refer to **SolidWorks Routing Help Topics** in the **Help** menu for more information on Excel file columns construction and file location for the SolidWorks part files.

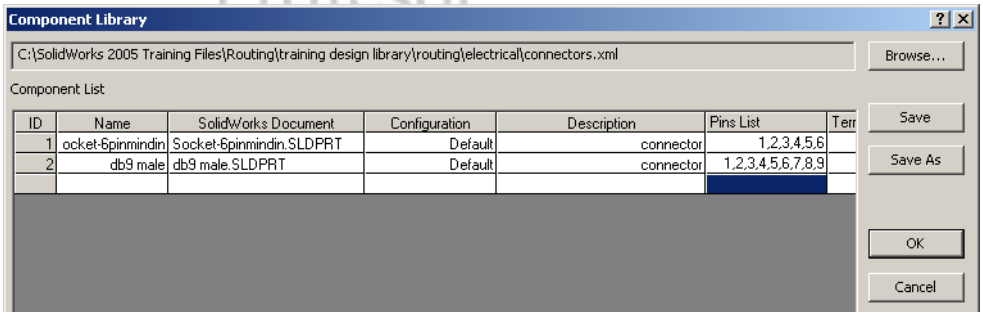
Click **OK**.

13 Confirmation.
The import is confirmed with a pop-up window. If there are import errors, they will be displayed here.

The library is now in .xml format.

14 Open the library.

From the Electrical toolbar, click **Component Library**  . The available components are displayed. This particular library contains *only* the components we need for this lesson. Click **OK**.



From - To List

The From -To list contains information on how the connectors are to be joined and with which wires.

With a from-to list, you can import both the connection data and the component data into a new or existing route sub-assembly. Alternatively, you can import only the connection data (no component data) or only component data (no connection data). You are not required to use a from-to list - you can also enter the connection data and insert components manually when you create a route.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

---

15 Open `from-to.xls` in Microsoft Excel.

Note the data, specific connection criteria for pins in the connectors.

Wire	Spec	From Ref	From Pin	Partno	To Ref	To Pin	Partno
P5V	9978-20GST-BRN	J1	1	Socket-6pinmindin	J2	1	Socket-6pinmindin
SW1	9978-20GST-R	J1	2	Socket-6pinmindin	J2	2	Socket-6pinmindin
INTRPT	9978-20GST-ORNG	J1	3	Socket-6pinmindin	J2	3	Socket-6pinmindin
RLGHT	9978-20GST-Y	J1	4	Socket-6pinmindin	J4	1	db9 male
ALERT	9978-20GST-GRN	J1	5	Socket-6pinmindin	J4	2	db9 male
SYNC	9978-20GST-BRN	J1	6	Socket-6pinmindin	J3	1	db9 male
SDATA1	9978-20GST-R	J2	4	Socket-6pinmindin	J4	4	db9 male
SDATA2	9978-20GST-ORNG	J2	5	Socket-6pinmindin	J3	2	db9 male
SDATA3	9978-20GST-Y	J2	6	Socket-6pinmindin	J4	4	db9 male

Close the Excel file.

Import From-To List

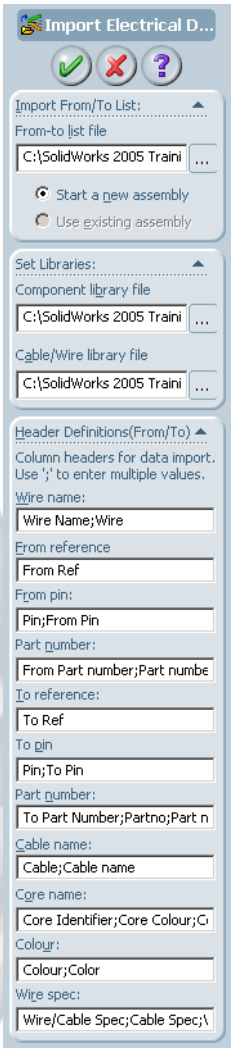
Where to find it:

- From the Electrical toolbar, click **Import Electrical Data**  ,
- From the main menu, select **Electrical, Import Data, Import Electrical Data...**.

16 Import the electrical data.
Select **Electrical, Import Data, Import electrical Data...**.

Under **Import From-To List**, select the **Browse** button to find the file `from-to.xls` in the `Case Study` folder of this lesson.

Note that, under **Set Libraries**, you may also select alternate cable / wire and component libraries.



17 Header Definitions.
As with the libraries, these entries must agree with column headers in the Excel file for import to be successful. If more than one header is possible for the same item, separate with a semi-colon.

Refer to **SolidWorks Routing Help Topics** in the **Help** menu for more detailed information on From - To list headers.

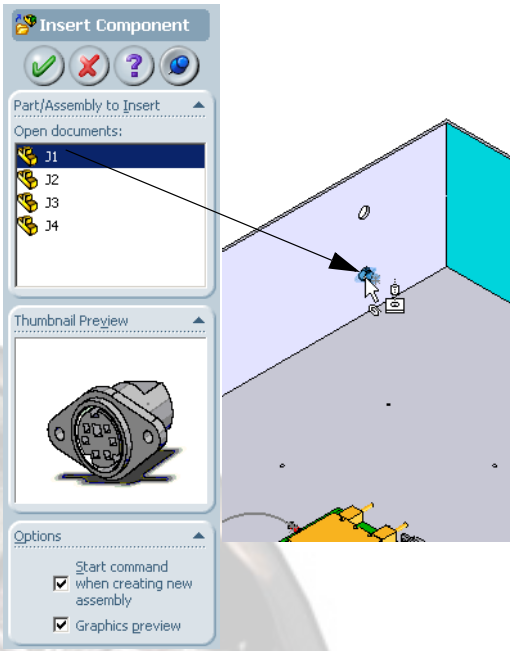
Click **OK**.

