SolidWorks®

Piping

SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA

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Piping Basics

Lesson 1
Piping Basics

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

- Activate SolidWorks Piping
- Set Piping Options
- Understand the required features of piping parts
- Understand the process of creating a piping sub-assembly

Introduction

SolidWorks Piping product brings the timesaving performance of SolidWorks to the task of designing piping systems. Built-in capability includes the ability to assemble piping networks built of standard content (from SolidWorks, fitting vendors or other suppliers) along with customer-developed content.

SolidWorks Piping was developed to be able to design systems requiring a wide range of piping technologies including butt-welded, socket connected, flanged, and thread-connected systems. In all cases, the fundamental capabilities of SolidWorks are maintained — including building on the 3D sketcher and the power of configurations to drive variations on the necessary fittings and parts. In fact, SolidWorks Piping only adds two more toolbar icons to the standard assortment provided by SolidWorks.

This guide provides detailed examples for developing pipe routes constructed of welded pipe and tubing components. These examples are included on the CD that accompanies this book.

A piping sub-assembly is always a top-level assembly component. When you insert certain components into an assembly, a piping sub-assembly is created for you automatically. Unlike other types of sub-assemblies, you do not create a piping assembly in its own window, and then insert it as a component in the higher-level assembly.

You model the path of the pipe by creating a 3D sketch of the pipe centerline. The software uses the centerline definition to generate the pipe and elbow components for the route.

The software makes extensive use of design tables to create and modify the configurations of piping components. The configurations are distinguished by different dimensions and properties.

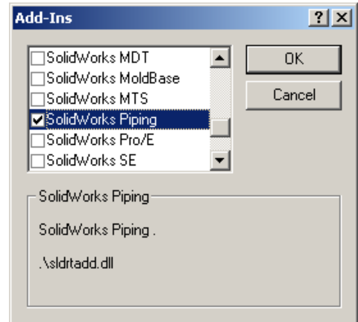
A pipe part contains a configuration for each type and size of raw stock. As you create and edit the route, a new configuration is generated automatically for each unique cut length of the selected stock. The configurations are saved in a new pipe part; the original pipe part in the library folder is not changed.

Where there are bends in the path, elbows are added automatically. You specify a default elbow fitting to be used at each bend in the route.

You can add various types of fittings to the route, such as flanges, tees, reducers, and so forth.

Activating SolidWorks Piping

Before beginning, make sure that SolidWorks Piping is active.



1 **Adding Piping.**
Click **Tools**, **Add-Ins**.

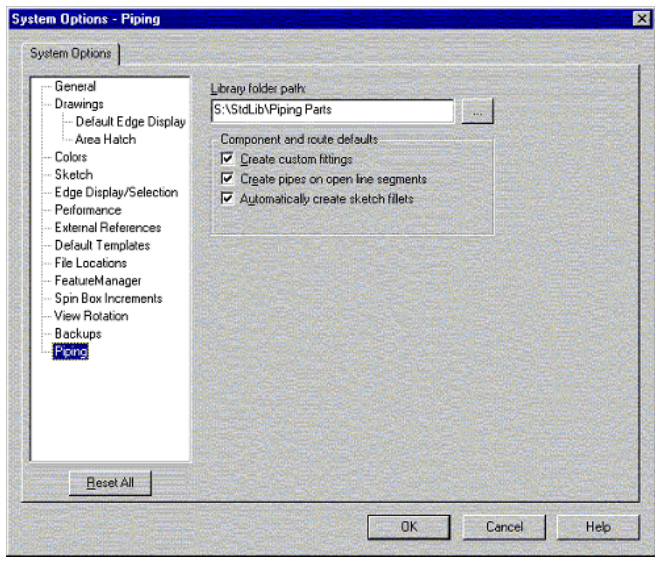
The **Add-Ins** window appears.

2 **Select SolidWorks Piping**
From the **Add-Ins** list, select **SolidWorks Piping**.

3 **Click OK.**
Click **OK** to confirm the selection of SolidWorks Piping.

Specifying Piping Options

Before you begin piping, set your piping options that will apply to all routes that you create.



1 **Options Window.**
Click **Tools**, **Options**, **Piping**.

The **System Options** window appears.

2 **Library Folder Path.**
Specify the **Library folder path** where piping components are stored. For example, `C:\StdLib\Piping Parts`.

When you specify a **Library folder path**, this path is automatically included in the folders list for palette parts (under **Tools**, **Options**, **External References**). This allows you to add fittings by dragging them from the **Feature Palette** window.

3 **Custom Fittings.**
Click **Create custom fittings**.

This automatically creates custom configurations of the default elbow fitting. These are created only in cases where a standard elbow configuration can be cut down to create a custom elbow. Some special cases may not be able to automatically create custom fittings.

4 **Pipes On Open Line Segments.**
Click **Create pipes on open line segments**.

This allows pipe to be created for 3D sketch segments that are connected to a fitting at only one end. For example, if you have not yet added a flange at the end of the route, the last segment of the sketch is an open pipe segment.

5 **Create Sketch Fillets.**
Click **Automatically create sketch fillets**.

This automatically adds fillets at intersections as you sketch. The fillet radius is based on the BendRadius@ElbowArc dimension in the elbow part that you select for the route. This option applies only to 3D sketches that are used as paths of routing assemblies.

6 **Confirm Selections.**
Click **OK** to confirm the **System Option** settings.

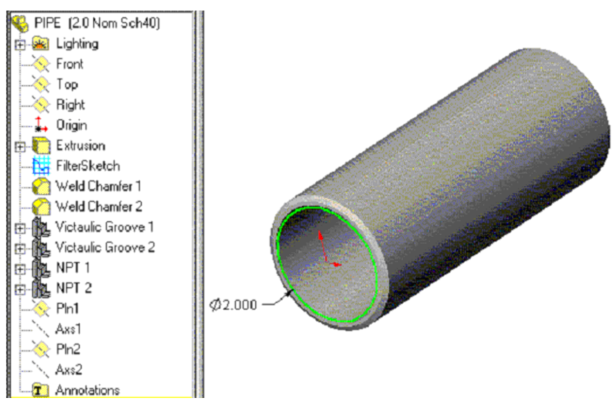
Required Features of Piping Parts

These requirements address typical parts. For more information see the SolidWorks Piping online help.

The software identifies parts as potential piping components when it detects the presence of certain items. Some properties are added automatically to such parts. These properties, named `Component type` and `Fabricated pipe ports`, are listed on the **Custom** tab of the **Summary Information** dialog box (click **File**, **Properties** to see them). The properties are not configuration-specific. Do **not** delete these properties or edit the values that are assigned to them.

Pipes

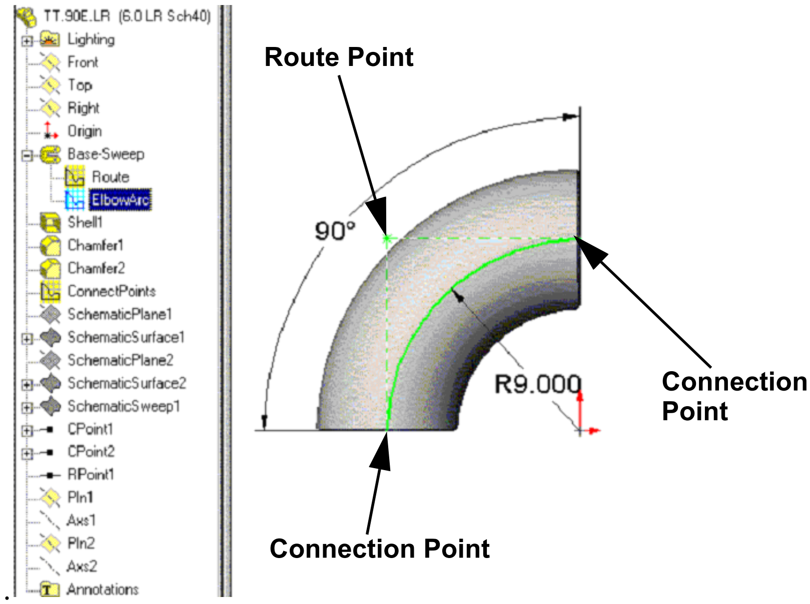
In a pipe part, each type and size of raw stock is represented by a configuration. In the piping sub-assembly, the individual segments are configurations of the pipe part that are based on the nominal diameter, the pipe identifier, and the cut length. Also, there are end conditions built into either end of the pipe. These features can be suppressed, or unsuppressed when needed in the pipe route. Configurations keep you from having to recreate those features each time that you make a pipe. The illustration shows an example of a standard 2-inch nominal pipe part from the Feature Palette.



Route Points

A point in a fitting that is used to position the fitting at an intersection or endpoint in the 3D sketch (or route).

A route point is located at the apex of the bend radius.



Connection Points

A connection point is a point in a fitting where a pipe segment begins or ends. A pipe segment can only be generated when at least one end is attached to a connection point. Each fitting must contain a connection point for each port, positioned where you want the adjoining pipe to begin or end.

An arrow indicates the direction that the pipe exits the fitting.

Elbows

An elbow is automatically inserted wherever there is a fillet in the 3D sketch. You can also add elbows manually. An elbow can contain configurations for many different types (schedules) and sizes of elbow.

Connection points are located at each end where the pipe exits the elbow.

Flanges

Flanges are often used at the ends of a piping run to connect the pipe to a fixed component — such as a pump or tank. Flanges can also be used to connect long straight sections of pipe. A flange is what is used in most cases to start the piping route by dropping it into the top-level assembly from the Feature Palette.

A flange must have one connection point. The connection point must be concentric with a circular edge of the fitting. The pipe ends at the connection point. If the pipe extends into the flange (such as socket or threaded connections), the connection point must be at the correct depth inside the fitting.

Reducers

Reducers change the diameter of the pipe at a selected location. A reducer (whether concentric or eccentric) has two connection points, one at each end, with different values for the diameter.

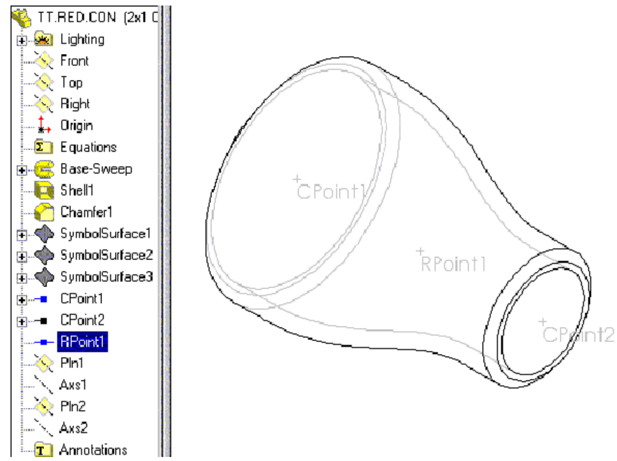
Note

The following is focused on butt-welded reducers or reducing couplings. In the case of socket or threaded-based versions. Reducing

Bushings are handled as combinations of half-pipe and half-fitting when they are constructed in SolidWorks Piping.

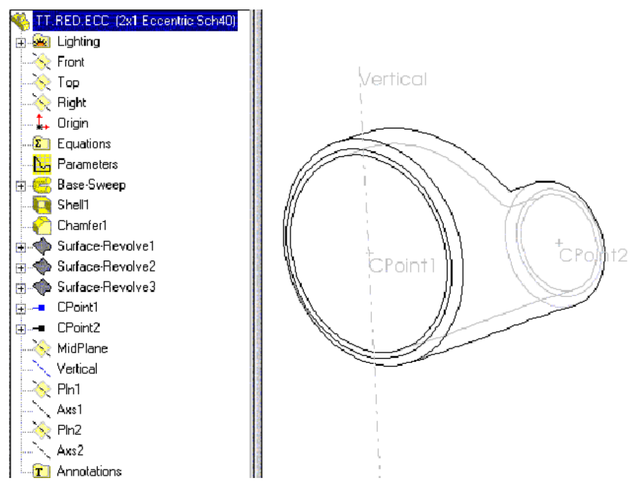
Concentric Reducers

A concentric reducer includes an optional route point between the connection points that is used to insert (drag and drop from the Feature Palette) it at a point in a 3D sketch segment.



Eccentric Reducers

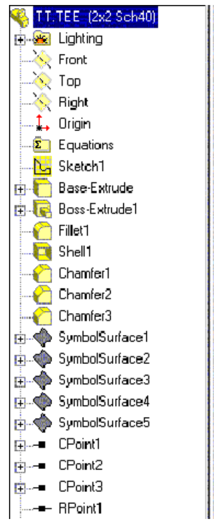
Eccentric reducers do not have route points. You must insert (drag and drop from the Feature Palette) eccentric reducers at the end of a sketch segment.

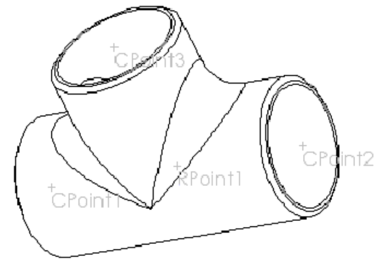


An axis is created as a feature in an eccentric reducer to control the angular orientation of the reducer in a piping sub-assembly. This axis must be named `Vertical`, and the axis must be perpendicular to the route passing through the reducer. When you insert the reducer into a piping sub-assembly, the `Vertical` axis is aligned to the first construction line drawn in the 3D sketch at the insertion point.

Other Fittings and Parts

You can also insert tees, crosses, and other multi-ported fittings at intersections in the 3D sketch.

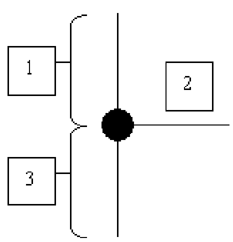




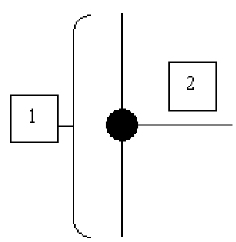
Fittings with multiple branches must have a connection point at each port and a route point at the intersection of the branches. For example, in the tee below, it has three connection points and one route point. When you insert the fitting into the 3D sketch, the route point is placed coincident with the intersection in the 3D sketch.

Intersections

In the 3D sketch, the straight run for a tee must consist of two separate segments — not one continuous segment — so that an intersection exists for the perpendicular section of the tee.



Correct



Not Correct

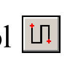
Fittings need one connection point at each port and one route point at the intersection of branches. This is the point where the fitting will drop into the Route or 3D sketch.

Creating a Piping Sub-Assembly

When you drag a flange into an assembly, a new sub-assembly is created and the 3D sketch is started automatically.

You can also begin the 3D sketch manually if the fittings are already in the assembly. In this case, right-click the connection point in a fitting and select **Start Route**.

The following are the general steps that are used to create a piping sub-assembly:

- Sketch the pipe path using the **Route Line** tool .
- If you selected the **Automatically create sketch fillets** option, fillets are added at corners. The default radius of the fillet is defined

by the bend radius of the elbow you select when you begin the route. To add sketch fillets manually, use the **Sketch Fillet** tool.

- Fully define the 3D sketch by adding dimensions and most types of relations using the same methods as you use in 2D sketching.
- Exit the 3D sketch. The software generates the pipe segments, inserts the elbows, and connects all the components into a continuous pipe assembly.

Each component in the sub-assembly is parametrically related to the 3D sketch. If you change the sketch, the pipes and fittings are updated.

You do not need mates or other relations between the components of the sub-assembly (route). The sizes and positions are driven by the 3D sketch.

Creating a Route

Now that you have reviewed the basics of routing, you are ready to route your pipe. Let's try creating a small route.