18 Start the route.
With the import of From-To data, the route is initiated. Accept the default route name and template by clicking **OK**.

SolidWorks confirms that components can be placed. Click **Yes** to begin placement.

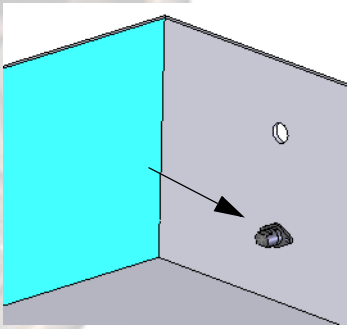
19 Insert components.
The available components for this route appear in the **Insert Components**, **Open documents** list.

20 Drop components.
Select the first connector, `J1`, and bring it into the assembly graphics area. Look for the "peg-in-hole" SmartMate cursor, and click to drop on the lower hole of the left side.



Tip You do not need to "drag and drop" components by holding the left mouse button. Selecting them is all that is required.

21 Insert J2.
Repeat with `J2` for the lower hole on the opposite side. Be sure the body is oriented toward the inside of the enclosure.

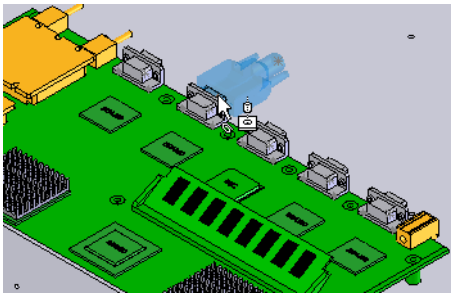


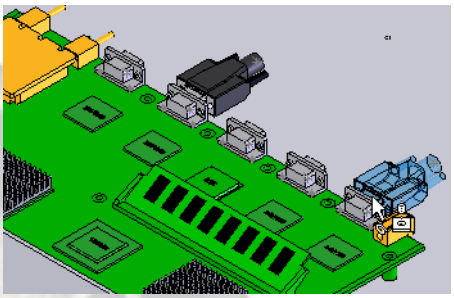
Note It is not necessary to insert all components at the beginning. You can resume insertion later by using **Insert Components** while editing the route.

22 Insert J3 and J4.

Adjust the zoom to see the circuit board clearly.

Insert J3 on the second and J4 on the 5th receptacle as shown.



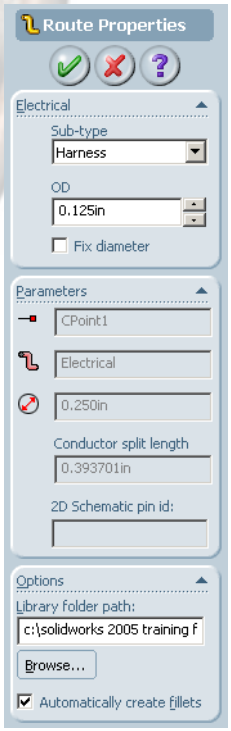


Note

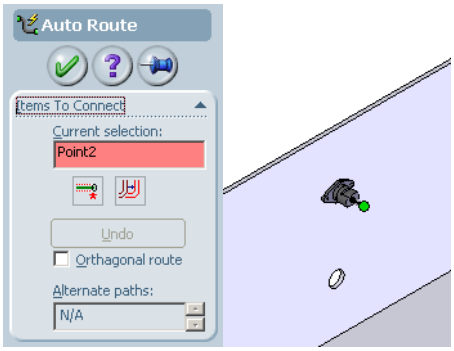
The connector DB9 male (J3 and J4) from the Training Design Library has a mate reference added to it that allows for easy SmartMates. The connector supplied in the SolidWorks Design Library does not have this mate reference by default, but it can easily be added.

23 Route Properties.

When all components are added, SolidWorks asks if you'd like to start modeling the route. Click **Yes**. The **Route Properties** dialog appears. Make certain **Fix diameter** is unchecked, and click **OK**.

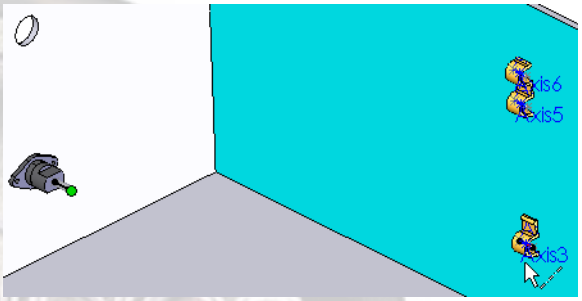


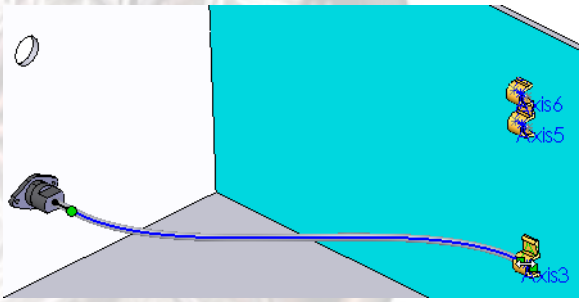
24 Auto Route.
Auto Route launches.



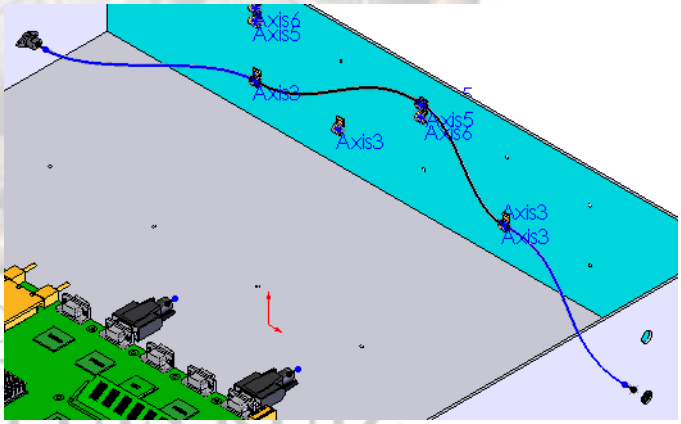
25 Route through clips.
Select **View**, **Axes** to turn on view of axes. It also may be helpful to turn on the **axes selection filter**.

Select the axis of the first lower clip; the route is propagated.





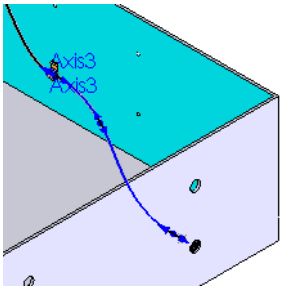
26 Continue
Finish the route through two more clips to the other 6-pin socket (J2) as shown, or improvise your own route.



27 Split entites.

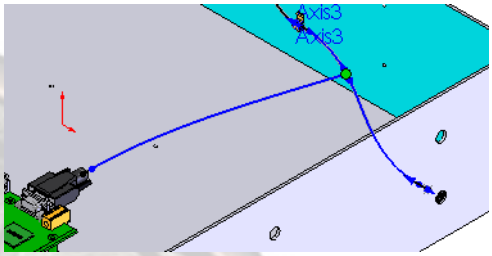
Right click on the route between the third clip and J2 select **Split Entities**.

Split the segment somewhere near the midpoint.



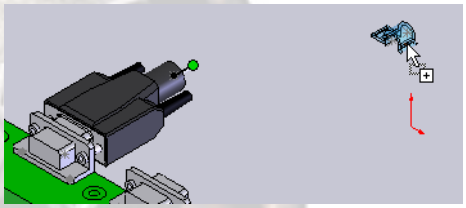
28 Add branches.

Right click on the endpoint of the route segment of the J4 DB9 connector and select **Auto Route**. Route to the split point. Click **OK** to close the **Auto Route** dialog.



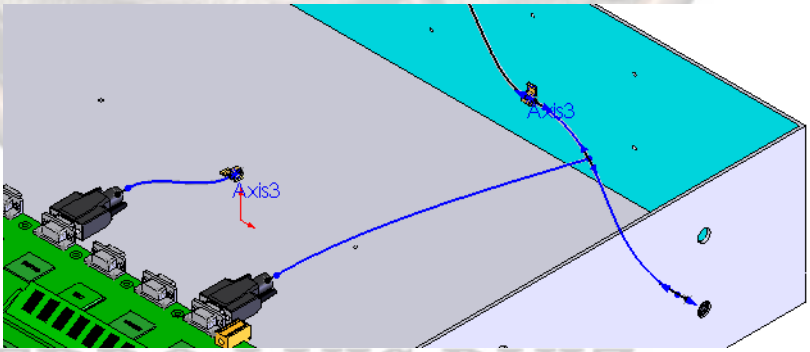
29 Route the other DB9 with clip.

Select the stub endpoint of the J3 DB9. From the Design Library, select richco_hurc-4-01-clip and drag it into the assembly.



Drag it over the hole in the bottom of the enclosure that's just behind the connector.

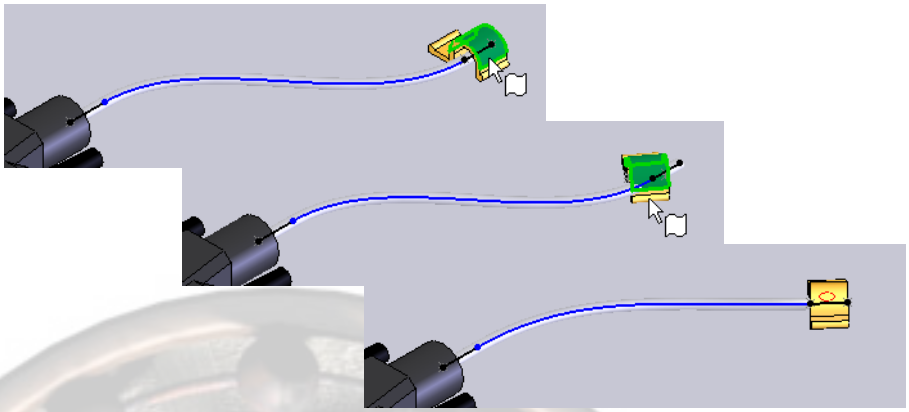
Drop it; the cable Auto Routes from the connector through the clip. **Close** the **Insert Components** dialog.



30 Orient the clip.

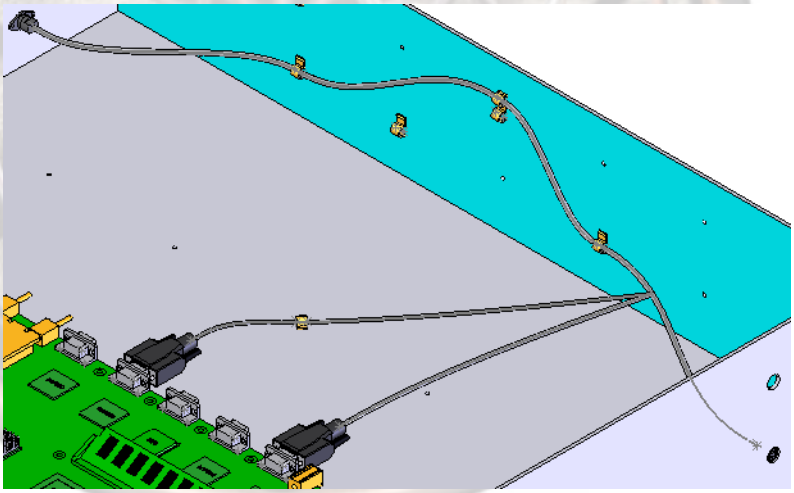
It is important to note that, after dropping a clip from the Design Library, you can reorient the clip as desired from within the sketch. If necessary, zoom the display in to the clip, and drag it to the desired

orientation. The route will update after a move.