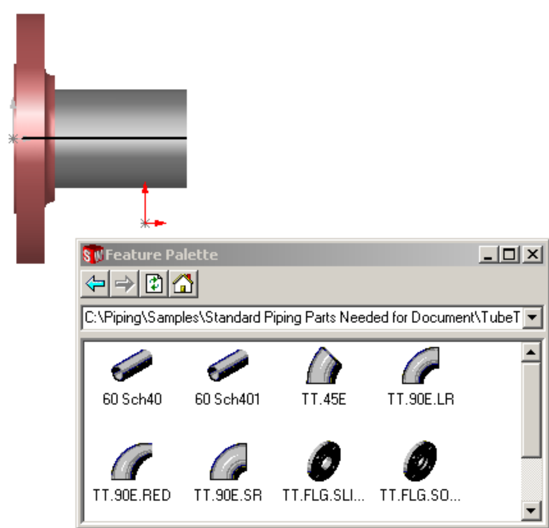
1 **Open an assembly.**
Open a new assembly in which to route. Save the assembly and give it a name.

2 **3D sketch.**
Click to begin creating a 3D sketch.

3 **Activate the Feature Palette.**
Click **Tools**, **Feature Palette** to activate the Feature Palette. The Feature Palette is a repository of components that you can drag into your piping routes.

4 **Place a flange.**
Browse through the Feature Palette and find a flange. For this exercise, use any flange.

Drag and drop the flange into the assembly.



The flange appears with a small section of pipe. The length of the pipe is not critical at this time. However, to change the length of the pipe, click and drag the end of the line.

5 **Add fittings.**
Add fittings as needed, by dragging the parts from the Feature Palette to end points or split points of the 3D sketch. The fittings snap to the

end of the route as needed.

Later you will learn how to orient the fittings. For now, don't worry about it.

6 **Add to Route.**
You add fittings to pipes and pipes to fittings. After you have placed a fitting — like an elbow — you can add pipe to that fitting to continue the route.

Select a CPoint (connection point) at the end of a fitting and right-click. Click **Add to Route**. Another section of pipe is placed on the fitting.

7 **Practice routing.**
Try to place some fittings and pipes in this way. Don't worry about the structure of the route. This one is just for practice.

8 **Exit the sketch.**
Exit the 3D sketch.

When you exit the 3D sketch, the pipe segments and elbows are inserted and all the components are connected into a continuous pipe assembly.

Each component in the sub-assembly is parametrically related to the 3D sketch. If you change the sketch, the pipes and fittings are updated automatically. You do not need mates or other relations between the components of the sub-assembly, because the sizes and positions are driven by the 3D sketch.

Editing a Piping Sub-Assembly

You may begin a route by simply sketching between the furthest two flanges or fittings and add the other components later. To edit a route:

- Right-click on the route feature and click **Edit Route**.
- Add fittings (tees, reducers, flanges, and so forth) while editing the 3D sketch.
- Perform the **Penetrate** command.
- Create a custom pipe configuration for selected segments.

In general, you should edit piping sub-assemblies in the context of the top-level assembly.

Adding Fittings

Elbows are added automatically at fillets in the 3D sketch. But all other types of fittings can be added from the Feature Palette onto an endpoint or split point in the 3D sketch.

The **Split Curve** tool  is used to split the 3D segment, exactly where you want the route point of the fitting to be located in the route. Be sure **Display entity points** is selected so that the **Split Curve** can be seen. To turn **Display entity points** on click **Tools**, **Options**, **System Options**, **Sketch**, **Toolbar** and select **Display entity points**.

Split Curve is not used for pipe penetration cases.

Manually Adding a Fitting

Fittings are generally placed by dragging and dropping them from the Feature Palette. Most commonly used fittings reside in the Feature Palette. In addition, you can create fittings and add them to the Feature Palette for future use.

1. Select a fitting.

   While editing the 3D sketch, drag a fitting from the Feature Palette window and point to the location in the 3D sketch where you want to place the fitting.

2. Orient the fitting.

   Press **Tab** to view the possible orientations of the fitting. When the alignment is correct, drop the fitting. The fitting snaps to the end of the route.

3. Fitting configuration.

   Select the proper configuration of the fitting. The configuration default is defined when you begin routing.

4. Accept fitting.

   Click **OK**.

Alternative Method for Adding Fittings

The method described above is the recommended method, but not the only method for adding fittings.

1. Select a point.

   While editing the 3D sketch, right-click on a point in the 3D sketch and click **Add Fitting**.

2. Select a fitting.

   Select the fitting from the **Select a Fitting** dialog box.

3. Drag the fitting.

   Drag the fitting and click and hold the right mouse button until the orientation is correct. Then click the left mouse button.

4. Fitting configuration.

   Select the proper configuration.

5. Accept fitting.

   Click **OK**.

Custom Pipe Configurations

When pipe segments are the same length, a single configuration of the pipe part is used for all instances within the assembly. But, if you add a feature to a selected pipe segment in the route assembly, you can create a custom configuration in the pipe part file.

An example of this would be if you added a hole to a pipe segment (say you were going to weld a coupling to the side of that pipe). But, you didn't want the hole to appear on any of the other pipe segments of the same length in the route assembly (which would all, normally, be the same name because they are the same length).

Whenever pipes are added to the route with different lengths, all the pipes are found in the same part file only in different configurations depending on their lengths. But, when a part has to be customized a bit (for example, have a hole cut in it, change it is end condition, and so forth), then you have to make a custom configuration of that pipe. After that, you can edit the pipe as you would any other pipe. The features that you add will only be show up in that configuration of the pipe part.

Our standard pipe part comes with many optional end conditions at both ends of the pipe. All but the chamfered ends are suppressed in the standard pipe configuration. But if you want another end condition on your custom configuration pipe, just **Unsuppress** the one you want.

1. Select the segment.

   Right-click on the pipe segment from which you want to create a custom configuration and click **Create Custom Pipe Configuration**.

2. Customize the segment.

   Add the hole feature to the selected pipe segment (either in **Edit Part** mode or open the pipe part on its own to make the changes).

Custom Elbow Fittings

If any custom elbow fittings are required (different than that of the default elbow), when you exit the 3D sketch, the filleted sketch segments are highlighted, and the dimensions are displayed for each individual case.

1. Select elbow.

   To select an alternate elbow part, click **No**. Browse to a part that contains a configuration with the necessary dimensions, and click **Open**. Select a configuration, and click **OK**. If the bend radius in the selected elbow is different from the fillet radius in the 3D sketch, the sketch is modified accordingly.

2. Create custom fitting.

   To create a custom elbow fitting, click **Yes**. Select the configuration to use as a basis for the custom configuration (if prompted), and specify a name. The custom elbow is saved as a new part.

Review of Basics

- A piping route assembly is always a top level assembly component, and usually created by dragging and dropping a flange into the top level assembly from the Feature Palette.
- A default pipe size and default elbow are established when the route assembly is started.

- You model the path of the pipe, by creating a 3D sketch of the pipe centerline.
- You can add various fittings to the route by dragging them from the Feature Palette to vertices (endpoints or split points) in the 3D sketch.
- Piping parts have some very unique features which include:

  Connection Points: are points in a fitting where a pipe segment begins or ends. A pipe segment can only be generated when at least one end is attached to a connection point. Each fitting part must contain a connection point for each port, positioned where you want the adjoining pipe to begin or end.

  Route Points: are points in a fitting that are used to position the fitting at an endpoint or intersection (split point) in the 3D sketch.

- When you exit the 3D sketch, pipe segments and elbows are generated.
- Each component in the pipe route assembly is parametrically related to the 3D sketch. If you change the sketch, the pipes and fittings are updated automatically.
- You do not need mates or other relations between the components of the sub-assembly, because the sizes and positions are driven by the 3D sketch.

Part Naming of Sub-Assemblies

Part Number	Description
D20000000	Layout
D20000200	route assembly 1 – EX: Suction Manifold
D20000300	Piping Sub Assembly 1 of this Route
D20000200.4IN	4-inch pipe in route assembly 1-NO MATTER What Sub Assembly or Length
TT.FLG.SLIP-ON	Std Lib Part – Unchanged
TT.90E.LR	Std Lib Part – Unchanged
D20000300.001	Custom Cut Elbow – Named for Sub Assembly (dot) Find # in Parts List
D20000300.002	Custom Cut Elbow – Named for Sub Assembly (dot) Find # in Parts List
D20000400	Piping Sub Assembly 2 of this Route
TT.90E.LR	Std Lib Part – Unchanged

Part Number	Description
D20000200.6IN	6-inch pipe in route assembly 1-NO MATTER What Sub Assembly or Length
TT.90E.RED	Std Lib Part – Unchanged
D20000400.001	Custom Cut Elbow – Named for Sub Assembly (dot) Find # in Parts List
D20000400.002	Bracket/mount for Pipe – Named for Sub Assembly (dot) Find # in Parts List
TT.TEE	Std Lib Part – Unchanged

Lesson 2
Tees, Flanges, and Standard Elbows

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

- Start a pipe route assembly by mating the first flange.
- Add a standard tee to the pipe route assembly.
- Add flanges and butterfly valves.
- Butt standard piping parts together.
- Add Victaulic couplings.
- Add other mated flanges to the same route assembly.

Introduction

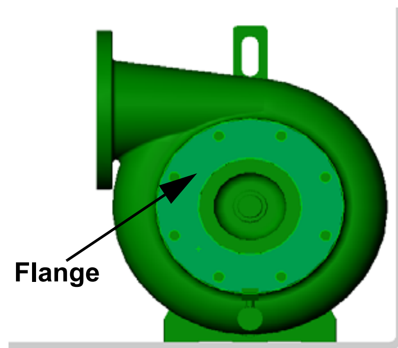
Now that you have been introduced to the basics of the 3D Piping software in SolidWorks, it is time to move into the more specific situation a one may run into while routing pipe. We all know that routing piping is much more than just straight line pipe and simple 90 degree elbows.

Copy the following folders from CD, to your local directory:

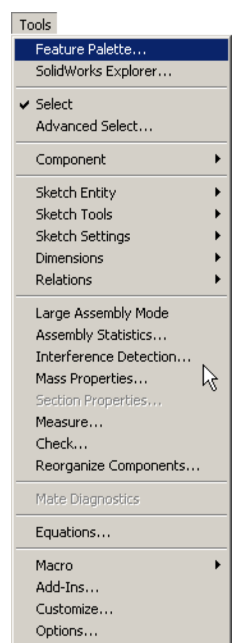
- `3D Piping – Automate Program`
- `3D Piping – Butting Cut Elbows`
- `3D Piping – Offset Pipe Penetration`
- `3D Piping – Routed Tubing`
- `3D Piping Example`

1 Open the assembly.
Open the assembly file `Small Layout 1`, found in `C:\My Documents\Sldworks_Local\3D Piping Example`.

2 Zoom to flange.
Zoom up close on he left hand pump in the assembly. To do this, click , select the flange, and click .



3 Open the Feature Palette
Click **Tools**, **Feature Palette**. The **Feature Palette** window appears.

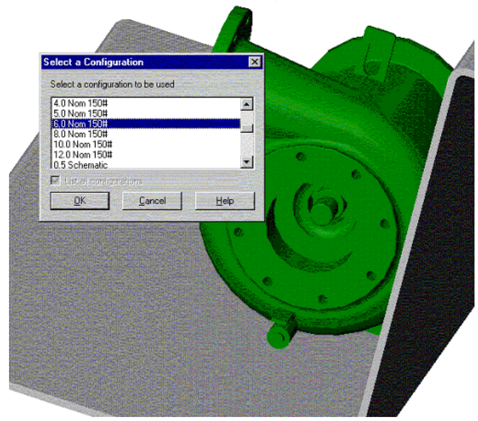


4 Find the flange.
Browse to `C:\StdLib\Piping Parts\TubeTurn`, so that you can see the socket weld flange you will use to start the piping route assembly.

5 Place the flange.
Drag and drop the `TT.FLG.SOC-WELD` flange onto a mating circular edge of the suction side of the pump. Before dropping the flange on the pump, note the graphics shows the flange to be snapping to the face of the pump.

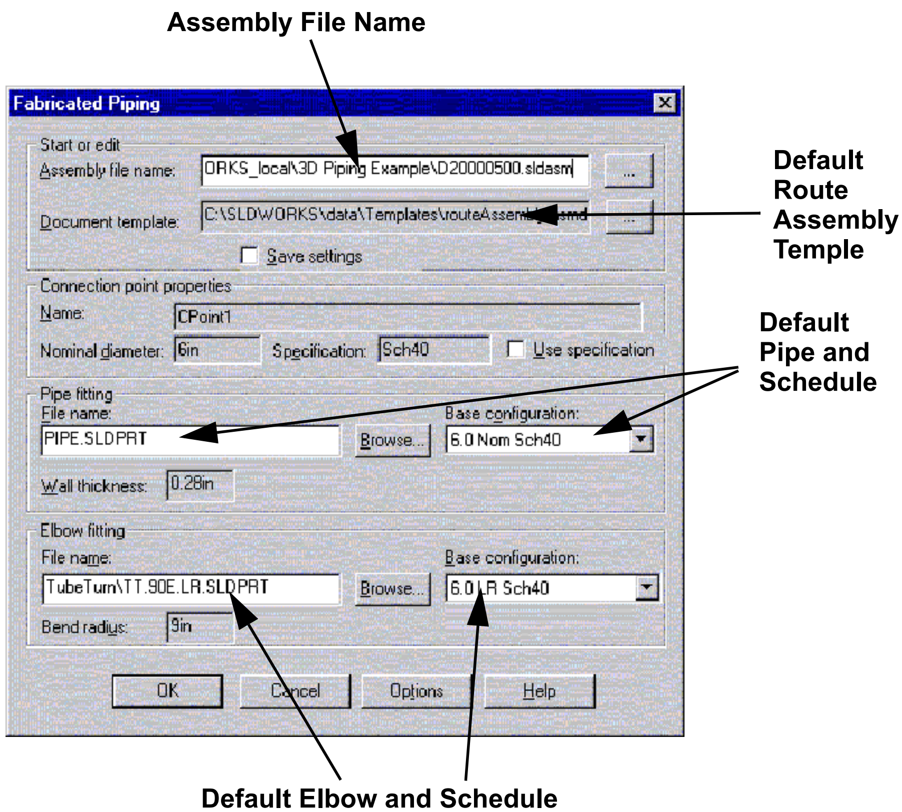
The flange appears as the default size.

6 Select the configuration
the system prompts you to pick the size flange that you need to use. For this example, let's select `6.0 Nom 150#` from the **Select a Configuration** dialog box.



Naming the Route Assembly

Set the properties of the piping route assembly in the **Fabricated Piping** dialog box.



7 File name.
In the **Assembly file name** field, make sure the path to the route assembly is correct.

8 Template.
Select the `routeAssembly Template`. You may have to browse to find this template.

9 Flange size.
The flange size determines the starting pipe size. You need to set the

schedule of the pipe.

Click **Browse** in the **Pipe fitting** section of the dialog box and select the standard 3D pipe in `C:\StdLib\Piping Parts`.

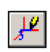
10 Elbow fitting.
Click **Browse** in the **Elbow Fitting** section of the dialog box and select the standard Long Radius 3D piping elbow in `C:\StdLib\Piping Parts\Tube Turn`.

This a standard ell. Later you will see that you can easily use short radius elbows.

11 Accept the settings.
Select **OK**, if all the selections match the screen shot below, to confirm your default pipe route properties.