Additionally, if a later **Add Wires** operation changes the size of the route beyond the capacity of the clip, properly size-configured clips will update accordingly.

31 Complete the route.
Complete the route by using **Auto Route** to route from the clip segment to the split point.



Edit Wires

After you define paths between connectors in the route sub-assembly, you can associate cable/wire data with the paths. The route diameter updates to reflect the diameter of the cables or wires you select for each path.
Edit Wires is necessary to insure that the From-To and wire sizing information is applied to the harness appropriately.

Where to find it:

While editing a route,

- From the Electrical toolbar, click **Edit Wires** ,
- From the main menu, select **Electrical, Edit Wires...**.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

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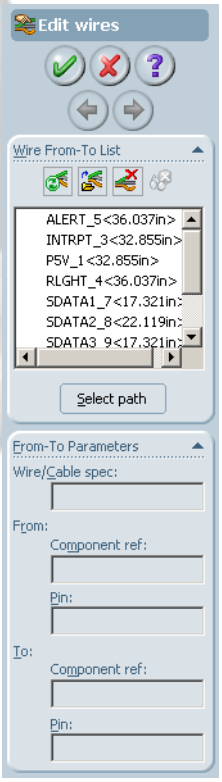
32 Edit Wires.
While still editing the route, click on **Edit Wires** .

Here you can add or delete wires from the harness, or reorder the segments.



- To add a wire, Click **Add Wire** . From the library, select a wire specification, Click **Add**, and **OK**. The new wire will appear as magenta in the list, indicating that the path is not defined. Next, select the wire and click **Select path** to specify which segments of the harness should carry this new wire.
- To delete, select a wire from the list and click **Delete Wire** .
- From-To Parameters will appear in the lower section when a wire is selected.

33 OK.
Clicking **OK** in the **Edit Wires** dialog does two things:

- Incorporates all changes made in this dialog into the route.
- Even if no other changes were made, **OK** will update all harness data per the From-To and cable/wire lists. This is necessary to resize all sections of the harness accordingly. Prior to this step, the harness sections were sized according to the CPoints on the connectors. When you click **OK**, you may see a subtle change in the section sizes.

34 Exit and Save.
Exit the sketch and **Save** the cable part.

Electrical Attributes

Electrical Attributes displays relevant information about a particular harness segment, including the name and length of each wire in that segment.

Where to find it:

- While editing a route, right-click a harness segment and choose **Electrical Attributes**.

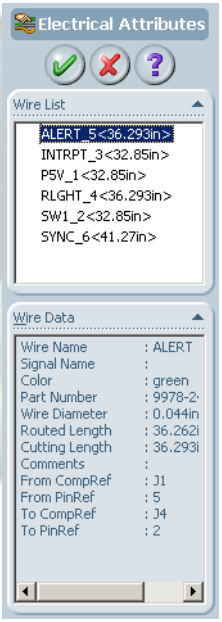
See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

35 Electrical Attributes.
Right-click on a segment of the route and select **Electrical Attributes**.

36 Wire Data.
Click on each of the wires in the **Wire List** in turn to see the **Wire Data** for that wire.

Click **OK**.

37 Save the top level assembly.
It is important to save the assembly before proceeding to Harness Drawing creation in order to insure propagation of all data, new and modified.



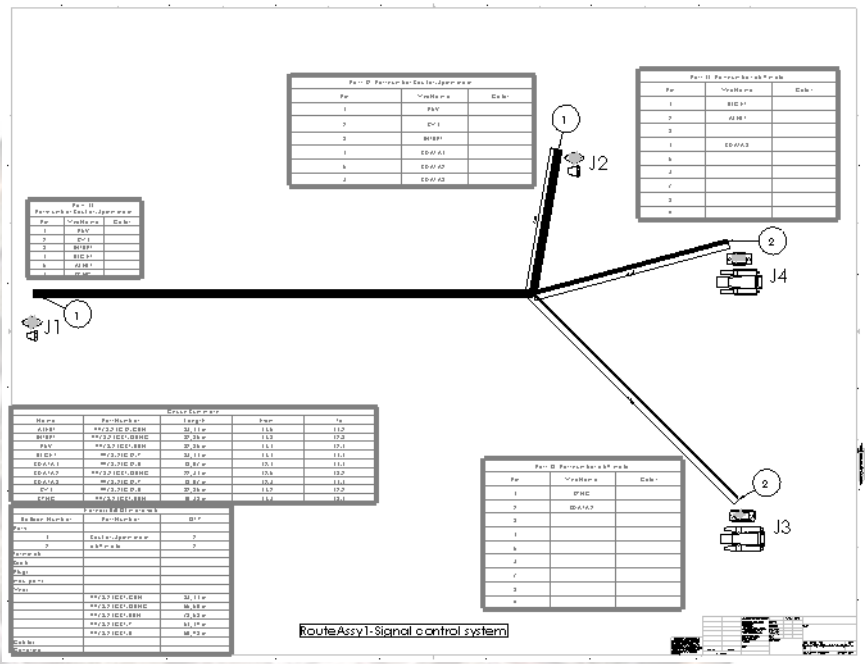
Harness Flattening and Detailing

Harness flattening and detailing functionality is used to develop 2D harness manufacturing drawings from 3D electrical route assemblies.

In SolidWorks, the 2D drawing is generated directly from the 3D route sub-assembly.

Note

Prior to *creating* or *updating* a Harness Drawing, it is important to **Save** the route assembly to be sure all changes are propagated.



Connector Views

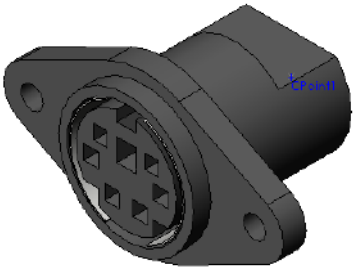
You can generate various 2D connector views for use in your Harness Drawing. For every unique connector in the harness, A SolidWorks block, generated from a drawing view of the connector, is inserted into the Harness Drawing near the segment endpoint where the connector is located.

The block is generated and inserted automatically into the Harness Drawing at creation or upon update if a SolidWorks drawing with a name that matches the connector file name exists in the folder `\\SolidWorks\data\design library\routing\electrical\harnessing\packages`.

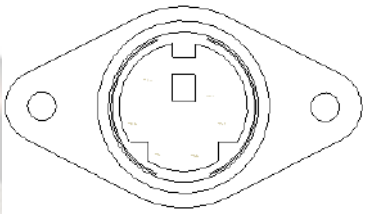
The source drawing need only have 2D geometry that has been converted from a SolidWorks drawing view or from another CAD file such as DXF, DWG, etc.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

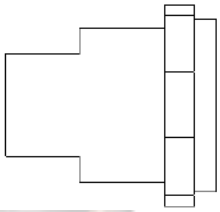
1 **Open part `J1 (socket-6pindinmin)` in its own window.**
The part is a relatively simple mechanical representation of the connector. It does not have any true internal geometry, such as electrical contacts. It does, however, have a mate reference and an electrical CPoint. **Close** the part.



2 **Open the block source drawing.**
In the directory `\\training design library\routing\electrical\harnessing\packages`, open the drawing file `socket-6pindinmin.slddrw`.



This drawing file contains simple geometry that will represent the connector when imported into our Harness Drawing in the form of a block.



The block can be customized with notes or other text, or with additional geometry.

Notice the text within the central arc, specifying pins 1 through 6. These notes determine the placement of wire color graphics, which display the colors of wires in connector pins or cavities in the harness.

Note

Drawing *views* themselves will not be brought into the Harness Drawing. All graphical data must be converted to lines and arcs using sketch tools.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information on connector blocks.

Close the file.

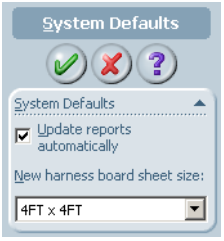
Harness Drawing

The Harness Drawing takes the 3D mechanical data of the route sub-assembly and applies it to a 2D harness manufacturing drawing, flattened and with detail.

See **SolidWorks Routing Help Topics** in the **Help** menu for more information.

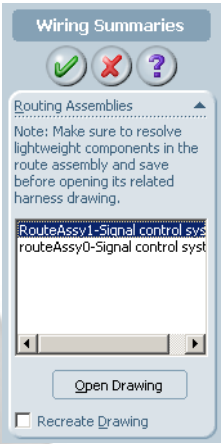
3 **Harness Drawing size.**
If not already, return to the main assembly, `Signal Control System`.

From the **Wiring** menu, select **System Defaults...**. Here we select the size of our Harness Drawing. Be sure **Update reports automatically** is checked, and choose **4FT x 4FT** from the list, then click **OK**.

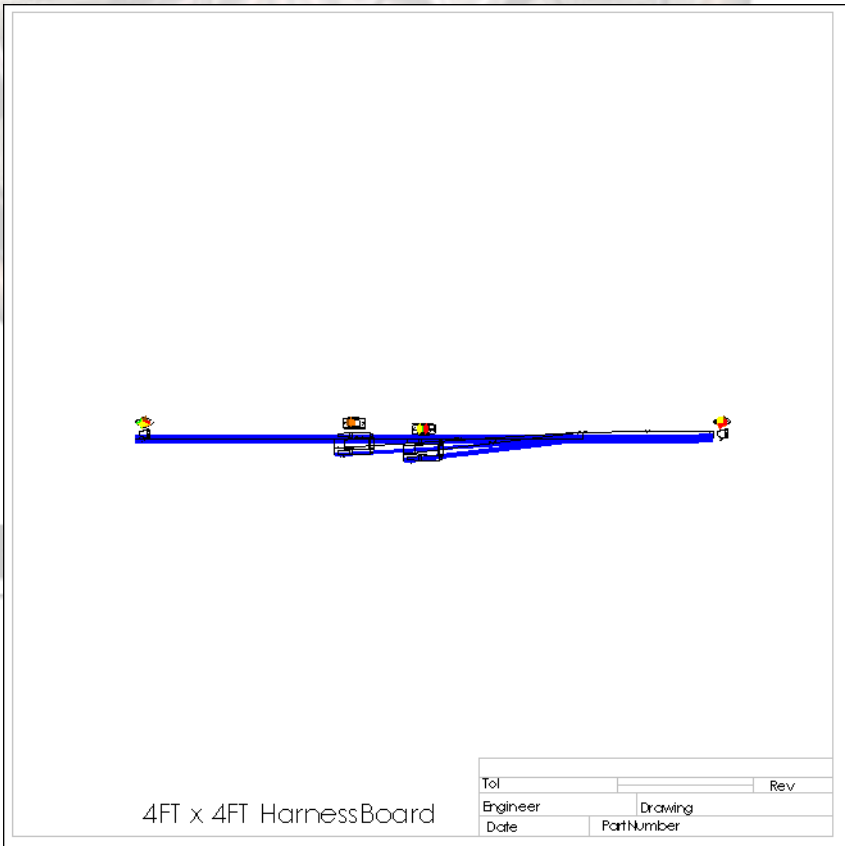


4 Create the drawing.

From the **Wiring** menu, select **Wiring Summary...**. The list displays both route sub-assemblies in this assembly. Select `RouteAssy1-Signal Control System`, and click **Open Drawing**.