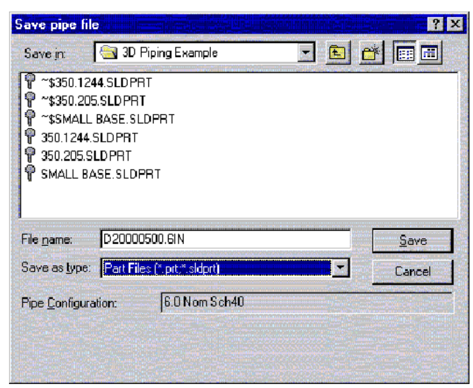
12 Viewing errors.
There will be some error displayed on the two mates (`Mategroup 1` of the route assembly). These errors will be resolved once we exit the 3D sketch.

13 Verify the rotation.
Because of the mate references on the flange, there is only one degree of freedom left for you to mate the flange onto the pump, and that is the rotation of the bolt hole pattern. Ensure that this rotation is correct.

14 Exit the 3D sketch **.**
The system prompts you to give a name to the straight pipe sections that will be in this route assembly (earlier in the route properties, we just set the default pipe size not its name).

Enter "`route assembly D# dot pipe size`" in the **File Name** field. Before you click **Save**, make sure you are in the proper directory.

15 Errors resolve.
As you exit the sketch, notice that the mating errors go away and piping parts that are in your route all appear in FeatureManager design tree.



As you build the pipe route, all 6-inch pipe has the same part name and the different lengths or configurations of that pipe size are appended to the pipe name.

16 Align flange.
Go ahead and get our first flange lined up with the bolt holes of the pump flange. Make sure you add this mate while you are in the Edit route assembly environment. just mate them together the old fashion

way. A **Concentric** relationship between bolt holes of each part will work.

Remember the only reason we can add mates is that the first flange placement is not driven by the route. If it were, it would not be possible to add mates to position it.

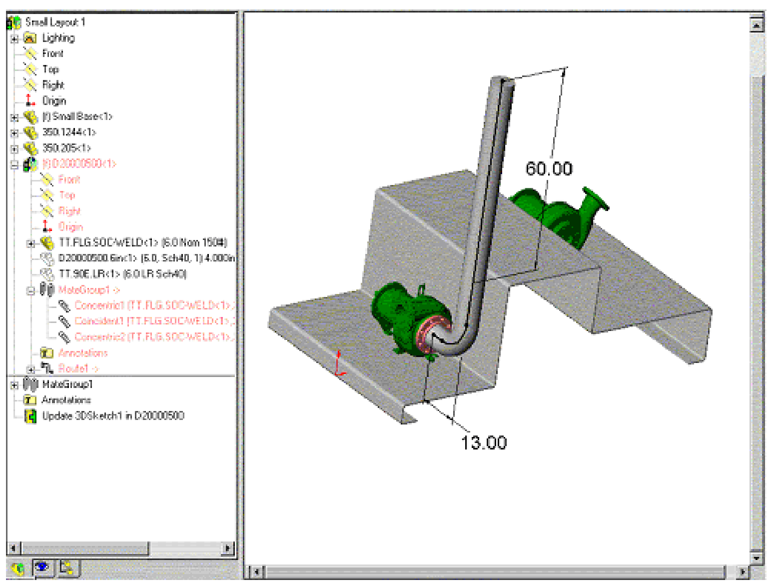
Adding More Pipe

With that done, we are ready to get back into the 3D sketch and route some more pipe.

17 Select the route.
Right-click on `Route1` from the Feature Manager design tree and click **Edit Route**.

18 Dimension the pipe.
Dimension the first straight section of pipe to 13.00.

19 Add pipe.
Click **Line**, click on the open endpoint of that pipe segment and draw a line straight up at 90 degrees from the existing pipe (use your tab key to change the drawing plane axis if you need to). dimension the endpoint of this line 60.00 Higher than the top face of the skid.



Even though we have an elbow and another straight length of pipe in our route at this time, they won't show up as parts in FeatureManager design tree until we exit the route (or 3D sketch).

Adding a Tee

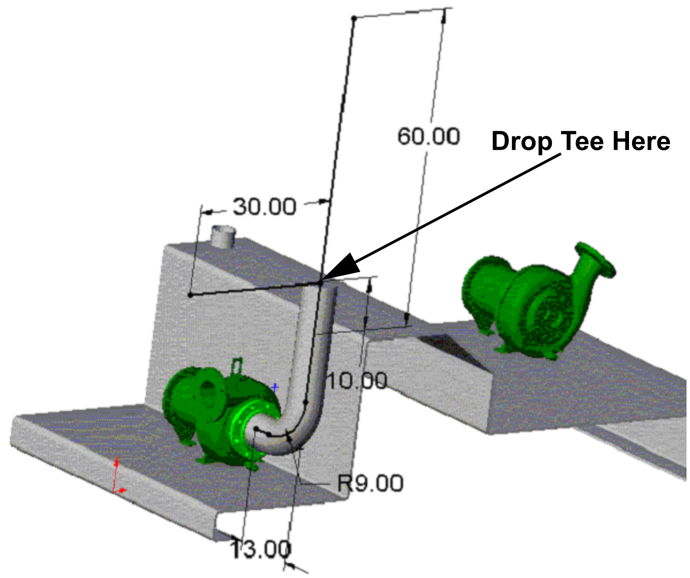
In order to add a tee to an existing route, you must have a split point.

There are two things that need to be added to the 3D sketch in order that a tee can be inserted:

20 A Split Curve  where the branch of the tee will be located.
Add a split point to the vertical pipe that is 10.00 inches above the top face (the center horizontal face) of the skid.

21 A line drawn from that split point, in the direction you want the branch of the tee to go.

From that split point, draw a 30.00 inch horizontal line, that is perpendicular to the vertical pipe run. let's put a weld tee fitting at the intersection we just created.



22 Add the tee.

Open the Feature Palette window and browse to `C:\StdLib\Piping Parts\TubeTurn`.

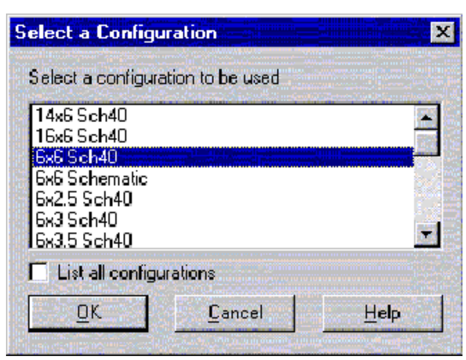
Tip

Pause as you drag the part into the graphics area and watch the tee snap to the proper orientation as you do this.

Drag and drop `TT.TEE` onto the split point you just made.

23 Configure the tee.

Select the `6x6 Sch40` tee from the list and click **OK**.



Notice that we are dimensioning (fully defining) the sketch as we go. If you have approximate dimensional and relational values, it is a good idea to put them in as you go. You can change them later if you need to.

24 Exit the sketch.
Exit the 3D sketch, so you can see all the new parts that have been added to the route assembly.

25 Save your work.
When prompted answer **Yes** to save any models in the assembly.

The 90 degree elbow and the 6-inch tee were placed in the route assembly.

26 Review the configurations.
Notice that there are four straight pieces of pipe in our route assembly and, even though they have different lengths, they are all the same part name. Right-click on one of them to open it. You see that the one pipe part has many configurations (or lengths) that are generated from the 3D sketch.

If you change the lengths of sketch lines, the lengths of these straight pipe configurations would automatically update and change as well.

Butting Standard Parts Together

Such as tees, reducers, and so forth butting elbows together is covered in another lesson.

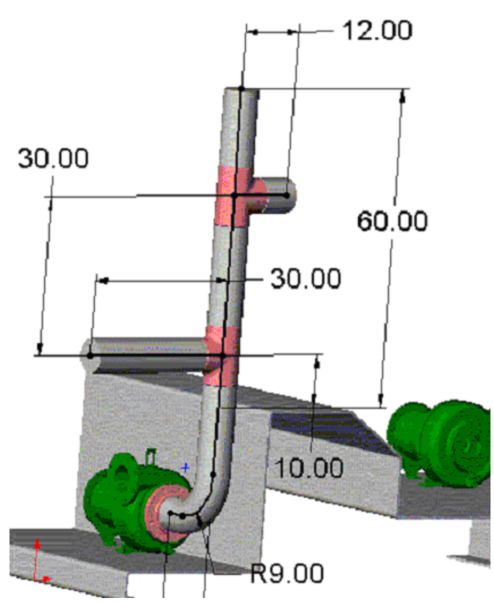
If you know the basic dimensions (face to face, or face of part to virtual sharp of the same part) of the piping parts they want to butt together, then the number of steps in this procedure can be greatly reduced.

Note

SolidWorks 2001 Service Pack 4.0 and later releases have new functionality to simplify the generation of butted fitting arrangements.

1 Select the route.
While editing the upper level assembly (in this case it is called `Small Layout 1`), right-click on `Route 1` and click **Edit Route**. This puts you back in the 3Dsketch.

2 Add pipe.
Add another 6-inch tee to this vertical run of pipe. Point the branch of the next tee in the opposite direction from the first. See the illustration for the dimensions to the split point and orientation of the tee branch.



3 Position the tee.
The pipe between the two tees is 18.75 inches long. Right-click on `Route 1` from the FeatureManager design tree and click **Edit Route**. Change the vertical dimension to 11.25.

4 Exit the 3D sketch.
As you exit the 3D sketch, the two tees are butted together face to face.

There are no minus signs next to any of the parts in the route assembly. All the parts are defined (or positioned) by the 3D sketch and only the original starting flange was mated.

Adding a Butterfly Valve and Flanges

Putting valves and flanges in the route assembly is not complex in itself because it is a matter of drag and dropping from the Feature Palette:

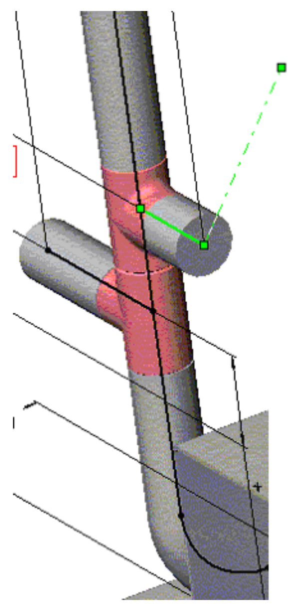
- Each part has an axis named `Vertical`. This axis is used to position the part within the pipe route to its exact rotation.
- At the point where the standard part is dropped onto the route, there must be a centerline. This centerline must be perpendicular to the straight run of pipe (or sketch line).
- The standard part snap its `Vertical` axis collinear to this centerline. If you drag the centerline, the part follows.
- To establish the exact rotation, you may need to insert another centerline. Then relate the new centerline to the original centerline.

1 Select the route.
Right-click on the route and click **Edit Route**.

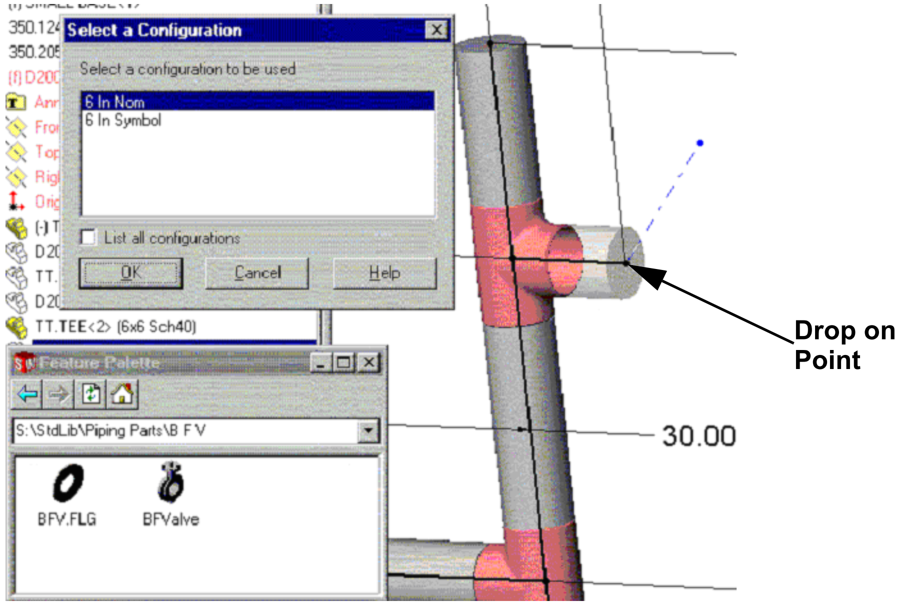
2 Add centerline.
Draw a centerline at the endpoint of the 12-inch line coming out of the upper tee. This line will be used to establish the rotation to the fitting.

3 Add relations to centerline.
Relate this centerline so that it is perpendicular to the straight run of pipe.

It may appear to be already perpendicular, but it is probably not.

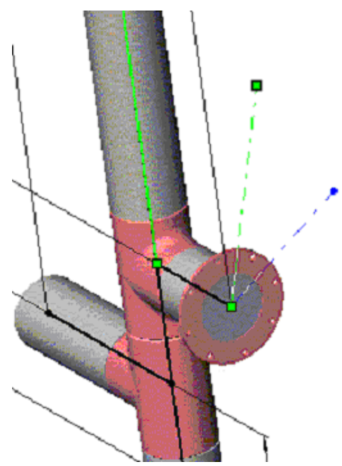


4 Add the butterfly valve.
Open the `BFV` folder (butterfly valve) folder in the Feature Palette. Drag and drop the flange onto the end of the pipe. When prompted, select the proper part configuration from the list.



5 Verify the flange rotation.
Drag the centerline around to see that the flange rotation follows.

6 Add a centerline.
Rotational location of a component is critical to ensure accurate assembly. So, let's ensure an exact rotation on this flange.

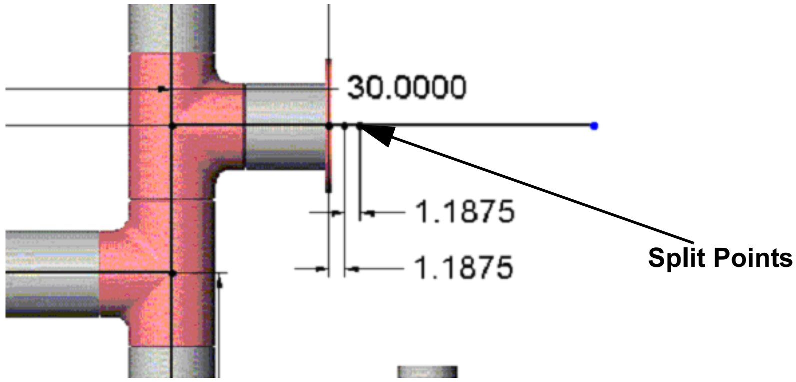


Starting at the same point where the first centerline intersects the straight run, draw a `vertical` centerline (make sure it snaps vertically).

7 Specify the angle between the centerlines.
Put a 35 degree angle between the two centerlines.

8 Add a line.
Draw a horizontal solid sketch line from the end of the pipe run so that it is collinear with the last horizontal run of pipe.

9 Create split points.
Create two split points in the line you just made. Make the first two segments 1.1875 inches long. In this case, this distance is half the width of the valve plus the setback of the pipe in the flange.



The two new split points will be connection points for the valve and the opposite side flange.

10 Add centerlines and add geometric relations.
These parts also need centerlines in order to snap their rotation. Put these lines in and relate them to the original centerline of rotation so that everything lines up properly.

Add centerlines (for rotation) from these new split points and give them a parallel relationship with the first centerline.