A new drawing file, `RouteAssy1-Signal control system-1.SLDDRW`, is created and saved automatically.



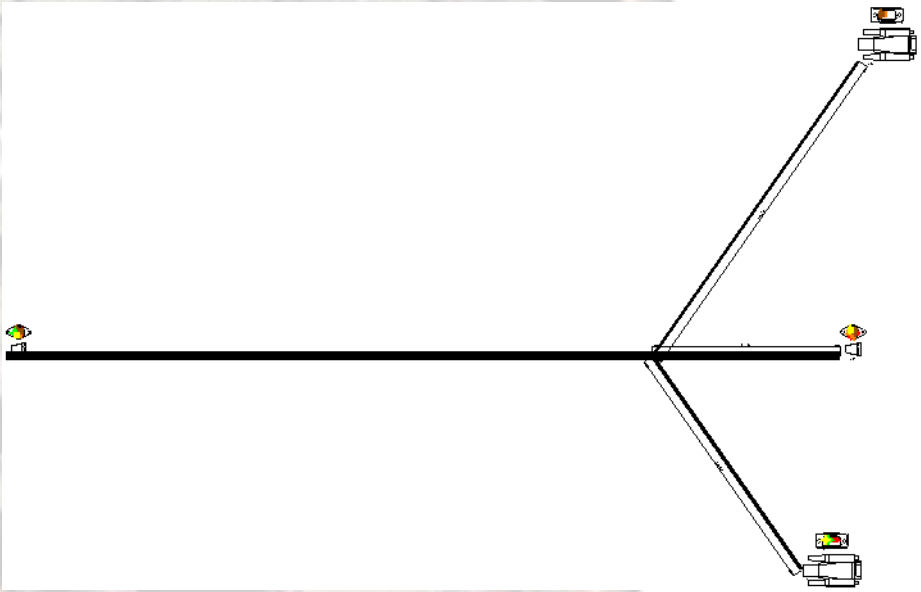
5 Arrange segments and add relations.

The Harness appears in a default orientation. Arrange the segments to your liking or for visibility or logical manufacture.

All segments are sketch lines that can be given relations and constraints:

- First, add a **Horizontal** relation to the long segment on the left.
- Next, switch on the **Point** selection filter, and **Fix** the 4-way junction point.

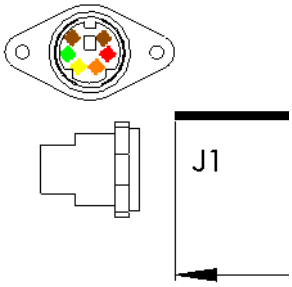
- Finally, spread the branches and lock them in place using either **Fix** constraints or dimensions.



6 The connector blocks.

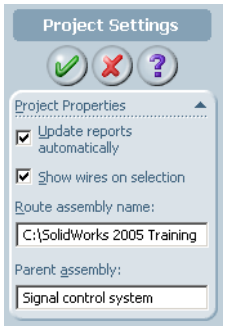
Zoom the display in on one of the block images. The block can be moved or rotated as desired.

The connector block also displays pin / wire data, in color.

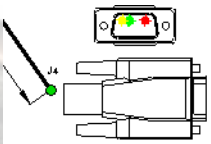


7 Show wires on selection.

Select **Wiring**, **Project**, **Properties...**, and check **Show wires on selection**, then click **OK**.



Selection of endpoints will then display what wires are present at that connector endpoint.



J4 Wires:
ALERT
RLGHT
SDATA1
SDATA3

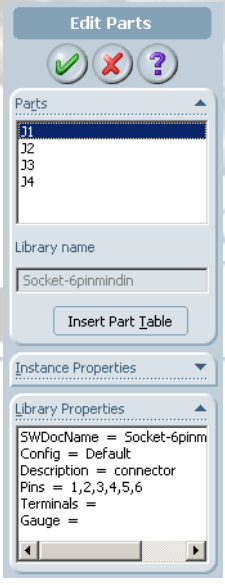
Other Data

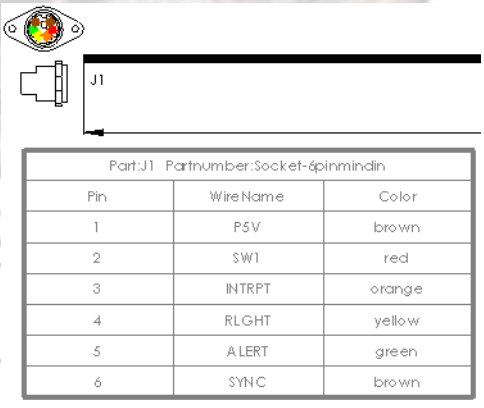
Other vital manufacturing data that can be added to the Harness Drawing include Part Tables, a BOM, and Circuit Summary. Additional components, including coverings and terminals, may also be added to the Harness.