11 Add the butterfly valve to the route.
Drag and drop a butterfly valve from the Feature Palette onto the middle point where the center of the valve would go. When prompted, select the proper (6-inch, in this case) size configuration for the valve you need, then click **OK**.

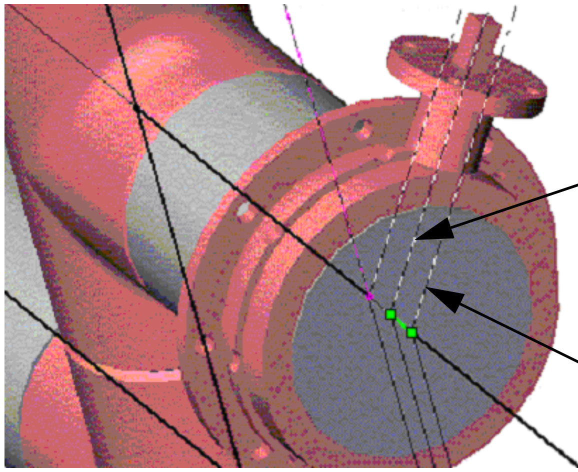
The valve snaps into place. The valves `vertical` axis follows the centerline that is on the hang point where you dropped the valve.

12 Change line properties and delete line.
A warning tells you that the assembly can't be rebuilt properly. That is because we left solid segments of pipe running through the middle of the valve.

Right-click the line, click **Properties**, and change the line to a construction line.

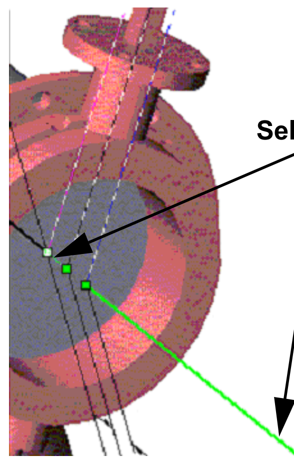
Delete the other 1.1875 inch line.

The next time that you exit the route you won't get rebuild errors.



13 Add geometric relations.

When dropping a piping part from the Feature Palette, normally you'd specify the part rotation angle. But, if there are two centerlines attached to the connection point in the sketch, placing the valve may not produce the results that you expect. With two centerlines, the valve may snap to the wrong one.



Having deleted the second short (1.1875 inch) line segment makes the sketch under defined.

Create a collinear relationship between the newest centerline and the line sticking out to the right.

14 Add another butterfly valve.

Drag and drop the next butterfly valve flange onto the proper point in the route. Select the 6-inch configuration and click **OK**.

The butterfly valve snaps to a proper rotation as well. Change the angle of rotation to 45 degrees.

15 Exit the sketch.

You may get rebuild errors. Exit the 3D sketch to resolve the errors.

16 Save and rebuild.

Save the assembly and rebuild if you are prompted to do so.

Adding a Victaulic (or flex) Coupling

Adding a Victaulic Coupling is very easy. All you need is a split point in your 3Dsketch to drop your coupling on. Once the coupling is inserted, the pipes that were made by the split in the line trim to the proper length.

1 Select route.
Right-click on the route and click **Edit Route**.

2 Split the route and add dimensions.

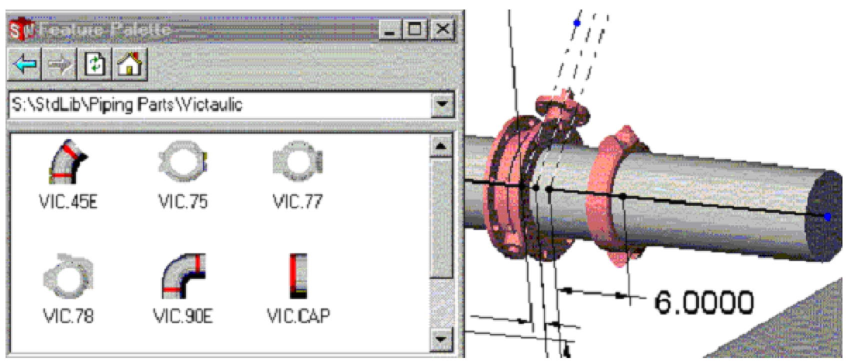
Click Split Curve and dimension this sketch segment to be 6-inches from the connect point of the last flange.

3 Add Victaulic coupling.
Open the folder in the Feature Palette named `Victaulic`.

Drag and drop a `VIC.75` onto the split point. You should see it snap into place.

If rotation of the coupling was critical, a centerline could have been placed in the sketch like it was for a flange).

4 Configure coupling.
Select the proper configuration size, and click **OK**.



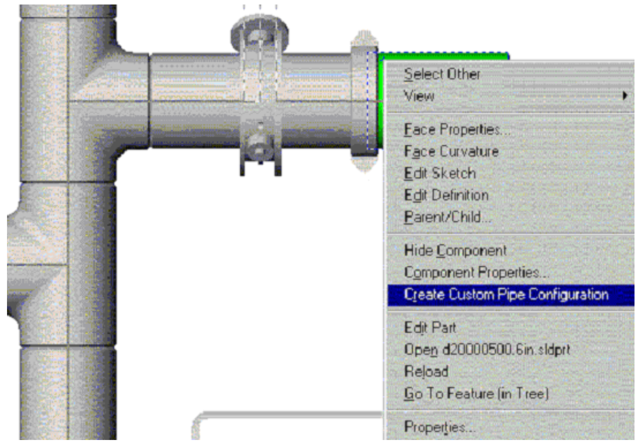
5 Exit 3D sketch.
Exit the sketch.

6 Look at gap in pipes.
Go to the Front View Orientation, change to **Hidden Line Gray** and zoom in close to see that the pipes have both been cut back to proper length so that the coupling may flex.

Groove Cut Pipe Ends

The Victaulic coupling needs grooves cut in the pipe ends.

1 Select pipe.
Right-click on one of the pipes that was cut back because we added the Victaulic coupling.



2 Edit part.
Right-click on the same pipe and click **Edit Part**. Expand the part in the FeatureManager design tree and observe its history.

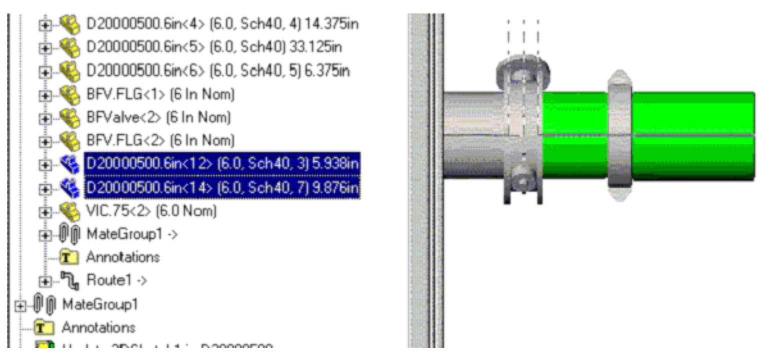
3 Suppress the chamfer.
Suppress the **Weld Chamfer** feature on the end of the pipe that has the Victaulic coupling on it.

4 Unsuppress the grove.
Unsuppress (resolve) the Victaulic Groove feature on that same end.

5 View the groove.
If you zoom in close, you will be able to see the groove cut in the pipe.

6 Change the other pipe.
Follow the same steps to change the end condition of the pipe on the other side of the Victaulic coupling.

7 Configurations.
Take note of the configuration number of both these pipes. Open up the pipe part (`D20000500 6in`) and see what exactly just happened.



8 View the FeatureManager design tree.
Click through all the configurations of the pipe to see that, except for the two we just changed, all the pipes have the same end conditions with different lengths.

9 Close the pipe.
Close the pipe. There is no need to save it.

10 Save the assembly.

Adding a Concentric Reducer

Adding a concentric reducer from the Feature Palette is done in much the same manner as the adding the Victaulic coupling to the pipe route. Adding a concentric reducer is like adding any part to the pipe route. you must be in the 3D sketch.

1 Select route.
Right-click on the route feature of FeatureManager design tree and click **Edit Route**.

2 Split Curve.

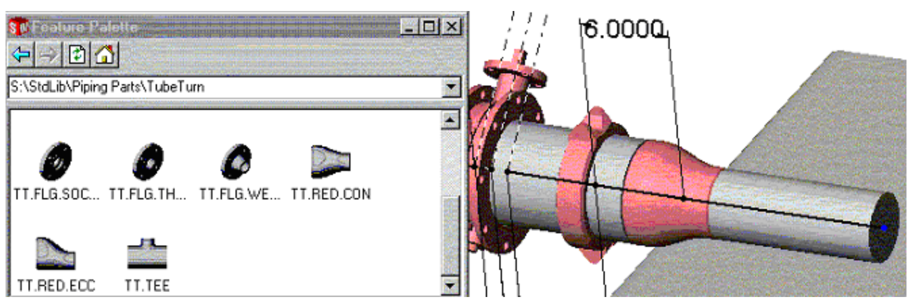
Add a **Split Curve** and dimension the sketch segment to be 6-inches from the hang point of the Victaulic coupling. If your line coming out of your coupling is not long enough, just drag its endpoint out a little.

3 Place the concentric reducer.
Open the `Tube Turn` folder in the Feature Palette.

Drag and drop the concentric reducer (`TT.RED.CON`) onto the split point.

You should see it snap into place.

4 Configure the concentric reducer.
Select the proper configuration size (`6 X 4 Concentric Sch. 40`) and click **OK**.

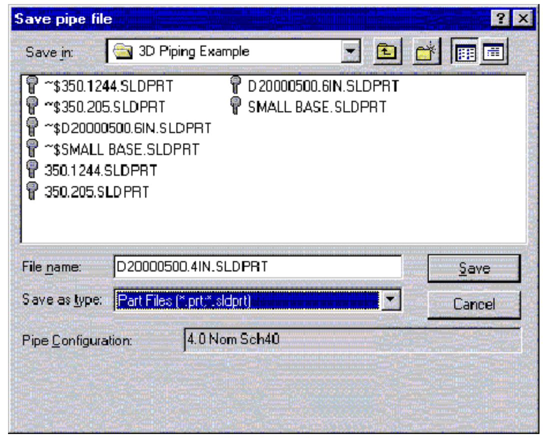


5 Change fitting orientation.
If the orientation isn't correct, right-click on the fitting and click **Change Fitting Alignment**. This flips the fitting to the opposite orientation and still keeps it in line with the pipe run. When it is in the orientation that you want, click the left mouse button to accept it.

Also, notice that the system is routing a new size of pipe out the other side of the reducer.

6 Exit the 3D sketch.

Because you've reduced the pipe you are routing down to a size different than that of the Default Route Properties setting, the system prompts you to name this new pipe part size.



Use the standard naming for pipe parts. Enter "`route assembly D# dot pipe size`" in the **File Name** field.

As you build your pipe Route, all 4-inch pipe will have the same part name, and the different lengths or configurations of that pipe size will show up out to the right of the pipe name. This is just as it did with 6-inch pipe.

Adding Other Mated Flanges to the Route Assembly

You can add other flanges at a different points in the route There is a new command that we will learn in the process of doing this. **Edit Route Assembly** allows you to add components to an existing route.

1 Zoom in on the suction flange.

Zoom in close on the suction flange of the pump on the right side of your frame.

2 Place the slip-on flange.

Drag and drop the slip-on flange (`TT.FLG.SLIP`) from the Feature Palette onto the edge of the pump flange.

3 Configure the flange.

Select the proper configuration size flange (`4.0 Nom 150#`) from the list, and click **OK**.

4 Add the flange to the current route.

By default, placing a piping part into the assembly begins a new piping route. However, at this time, we do not want a new piping route. Click **Cancel** in the **Fabricated Piping** dialog. Then, click **OK** in the next dialog box. This will add the new flange to the current pipe route.

5 View the flange mates.

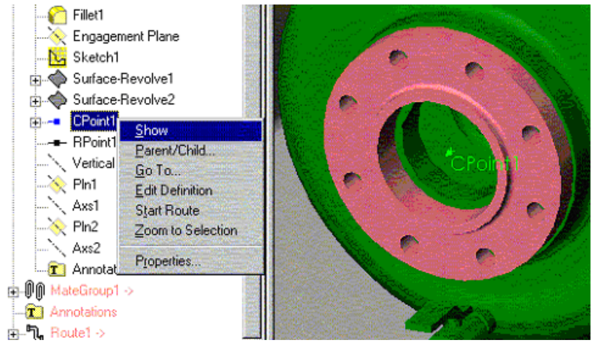
The flange is still under defined.

Expand `Mategroup1` of the route assembly. Notice the last two mates were created with the last drag and drop operation. So you still have one more mate to put on this flange to fully constrain it.

6 Display piping points.
Click **View, Piping Points**.

7 Display the connect point.
From the FeatureManager design tree, expand the flange you just placed.

Right-click the Cpoint, click **show**.



8 Edit the route.
The next thing we need to do is run pipe out from the CPoint (or connection point) of the flange we just inserted.

Right-click on the route and click **Edit Route**.

9 Add pipe.
right-click on **Cpoint** and click **Add to Route**.

This adds a short piece of pipe leading out of the flange in the proper direction. We will deal with the length of this pipe in the next lesson.

10 Hide connect point.
Right-click **Cpoint** (in the FeatureManager design tree) and click **Hide**.

11 Exit the sketch.
Exit the 3D sketch and save your work.

Do you see the last 4-inch piece of pipe in your Feature Manager design tree history? Remember it is the same part number (`D20000500.4IN`) as the other 4-inch piece of pipe you made earlier. But because of their different lengths, they become separate configurations within that same part file.

Lesson 3
Non-standard Elbows

Upon successful completion of this lesson, you will understand how to:

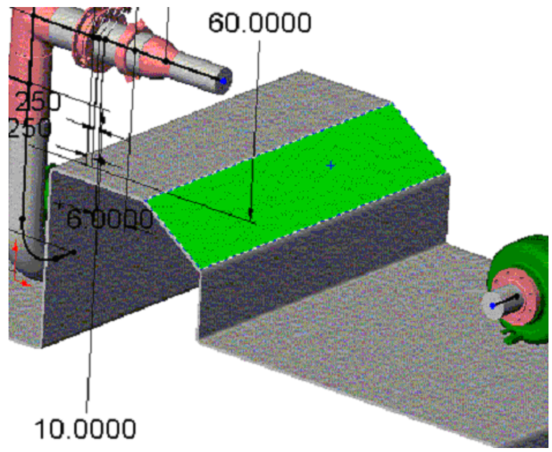
- Add a custom elbow (cut elbow other than the Default 90 degree that is most applicable to butt welded pipe cases.)
- Route pipe in the 3D sketcher in reference to (directionally) other surfaces of the assembly. Getting away from the standard (3 tab key) axis's that gets its orientation from the origin of the assembly.
- Change a Long radius elbow (the Default) to a Short radius.
- Add a standard 45 degree (long radius) elbow the route assembly.
- Add a reducing elbow.
- Add an eccentric reducer.
- Butt custom elbows together.

Introduction

In this lesson you will be adding to the assembly that was started in Lesson 2. The piping you route, many times needs to be plumbed at different angles or because of clearance problems, certain types of non-standard fittings need to be used. So, in this lesson you will learn some procedures that show you how to add non standard elbows.

Adding a Custom Elbow

The design intent of our pipe route is, defined by the 4-inch pipe that is connected to the 6 X 4 reducer. It is an angled pipe that runs parallel to the slanted pump mount face.

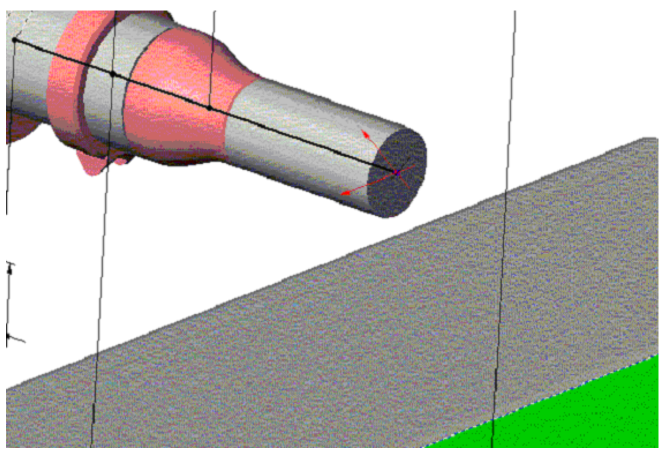


1 **Select the route.**
Right-click on the route and click **Edit Route**.

2 **Select face.**
Select the slanted face of the `Small Base`. See face selected in illustration.

3 **Sketch line.**
As you start sketching a line on end of the 4-inch pipe, look at the cursor feedback. See how each sketch plane has a direct orientation to the slanted plane?

Draw a line downward parallel to that new 3D sketch plane.

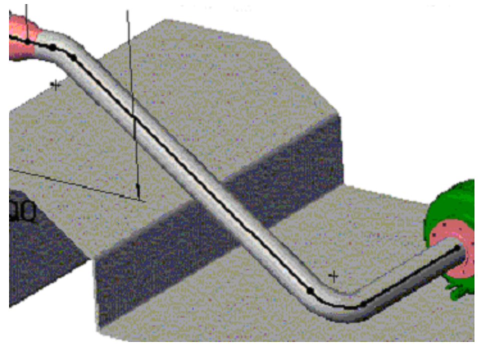


Remember to select the face you want to relate this new temporary sketch plane to before selecting **Line**.

4 **View pipe from a different angle.**
If you switch to the `Front` view orientation, you will see that the angled pipe we just made is not positioned such that it would intersect the other 4-inch pipe going into the pump suction inlet. So, in order to make this happen, we need to add some sketch relations to the ends of these two pipes so they intersect to form an automatic elbow.

5 Add geometric relations.
Add a **Merge Points** relationship between the two endpoints of the 4-inch pipes.

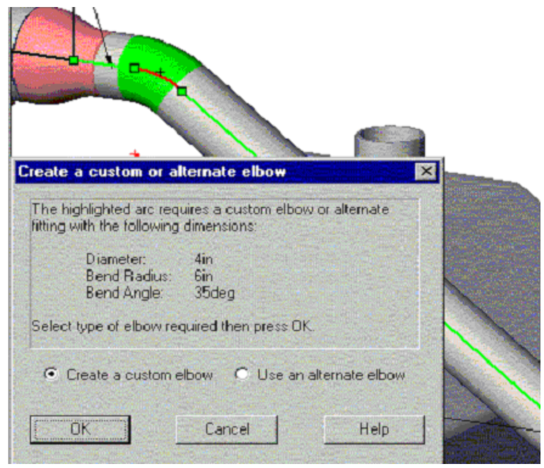
Adding the relations fully defines the sketch and forms a smooth 90 degree elbow at the intersection of the two pipes.



6 Exit the 3D sketch.

7 Create a custom elbow.
The **Create a custom or alternate elbow** dialog box appears.

Since the elbow is not a standard 90 or 45 degree ell, select **create a custom elbow** and click **OK**.

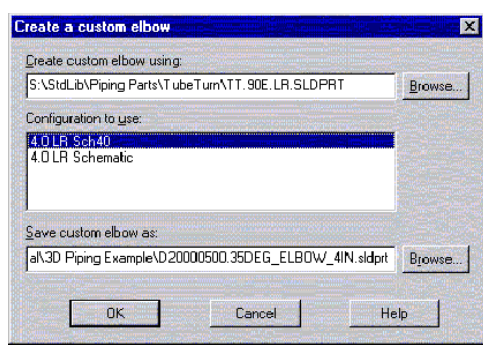


To create a custom fitting for a 35 degree cut elbow that is size is 4-inches, you must specify the following:

- Components used to create the customized part
- Configuration (schedule, size)
- Name of this customized part
- Folder to save the customized part to

Because at this point in the design we may not know what sub-assembly this elbow will be a part of, let’s just give it a name that describes what it does and we will rename it later. Make sure you are in the correct directory and enter `D20000500.35DEG_ELBOW_4IN` as the file name.

8 Accept configuration settings.
When all information is entered, click **OK**.



9 View the FeatureManager design tree.
Notice all the new parts models that just showed up in the Feature Manager design tree.

- 35 degree custom cut elbow
- 90 degree elbow

- Two straight pieces of 4-inch pipe

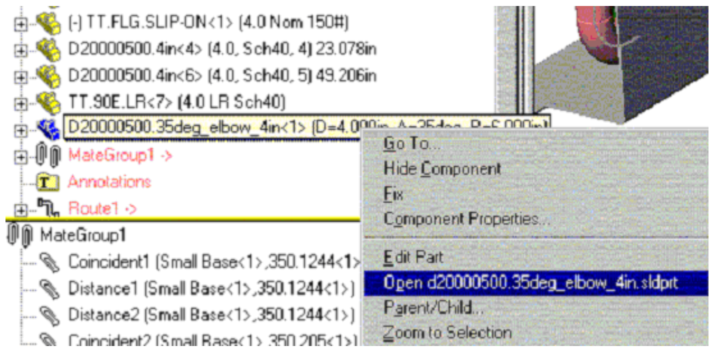
Looking at the Custom Elbow

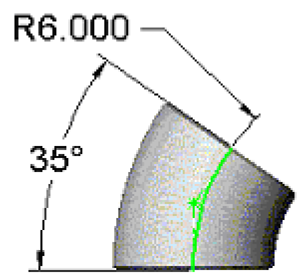
Let's look at what is unique about a custom elbow.

1 Open
Right-click on the `35 degree custom cut` elbow from FeatureManager design tree and click **Open**.

Note

Remember this is not a necessary process, we just want to see what happened when the elbow got created.





As you can see, when a custom elbow is created, different things happen than when a custom straight length of pipe is created in a pipe route. A custom pipe has one file with many configurations that show the different lengths of pipe. A custom elbow has one part file that represents the one angle cut configuration of the custom elbow. Every different angled cut elbow has its own separate part file.

2 Close the elbow file.

Breaking Up the Route

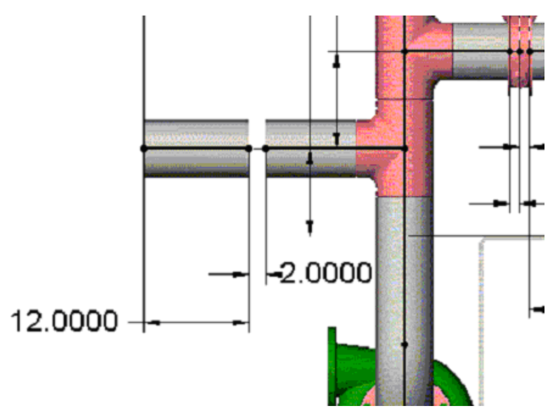
There may be times when you want to put a break in a pipe route. You may want to break the line so that the route is in more manageable sub-assemblies or to place flex coupling in the line.

1 Change view.
Change to the `Front` view.

2 Split the line.

Split Curve  the line in two places.

3 Change pipe properties.
Right-click the center section of the split, click **Properties**, change it to a

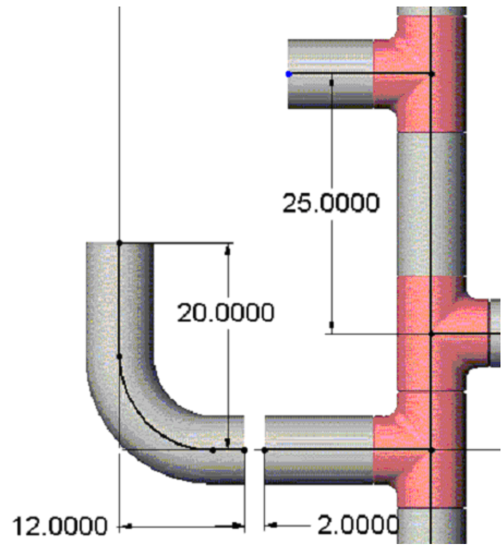


construction line. Add the 12.00 and the 2.00 dimensions as shown.

The pipe we just split will appear in the FeatureManager design tree as two pipes.

Changing an Elbow

Before we go on to the next new piping command, let's add a couple of parts to our route.



1 **Add a tee.**
Add a 6-inch tee from the Feature Palette.

2 **Add an elbow.**
Add a default long radius 90 degree 6-inch elbow into our piping route.

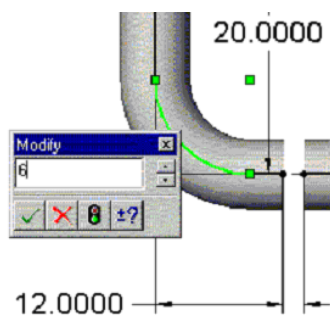
3 **Add dimensions.**
Dimension the sketch as shown in the illustration.

4 **Exit the sketch.**
Exit the 3D Sketch .

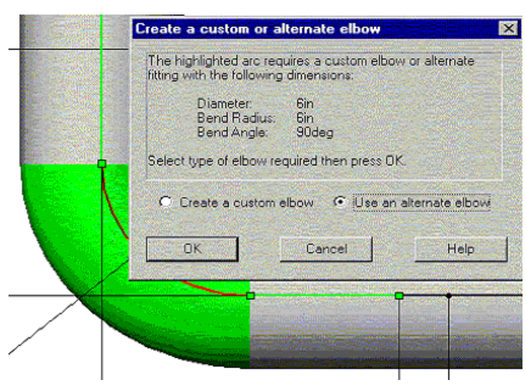
Take note of the new parts that have been added to the route assembly and the parts that changed.

5 **Edit the route.**
Right-click on the route and click **Edit Route**.

6 Change the dimensions.
Double click on the 9.00 inch dimension that represents the long radius elbow we just inserted and change the value from 9.00 inches to 6.00 inches (which is the centerline radius value for a short radius 6-inch standard elbow).



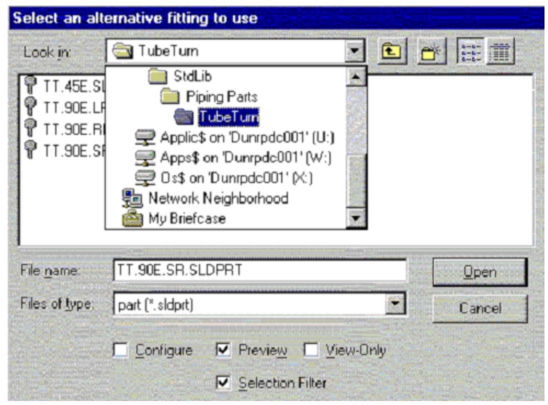
7 **Exit the 3D sketch.**
At this time the system prompts you (with the same message that it gave when you cut the Custom Elbow) that the radius in the sketch doesn't match the arc radius for the default size elbow in the **Route Properties** dialog (which was 9 inches). Since



we know we have a standard fitting (model) out on the system for a short radius elbow, click **Use an alternate elbow** bullet and click **OK**.

8 **Select elbow.**
When the browser window pops up, browse to the proper directory out on the network where the standard piping parts are located. Select the standard short radius elbow, `TT.90E.SR` and open that file.

9 **Configure elbow.**
Select the appropriate schedule of elbow, in this case 6.0 SR Sch40, from the configuration dialog box, and click **OK**.



If you look in the Feature Manager design tree, the short radius elbow will have replaced what used to be a long radius elbow in the Route.

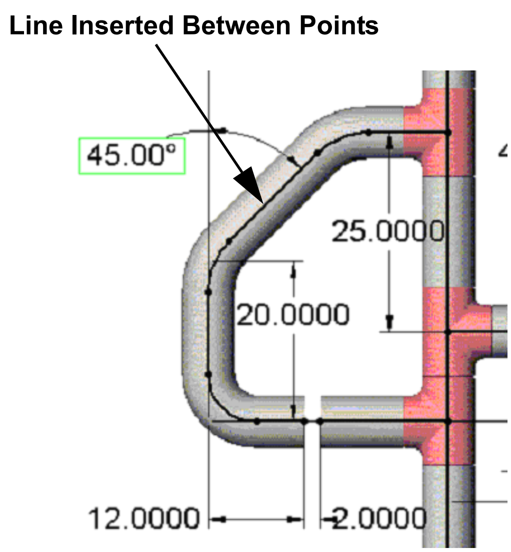
10 **Save your work.**

Adding a Standard 45 Degree Elbow

Adding a standard 45 degree elbow is much like adding a Custom cut elbow, with one exception. Because we have a standard 45 degree elbow in our system, we will put that into the Route instead of creating a Custom cut elbow part (cut at a certain degree, like we did in an earlier exercise).

1 **Select route.**
Right-click on the route and click **Edit Route**.

2 **Sketch line.**
In the section of the pipe we inserted the short radius elbow, use the sketch line command, and connect the open ended vertical pipe line with the open ended horizontal pipe line. After this is done, the arcs will automatically be created that connect the pipes together. Put a 45 degree dimension on that lines as shown in the illustration.

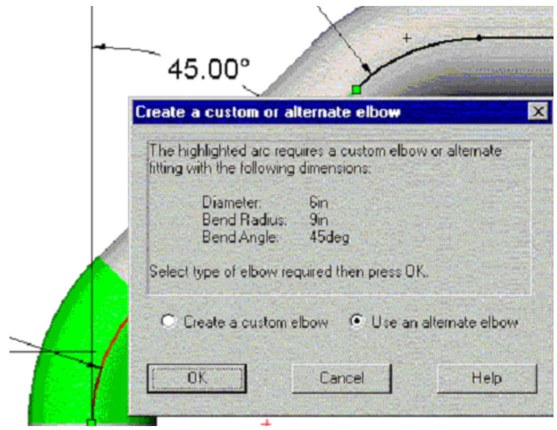


3 **Exit the sketch.**
As you exit the sketch, the system tries to build all the parts it can from

the sketch elements you have created. By default, it can create everything except a part for the two arcs that are in the 3D sketch.

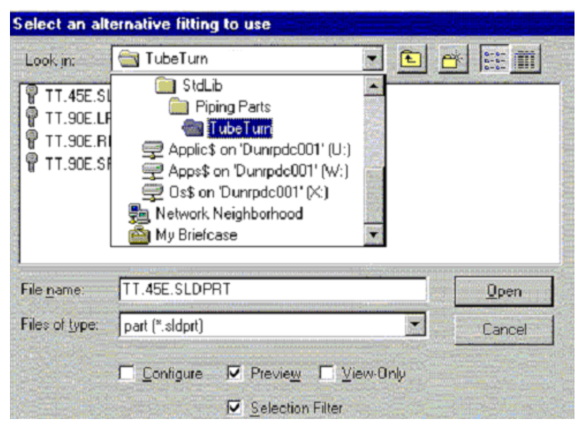
The system recognizes that the elbows that need to be created by these two arcs will not create a Default 90 degree long radius elbow.

4 Create alternate elbow.
Select **Use an alternate elbow** and click **OK**.



5 Select alternate.
When the **Select an alternative fitting to use** window pops up, browse to the proper directory where the standard piping parts are located.