8 Insert Part Tables.

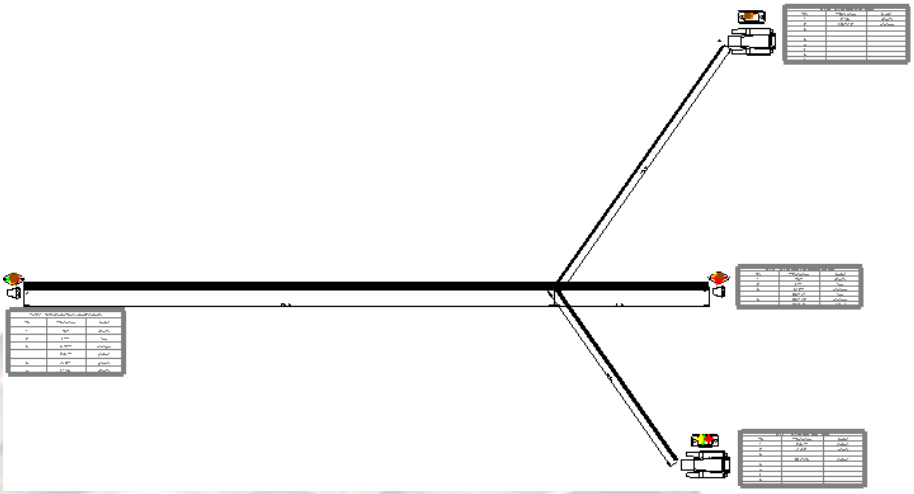
Select **Wiring**, **Edit**, **Parts...**, or click **Edit Parts** on the Wiring Toolbar.

Select `J1` from the list, and click **Insert Part Table**.





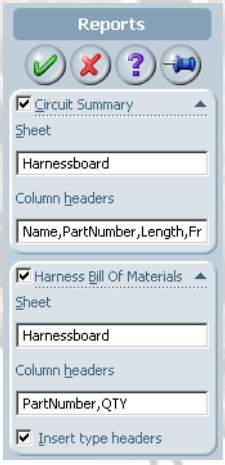
You can add Part Tables for the other connectors in the assembly as well, and reposition them as needed.



9 Add Wiring Reports.

Wiring Reports include Bill of Materials (BOM) and Circuit Summary tables.

Select **Wiring**, **Reports...**. Check both the **Circuit Summary** and **Harness Bill Of Materials** check boxes. Click **OK**.



Harness Bill Of Materials

PartNumber	QTY
Parts	
9ocket-6pinmindin	2
db9 male	2
Terminals	
Seals	
Plugs	
Misc parts	
Wires	
9978-20 GST-GRN	26.50 in
9978-20 GST-ORNG	55.97 in
9978-20 GST-BRN	74.02 in
9978-20 GST-Y	54.94 in
9978-20 GST-R	52.06 in
Cables	
Covering	

Circuit Summary

Name	PartNumber	Length	From	To
ALERT	9978-20 GST-GRN	26.50 in	J1-5	J4-2
INTRPT	9978-20 GST-ORNG	33.61 in	J1-3	J2-3
PSV	9978-20 GST-BRN	33.61 in	J1-1	J2-1
RLGHT	9978-20 GST-Y	26.50 in	J1-4	J4-1
SDATA1	9978-20 GST-R	18.45 in	J2-4	J4-4
SDATA2	9978-20 GST-ORNG	22.26 in	J2-5	J3-2
SDATA3	9978-20 GST-Y	18.45 in	J2-6	J4-4
SWI	9978-20 GST-R	33.61 in	J1-2	J2-2
SYNC	9978-20 GST-BRN	40.41 in	J1-6	J3-1

The tables are added in the default (lower left) location, but can be repositioned or resized as desired.

10 Other parts in the Harness.

Various other components may be required to complete this Harness, including, but not limited to, protective coverings, contact terminals, or seals. When added, the Harness BOM will update.

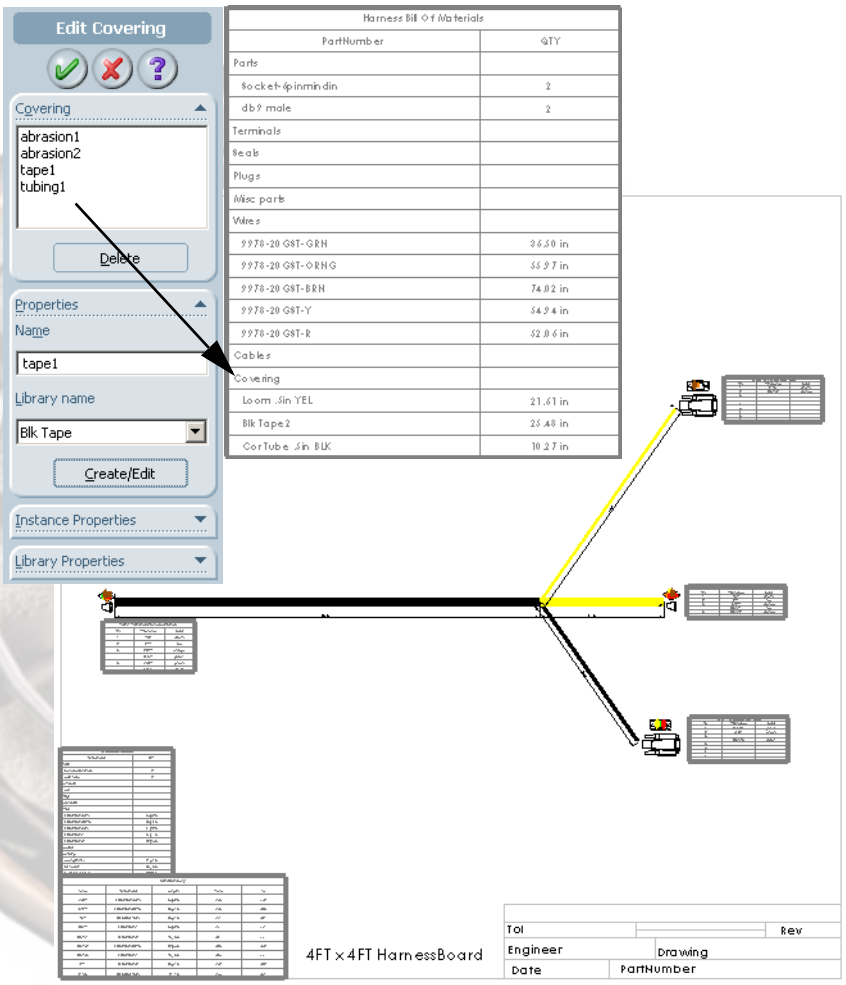
See **SolidWorks Routing Help Topics** in the **Help** menu for more information on the Harnessing Libraries.

11 Add Covering.

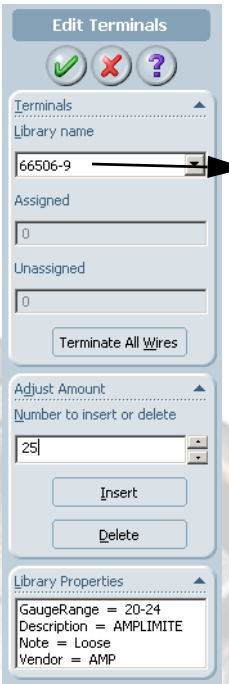
Select **Wiring**, **Edit**, **Covering...**.

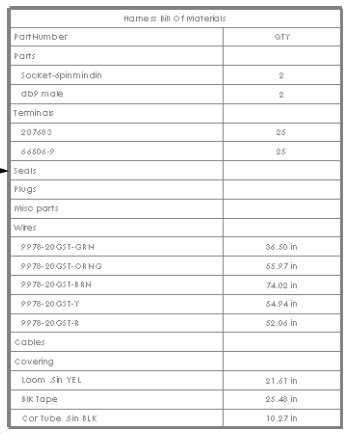
From the **Library name** list, let's add `Cor Tube .5in BLK` and give it the custom **Name** `tubing1`. Select the short segment to the `J4 DB9` connector, and click **Create/Edit**.

Add other coverings as you see fit. Coverings with colors will change the color of the Harness segment.



12 Add Terminals.
Select **Wiring**, **Edit**, **Terminals...**.
It is typical to add many more terminals to the assembly than is needed to account for drops, losses and breakage. Several terminal types are supplied in the Terminals Library.





Lets add 25 of `66506-9` and 25 of `207683`. Enter **25** in **Number to insert**, and click **Insert**. 25 appear as **Unassigned**. Repeat for the other terminal.

Click **OK**.

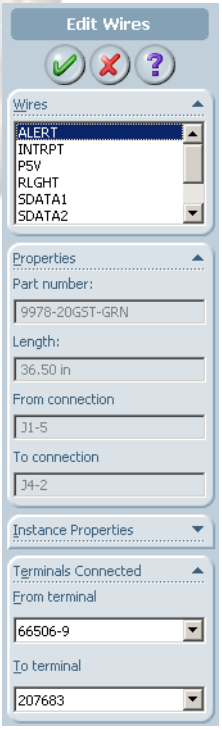
The BOM updates accordingly.

13 Edit Wires.
By editing the wires in the Harness, we can assign terminals to the wire ends.
Select **Wiring**, **Edit**, **Wires...**, or click **Edit Wires** on the Wiring Toolbar.

14 Assign terminals.
Let's assume the `6 pin` (`J1` and `J2`) connectors require `66506-9` terminals, and the `DB9 (J3` and `j4)` connectors require `207683` terminals.

As you select each wire in the list, you can see its **From-** and **To-** connection info. Under **Terminals connected**, add the appropriate terminal to each.

When complete, click **OK**.



15 Edit Terminals dialog.
Return to **Edit Terminals**. The dialog displays some of the terminals as **Assigned**. Close the dialog.

Save the drawing.

Editing Libraries

Libraries of Harnessing parts can be edited from within SolidWorks.

Where to find it:

Select **Wiring**, **Library**, **Edit...**,

or click **Edit Library** on the Wiring Toolbar.

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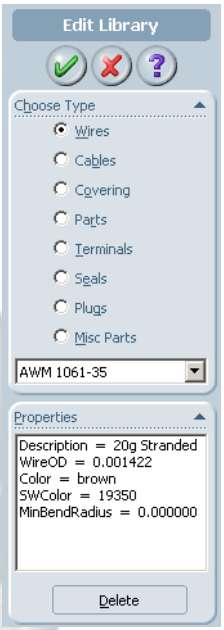
16 Edit Library.

From within this dialog you can **Edit**, **Create** or **Delete** entries in any of the Harnessing component library .xml files.

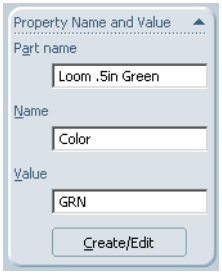


17 Existing covering.

Let's look at wire coverings. Under **Choose Type**, select **Covering**.
From the list, find `Loom .5in YEL`.
Under **Properties**, we can see the properties for this covering.

18 Create new.

Under **Property Name and Value**, change **Part name** to `Loom .5 in Green`.
Under **Name**, type in **Color**.
Under **Value**, type in **GRN**.



Click **Create/Edit**.

This new green loom covering will now be available for future use.

Add other property names and values as desired to complete the description.

Click **OK**, **Save** and **Close** the drawing.

3D STUDIOS
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