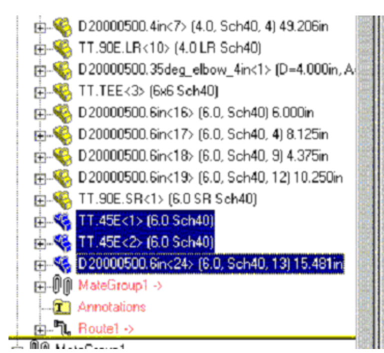
Open the standard 45 degree elbow, `TT.45E`.

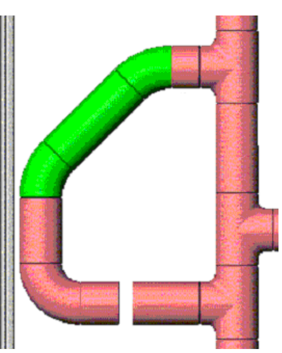


6 Configure alternate elbow.
Then select the appropriate schedule of elbow, in this case 6.0 Sch40, from the configuration dialog box, and click **OK**.

7 Configure the other 45 degree elbow.
Repeat this procedure for the other 45 degree elbow as well.

8 View FeatureManager design tree.
After Exiting the 3D Sketch, you can look in the FeatureManager design tree and find two instances of the `TT.45E` as well as another piece of straight pipe.





Adding a Reducing Elbow

Because a Reducing Elbow is not the Default Elbow in our Route Properties dialog, we will have to add it in much the same way as any fitting. Drag and drop it from the Feature Palette and establish its rotation by dimensioning that angle from a construction line in the 3D sketch.

In order for the elbow to fit atop the long vertical run of pipe (the one with the three tees in it), we will need to lengthen the pipe at the top.

1 **Select the route.**
Right-click on the route and click **Edit Route**.

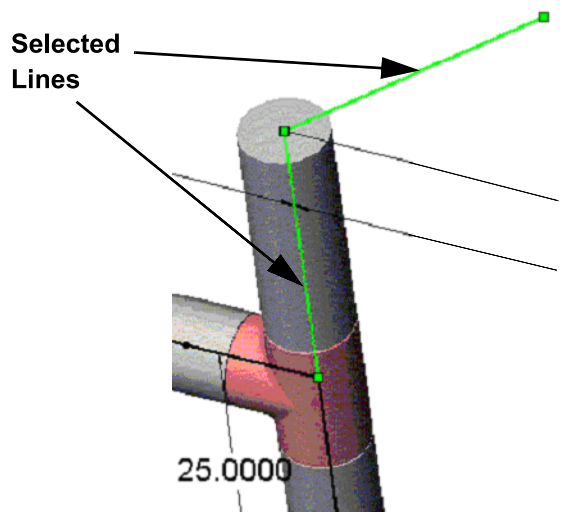
2 **Change dimensions.**
Change the 60.00 inch dimension from the top of the pump base to the top of the vertical pipe to be 70.00 inches. This will lengthen only the pipe at the top, because we have define all the rest of the3D sketch (and thereby the pipe parts as we have went along).

3 **Create elbow.**
Click **Line** and draw a line some distance (don't worry about how long) out from the point at the top of the vertical pipe you just lengthened. This puts a default (long radius 90) elbow at the intersection of these two lines.

4 **Undo.**
Remove the elbow, but keep the sketch line.

Click the **Undo** tool to remove the elbow, but leave the sketch line.

To drop an elbow at the intersection of these two lines (the one we just drew and the vertical one), make them perpendicular to each other.



5 **Add elbow.**
From the Feature Palette, open up the `Tube Turn` folder and locate the reducing elbow (`TT.90E.RED`) and drag but don't drop it onto the intersection point that we established earlier.

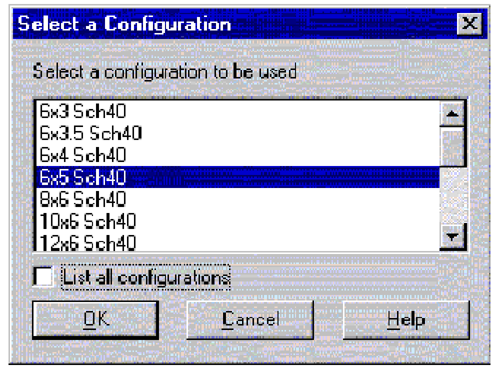
6 **Verify the relationship of the elbow to the route.**
Drag the line around (not by its endpoint, that will only change its length) to see that it is perpendicular to the vertical line but the rotation has not been established yet.

7 **Specify orientation.**
While holding onto the elbow with one hand, reach over with your

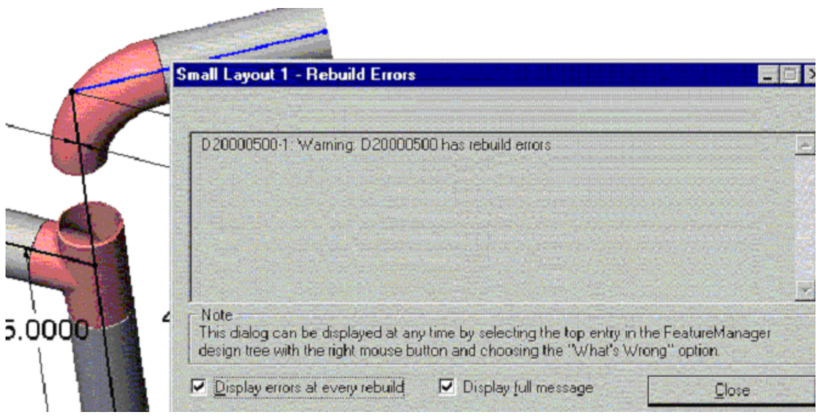
other hand and press the **Tab** key a couple of times. Still, don't let go of the elbow. Do you see how it toggles through the orientation possibilities for the elbow?

8 Place elbow at wrong orientation.
Toggle to the wrong orientation and let go of the elbow.

9 Configure elbow.
In the next dialog box, select the `6x5 Sch40` configuration of the elbow and then click **OK**.

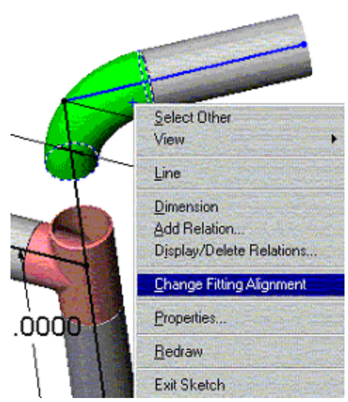


A dialog box appears and you see that because of the orientation of the reducing elbow the system was unable to generate pipe between the elbow and 6-inch tee.



In the earlier step we could have used the **Tab** key to orient the elbow correctly in the first place. If the elbow was oriented, we could skip some steps. Click **Close**.

10 Change elbow alignment.
Right-click on the reducing elbow and click **Change Fitting Alignment**.



Take note of the change to the cursor, and what each mouse button does.

- Right mouse button toggles through possible orientations of the fitting.
- Left mouse button confirms the orientation.

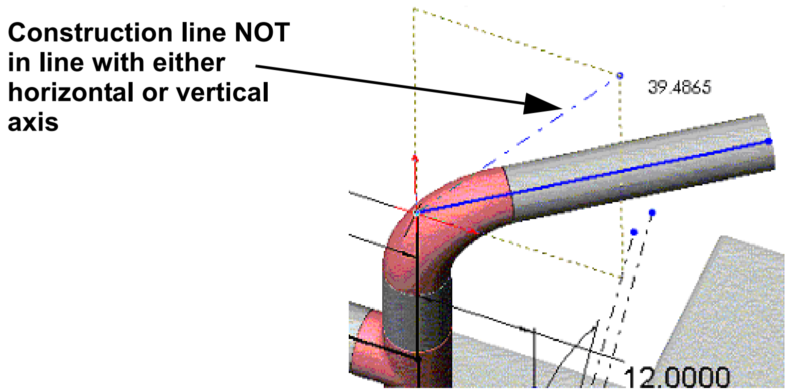
11 Orient the elbow.
Click the right mouse until you see the elbow turned the correct way and then click the left mouse to confirm your changes.

Like the rotation of the bolt holes in a flange, the rotation of the reducing elbow follows the horizontal line (blue, under defined). Drag the line around to see what happens.

12 Add geometric relations.

As with the rest of the pipe route, the elbow needs to be oriented into an exact position. To do this, we will establish a construction line from the apex of the elbow that is coplanar with the rotation line of the elbow.

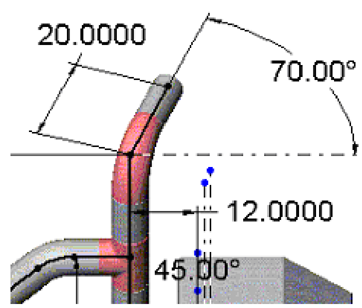
Draw a construction line from the apex of the reducing elbow at some skewed angle. The angle is not critical at this time.



Add a parallel relationship between the construction line horizontal of pipe just below the reducing elbow. This gives us a piping element within the assembly to dimension the rotation of the elbow from.

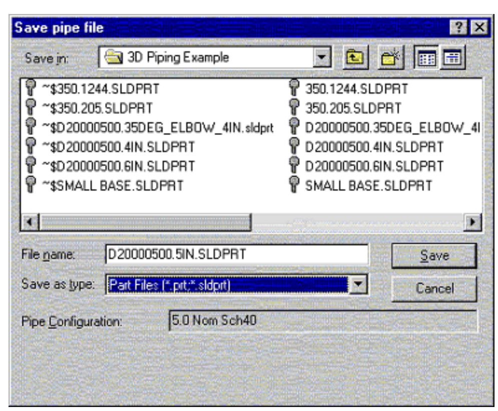
13 Specify an angle.

put a 70 degree angled dimension between the construction line and the (rotation) line pipe. Also, specify a 20 inch dimension for the new pipe.



14 Name the elbow.

Because you are adding a 5-inch pipe to the route (coming out of the reducing elbow) for the first time, the system prompts you to save this pipe and give it a name. Don't use the generic file name. Use the naming convention that we have used throughout this document, which is your `"route assembly`



D#.pipe size".

The history of the route should have the reducing elbow and an 11.00 inch piece of 5-inch pipe in it.

15 Save your work.

Adding an Eccentric Reducer

Eccentric reducers are different than concentric reducers. Placing eccentric reducers is more like placing reducing elbows than concentric reducers. In particular, it is very important to establish the correct orientation for eccentric reducers.

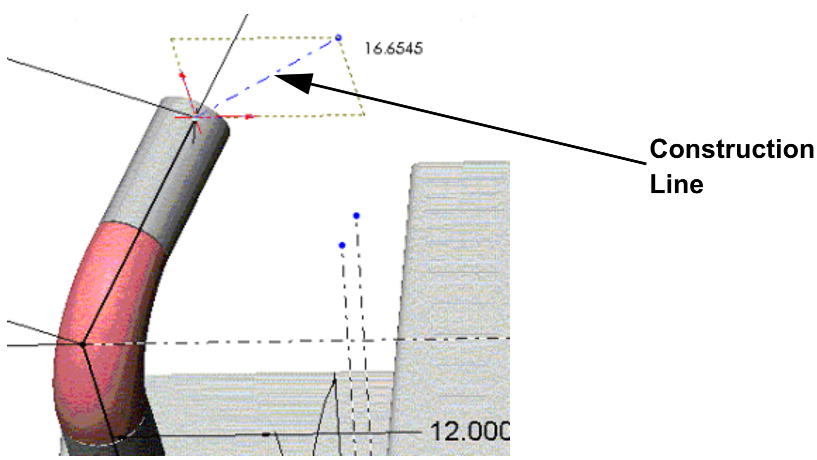
The following are the general steps for placing eccentric reducers:

- Insert a construction line for the vertical axis of the reducer.
- Drag and drop the eccentric reducer from the Feature Palette.
- Insert a second construction line which starts at the same point as the first construction line.
- Establish a sketch relationship between the second construction line and another entity in the assembly that can be used to constrain the reducer.
- Add dimensions.

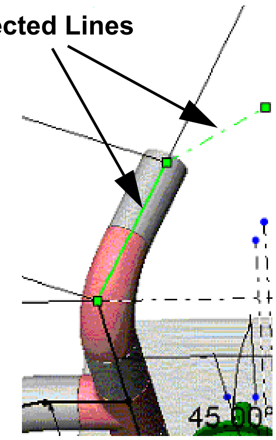
To add an eccentric reducer:

1 Edit roue.
Right-click on the route assembly and click **Edit Route**.

2 Create construction line.
Sketch a construction line from end of the 5-inch pipe, at some skewed angle. The angle is not critical at this time, we just don't want any geometric constraints of horizontal or vertical with the (red) sketch axis as make this line.



3 Add geometric relations.
In order to drop an eccentric reducer that will snap to this construction line for rotation, we have to make it perpendicular to the line it was started from.



Add the perpendicular relationship now.

4 Place eccentric elbow.
From the Feature Palette, open up the **Tube Turn** folder and locate the eccentric elbow (`TT.RED.ECC`) and drag and drop it onto the endpoint of the construction line (where it intersects with the pipe).

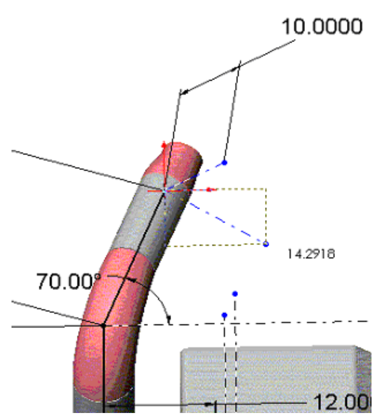
5 Select elbow configuration.
Select the `5x4 Eccentric Sch40` configuration of the reducer from the dialog box, and click **OK**.

The vertical axis of the eccentric reducer part snapped to the construction line to establish its rotation.

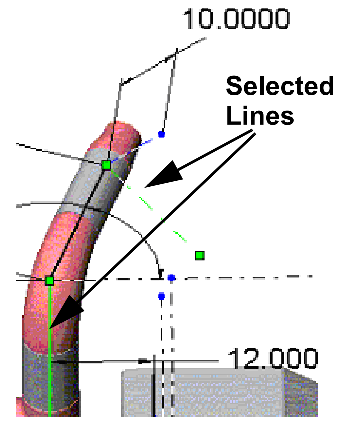
If you drag the construction line (not by its endpoint) around, it changes the rotation orientation of the reducer. To establish an exact angle of rotation we will have to create another construction line in the same plane as the first construction line.

6 Add dimensions.
Add a dimension of 10 inches to the first construction line.

7 Add construction line.
Draw a construction line from the same point where we started the first (10.00 inch) construction line (the point where the reducer dropped onto the 5 inch pipe) at some skewed angle. The angle is not critical at this time, we just don't want any geometric constraints of horizontal or vertical with the (red) sketch axis as we make this line.

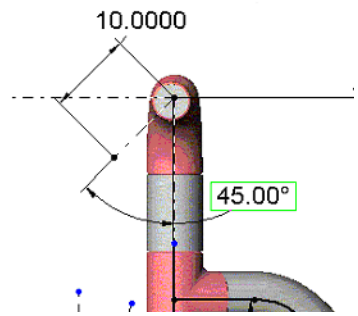


8 Add geometric relations.
Add a parallel relationship between the second construction line and the vertical sketch line of the reducing elbow. This gives us a piping element to dimension the rotation of the elbow from.



9 Add an angle.
Add a 45 degree dimension between the two construction lines.

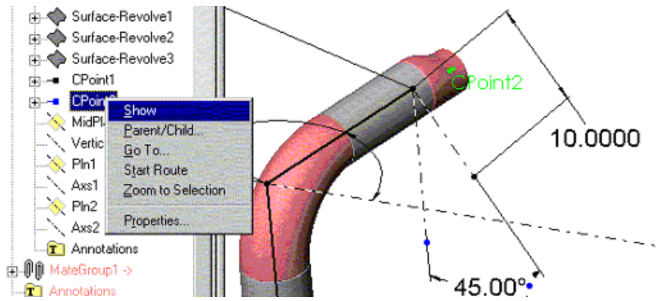
You should be able to see how the reducer is rotated.



10 Add pipe.
Run a 4-inch pipe from the **CPoint** on the open end of the reducer. It is done much the same way in which we added pipe to the route from the 4-inch slip-on flange in an earlier lesson.

11 Show piping points.
Click **View**, **Piping Points**.

12 Show connect point.
From the FeatureManager design tree, expand the reducer (`TT.RED.ECC`) that you just added. Right-click the appropriate **Cpoint**, click **Show**.



13 Add to route.
Right-click the connection point and click **Add to Route**.

A short piece of 4-inch pipe is added, leading out of the reducer, in the proper direction. The length of this pipe is not important at the moment.

Why didn't we have to name this 4-inch pipe we just added to the Route? Do you see the last 4-inch piece of pipe in your

FeatureManager design tree history? It is the same part number (`D20000500.4IN`) as the other 4-inch pipe. But, because of their different lengths, they become separate configurations within the same part file.

14 Save.

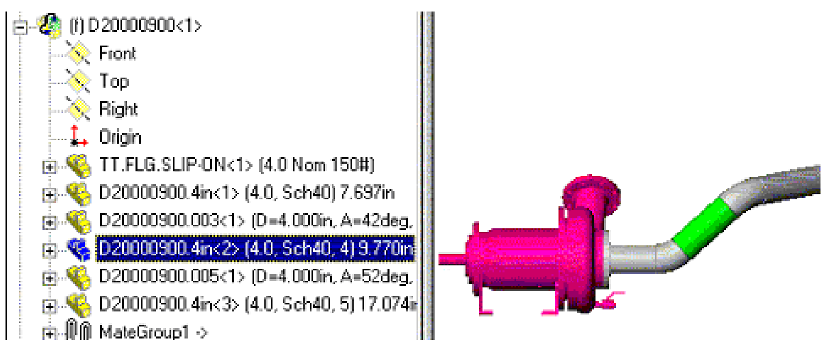
Save the work you've done and close the assembly.

Butting Elbows Together

Earlier we butted two standard tees together, thereby removing the straight pipe between them. But, the procedure for butting elbows together is slightly different than the procedure for butting tees. The difference is that, elbows are defined by their sketch radius fillets. These sketch radius fillets require a tangent straight line section to be in the 3D sketch (at both ends of the fillet) in order for them to resolve. So, when making an elbow to elbow connection, you must leave a small length of straight pipe between the elbows. If you make the straight line a construction line, then the small straight length of pipe won't show up in the FeatureManager design tree, but will still be adequate to butt the elbows.

1 Open the assembly.
Open the assembly file named, `Butting Cut Elbows`, found in `C:\MyDocuments\Sldworks_Local\3D Piping - Butting Cut Elbows`.

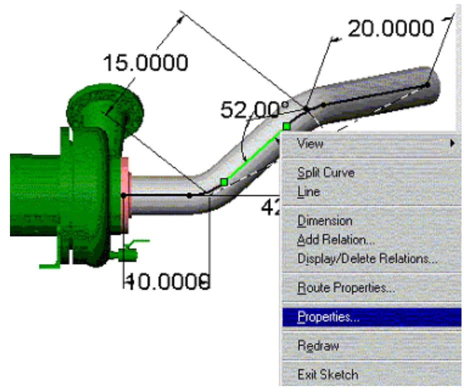
2 Select the pipe section.
Select the straight section of pipe between the two cut elbows. It is 9.77 inches long. This is the section of pipe we need to remove most of in order to establish an elbow to elbow connection.



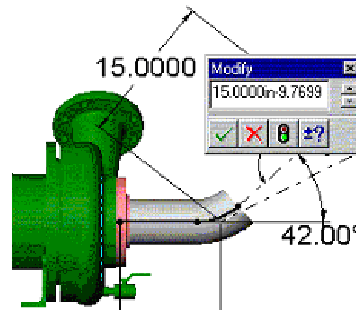
3 Edit route.
Right-click on the route and click **Edit Route**.

4 Change a line to construction.
Right-click on the sketch line between the two radii (that represent the elbow parts) and click **Properties**.

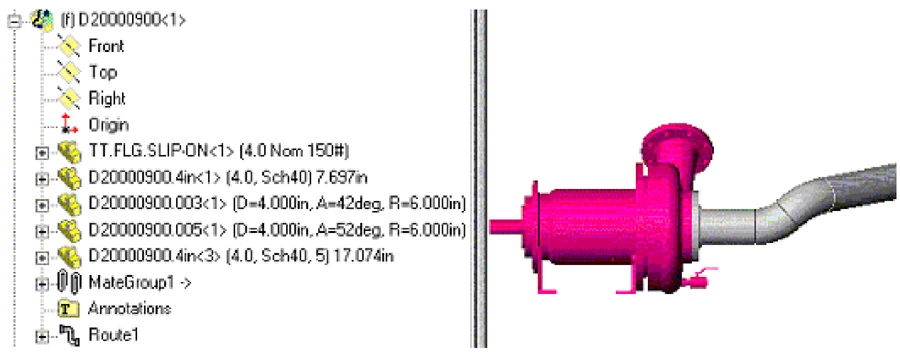
In the PropertyManager, select **For Construction**.



5 Remove section between fillets.
Double-click the `15.00` inch dimension that represents the distance between the virtual sharps of the fillets. Change the dimension to `.0001`. Click **OK**.



6 Exit the 3D sketch.
This procedure has removed the 9.77 inch pipe from the Feature Manager design tree and butted the two elbows together.



Lesson 4
Non-Standard Pipe Connections

Upon successful completion of this lesson, you will understand:

- Pipe to pipe penetration (or pipe saddling, chemical injection points, drain points, and so forth.)
- Stub-in or offset pipe intersections (example: Headers)
- Adding mounting brackets to the pipe route.
- Forming Sub-assemblies.
- Adding another route assembly (using `START.PART`)

Introduction

In this lesson you will be adding to the assembly that was started in Lesson 2. There are times that standard fittings, such as tees, are not preferred. When these instances occur, there is a need to saddle and weld standard pipe (couplings of other fittings are also used) into runs of pipe.

Pipe to Pipe Penetration

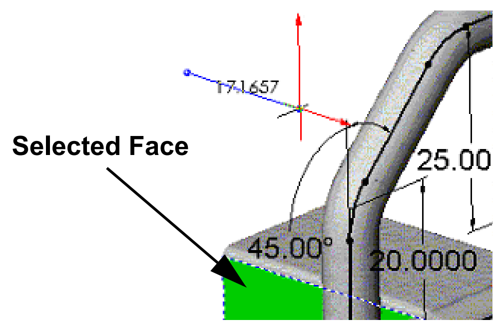
Pipe to pipe penetrations are also known as pipe saddles. There are two things that should be kept in mind while performing the saddling of one pipe into another. Both sections of pipe must have size assigned to them, and always stay aware of the environment you are in during each step of this operation.

1 Open the assembly.
Open the assembly file `Small Layout 1`, found in `C:\My Documents\Sldworks_Local\3D Piping Example`.

2 Edit Route.
Right-click on the route feature, to **Edit Route**.

We will add our pipe saddle to the pipe that is between the two standard 45 degree elbows. We will point the saddled pipe out the back side of the 45 degree pipe.

3 Add line.
Construct this line in a plane that is parallel with the large vertical face of the pump mount.



Select the large vertical face of the pump mount

Click **Line** and sketch a line in the X direction (watch the cursor feedback) in the general area of where you want it to be, but not touching the line (at this time) it will intersect with.

4 Add geometric relations.
give a coincident relationship to the endpoint on the line that you just made and the pipe line at the 45 degree angle.

The line could be at an angle to the pipe centerline if desired. This can be useful for mounting sensors and so forth.

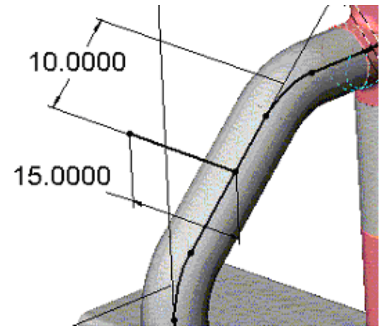
Notice the differences between this operation and simply adding a standard tee to a straight run of pipe. There is no split point at the intersection of these lines, that would create three sections of pipe, like is done when adding a tee. The straight run of pipe must be a single segment as is the line saddled into the straight run.

Note

You cannot make a pipe to pipe penetration at the intersection of more than two elements.

5 Define the sketch.

Fully define this sketch line (which has been the practice throughout this exercise) with the proper dimensions.



Before the **Penetrate** can be done, both pipes have to have size. So when the system saddles them together, it will know what size to make the cut in one pipe and the size of the saddle in the other.

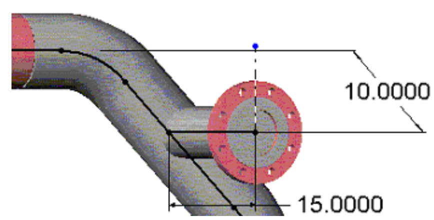
6 Add flange.

From the Feature Palette, drag and drop a 4-inch, Schedule 40 socket weld flange onto the open end of the line we have just defined.

An errors dialog box appears. This indicates that there is no split point or penetration yet. Close this dialog box. We will resolve this problem later.

7 Define flange rotation.

Let's give the flange an exact rotation by putting a vertical construction line at the end of the line where the flange is located.



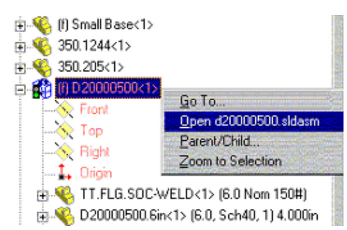
The construction line has to be perpendicular to the line representing the pipe.

This will two-hole the flange at 12 o'clock, because the vertical axis of the flange is located between two of the bolt holes.

8 Exit the 3D sketch.

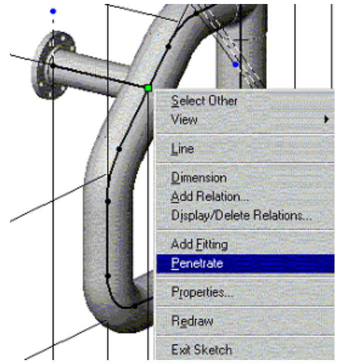
9 Select and edit route assembly.

Right-click on the route assembly in the Feature Manager design tree and open it in its own window.



10 Edit route.

Right-click on the route and click **Edit Route**.



11 Orient graphics display.
Orient the graphics so that the point at which the two pipes intersect can be seen.

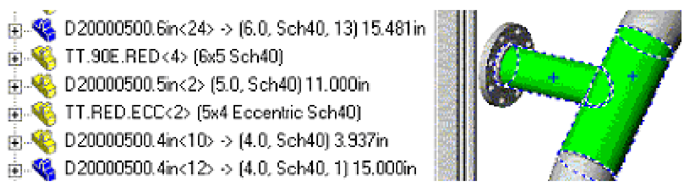
12 Pipe to pipe penetration.
Right-click on that point and click **Penetrate**.

This creates a pipe saddle.

13 Exit the sketch.
The cuts are created as in-context features of the sub-assembly. Notice an **Update Sketch** feature at the bottom of Feature Manager design tree for each pipe that was cut.

The system created a custom configuration of each pipe in its respective pipe file. Those pipe configurations have the cut features unsuppressed to cut each pipe appropriately.

14 View FeatureManager design tree.
Locate the two pipes in FeatureManager design tree and take note of their configuration numbers. The 6-inch pipe is `configuration 13`, and the 4-inch pipe is `configuration 1` in its part file (your configuration numbers may differ).



Open the 6-inch pipe file. Switch to the configuration with the hole in it that was made by the **Penetrate** command. This is called a custom pipe configuration. Take note of the in-context **Cut Extrude** feature that is unsuppressed in this configuration. If you switch to any other configuration of the pipe, the **Cut Extrude** feature is suppressed.

15 Close the pipe part.

Note

Straight pipe Custom Configurations — the custom pipe configuration resides in the same part file as all the other same size, pipe configurations.

Elbow Custom Configurations — the custom elbow configuration is created and resides in a separate part file unto itself.

16 Close assembly.
Close the route assembly, rebuild, and save all your work.

Creating an Offset Pipe Penetration (or Stub-in)

There are times when designing pipe systems that the need will arise for a pipe to pipe penetration that is slightly different than the one discussed in the last lesson. The intersecting pipe doesn't line up with the centerline of the straight pipe run — as was the case in the last lesson.

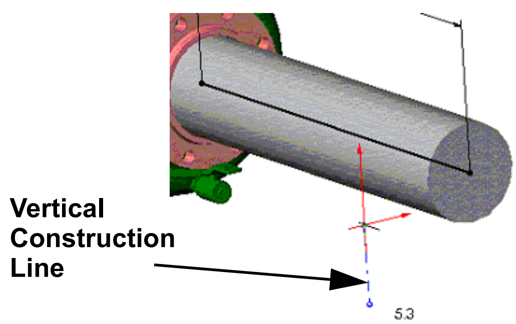
With this type of pipe to pipe construction, the "Pipe Penetration" command from the last lesson will not work. We Will have to create a "custom pipe configuration" ourselves and then edit the part, in the context of the assembly to fashion the cuts.

1 Open assembly.
Open the assembly file `OFFSET STUB IN PIPE`, found in `C:\My Documents\Sldworks_Local\ 3D Piping - Offset Pipe Penetration`.

2 Edit route.
Right-click on the route and click **Edit Route**.

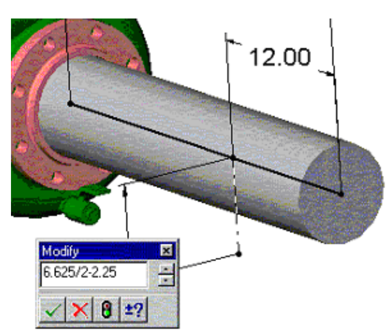
The design intent for this header is that the 4-inch pipe stub-in also acts as a drain for the 6-inch header. In order to do this, we will add construction geometry to help us properly locate the stub-in. So, the lower outside dimension of the 4-inch pipe will be flush with the lower outside dimension of the 6-inch header.

3 Add construction line.
Sketch a vertical (watch the cursor feedback in reference to the red axis) construction line out in space.



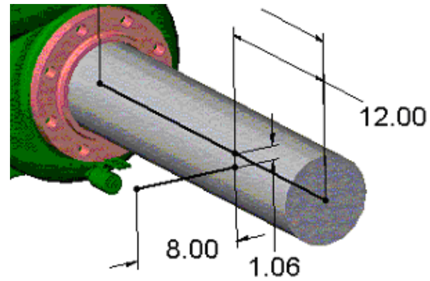
4 Add geometric relations
Define a coincident relationship to an endpoint on the construction line and the sketch line that represents the 6-inch pipe (we will call it the header from now on).

5 Add dimensions.
Add the dimensions shown in the illustration that define the length and location of the construction line. Note that the values in the equation represent the following:



- 6.625/2 — Half the 6-inch pipe outside dimension.
- 2.25 — Half the 4-inch pipe outside dimension.

6 Add pipe.
From the open endpoint of the construction line, sketch a horizontal line that is 8-inches long, this will become the stub-in 4 in pipe.



7 Add START.PART to route.
The line does not have route properties associated with it because it is separated from the routed pipe sketch entities by the construction line. To establish route properties for the line, a connection point must be inserted on the end of the line segment.

Drag and drop a 4-inch fitting on the endpoint of the line, and then hide the flange or drag and drop a 4-inch `START.PART` on the endpoint of the line, and then hide it.

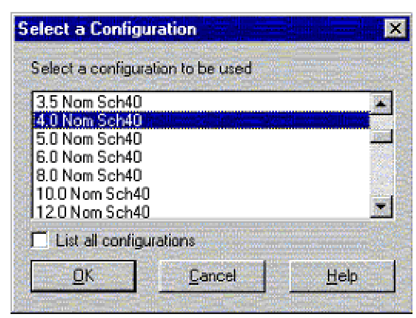
The `START.PART` is a piping part model that is shaped like a disk. It has a connection point and route point feature. If `START.PART` is dropped onto an endpoint of a 3D sketch line in a pipe route it can give that line route properties and pipe size.

The single part file has a configuration for every pipe and tubing size available. It also has a `Vertical` axis feature that gives you the ability to rotate it to exact angles.

`START.PART` was created to start a pipe route without having to drag a flange off the Feature Palette as the initial part. When `START.PART` is used it should be subsequently hidden in the route assembly. Think of `START.PART` as construction geometry.

Open the Feature Palette to the `C:\StdLib\Piping Parts` folder. Drag and drop the `START.PART` onto the open endpoint of the sketch line (it should snap into place).

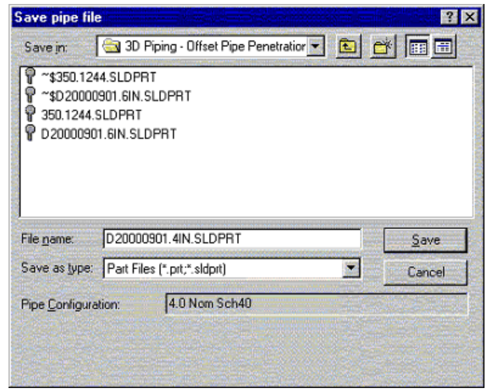
8 Define pipe size.
In the **Select a Configuration** window, click `4.0 Nom Sch40` and click **OK**.



9 Resolve errors.
Rebuild errors may appear. If so, these indicate that there is no split point or penetration where the two pipes intersect. Close this dialog, we will deal with the intersection of the pipes later.

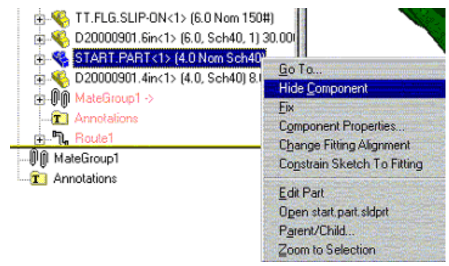
Also, don't be concerned if you see shadow of `START.PART` in the 4-inch pipe. We will hide it once we exit the 3D sketch.

10 Exit the 3D Sketch.



Each unique pipe within the route assembly needs to be saved in its own file. The pipe that you just created is a different pipe size than any other in the route assembly. So, you must name this new pipe part size. Remember to use the standard naming for pipe parts. Type in your “`route assembly D# dot pipe size`” in the **File name** field and click **Save.**

11 Hide START.PART graphics.



Right-click on the `START.PART` in the Feature Manager design tree and click **Hide Component**.

Remember, this only removes it graphically. If you suppressed it instead, then the 4-inch pipe would lose its size reference.

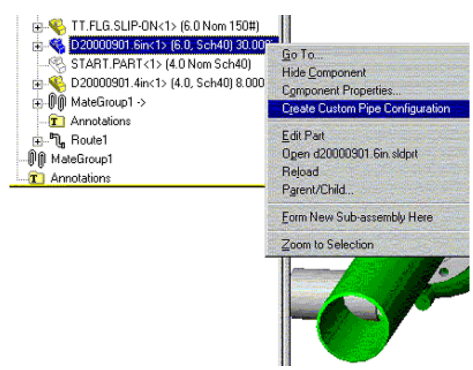
12 View assembly.

Look at the route assembly from different angles to see that 4-inch pipe is the correct position.

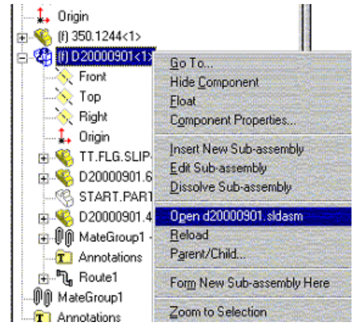
The 4-inch pipe needs to be saddle cut around the 6-inch pipe. The 6-inch pipe needs a hole cut into it to the inside diameter of the 4-inch pipe.

Another command we will use in this process, is **Create Custom Pipe Configuration**. This command, does by hand, what basically happened automatically with the **Pipe Penetration** command. A configuration is created in the pipe part file, being used in the route assembly, that not only cuts the pipe to a specific length, but also creates (and unsuppresses) the cut features unique to that particular configuration.

13 Create custom pipe configuration.



While editing the `OFFSET_STUB-IN PIPE` assembly (which in this exercise represents the upper level assembly), right-click `D20000901.6IN` (6-inch pipe) and click **Create Custom Pipe Configuration**.

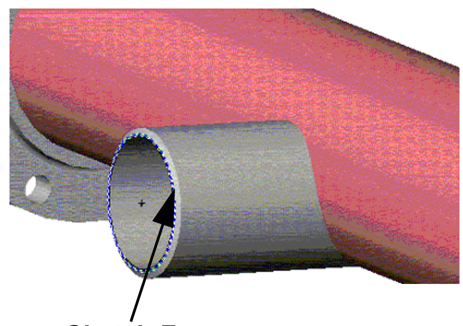


14 Open part.
Right-click `D20000901` (the route assembly) and open it in its own window.

15 Edit part.
Converting edges from the other pipes creates an in-context relationship between the two. Right-click on the larger pipe and click **Edit Part**.

Click **OK** to the next message.

16 Zoom in on model.
Zoom in close and position the assembly so you can start the next sketch on the small face on the outside end of the 4-inch pipe.



17 Select sketch face.
Select face to sketch upon.

Click .

18 Convert entity edges.
Select the inside diameter edge of the 4-inch pipe click  to convert the edges to a sketch.

19 Cut extrude.

Click **Cut Extrude** and cut the 6-inch pipe with a **Blind** end type to a depth of 8.00 inches.

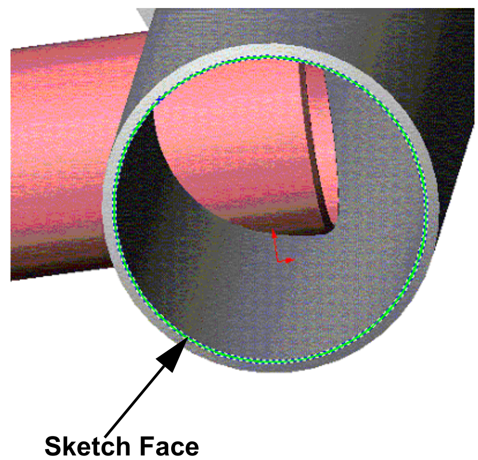
20 Edit part.
Right-click in graphic area and click **Edit Assembly: OFFSET STUB-IN PIPE**.

21 Create custom pipe configuration.
Right-click on the 4-inch pipe and click **Create Custom Pipe Configuration**. Right-click on the 4-inch pipe, click **Edit Part** and click **OK.**

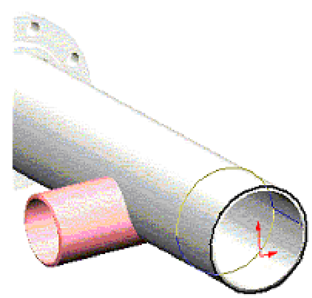
22 Zoom in on model.
Zoom in close and position the assembly so you can start the next sketch on the small face at the end of the 4-inch pipe.

23 Select sketch face.
Select face to start sketch and click .

24 Convert entity edges.
Select the outside diameter edge of the 6-inch pipe, and click to convert the edge to a sketch.



25 Cut extrude.
Now that we have a fully defined sketch, let's make our cut. Select **Cut Extrude** and cut the 4-inch pipe with a **Through All** end type in the proper direction.



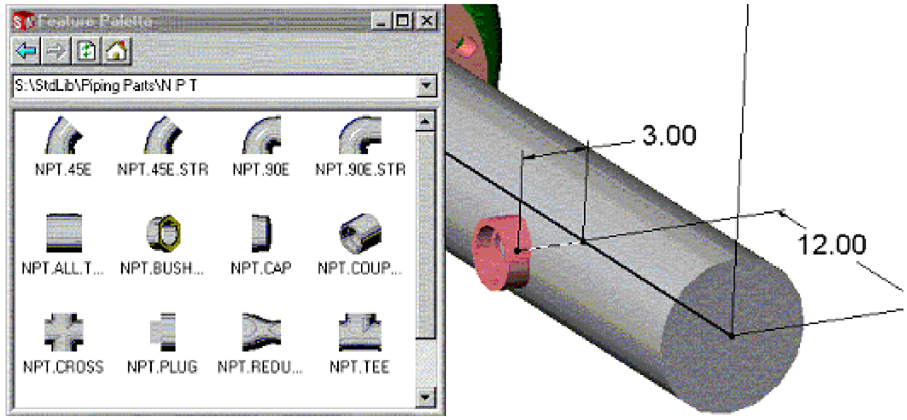
26 Save.

Save your work and close this assembly.

Adding a Coupling

Adding a coupling (2-inch in this case or other fittings) to a piping route is done in much the same manner as adding the stub-in that we just completed.

Remember that any time the situation varies from two pipes intersecting at a single point you can not use the **Pipe Penetration** command. An example of how a coupling might be dropped in the 3D sketch is shown in the illustration.



The following is the basic procedure for adding a coupling:

- Add construction geometry in the 3D sketch to position the coupling. Do not split the straight run of pipe but make one end of the construction geometry concentric with the straight run.
- Drag and drop the coupling's route point from the Feature Palette onto the endpoint of the sketch geometry.
- Exit the 3D sketch and **Create a Custom Configuration** of the pipe.
- Edit the pipe and coupling to make the appropriate cuts.

Adding a Mounting Bracket

Once all the piping has been routed in an assembly, holding the pipes in place becomes the next problem. Because the mounting brackets are part of the piping sub-assemblies themselves, there is a generic piping mount in the library palette of parts. This part can be added (drag and drop) to the 3D sketch just like any other fitting.

1 **Open assembly.**
Open the assembly file `Small Layout 1`, found in `C:\My Documents\Sldworks_Local\3D Piping Example`.

2 **Edit route.**
Right-click on the route feature and click **Edit Route**.

Similar to the flanges we put in the route earlier, the generic pipe mount has an axis feature that is named `Mate`. This axis determines aspects of the bracket's orientation. When the generic mount is dropped onto the endpoint of a construction line, then the `Mate` axis snaps collinear to whatever the orientation of the construction line is in the 3D sketch.

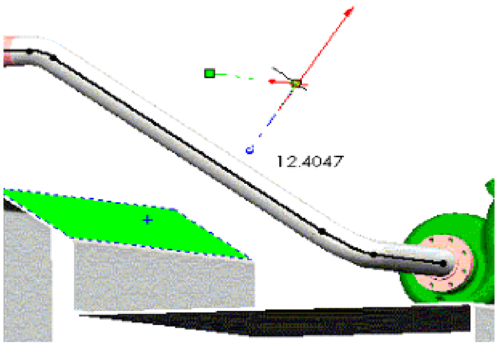
3 **Select sketch face.**
Select the angled face of the skid base to set the sketch plane in relation to it. We will now sketch two intersecting construction lines.

One construction line will be used to help snap the mount to the proper angle orientation.

The two lines create a plane that holds the rotation of the mount in relation to the run of pipe.

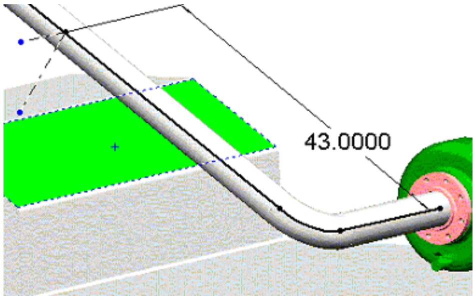
4 Sketch first construction line.
Sketch the first construction line so that it is perpendicular to the angled face of the skid base. The first construction line made will be the one the `Mate` axis of the mount will snap to.

5 Sketch second construction line.
Sketch a second construction line that is parallel to the angled face and perpendicular to the first construction line.



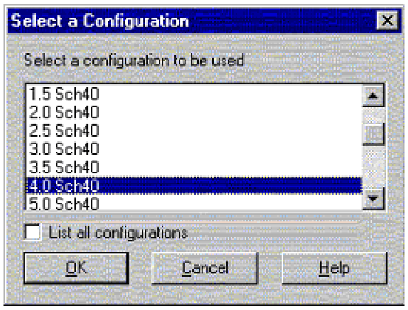
6 Add geometric relationship.
Add a coincident relationship to the intersecting point of those construction lines and the angled line of the pipe.

7 Position the construction line.
Position the construction line along the run of 4-inch pipe to 43.00. However, the length of the construction line is not critical.



8 Add pipe mount.
drag and drop from the Feature Palette (`C:StdLib\Piping Parts`), the `PIPE.MOUNT.GENERIC` to the intersecting point of the construction lines and the 4-inch run of pipe.

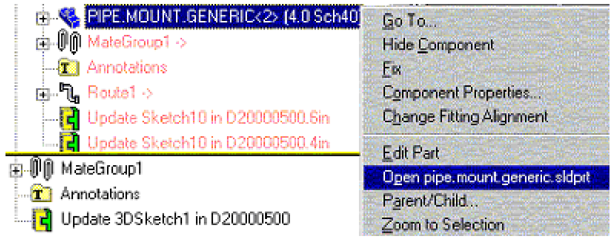
9 Configure pipe mount.
Like all piping parts we have added to the route, the generic mount has configuration size choices to make. Select the configuration `4.0 Sch40` (the mount fits the 4-inch pipe outside diameter cutout) and click **OK**.



10 Close the 3D sketch.

11 View FeatureManager design tree.
At this point, a read-only copy of `Pipe.Mount.Generic` should be the last part under the route assembly in the Feature Manager design tree. We want to start our mount design with this part, but in order for us to edit the mount and save the changes, we need to have write access to it. Let's change its name, so we can do this.

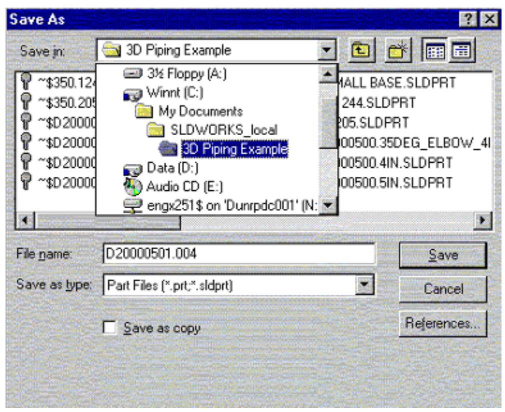
12 Open pipe mount part.
Right-click on `Pipe.Mount.Generic` and click **Open.**



13 Save pipe mount part to new file.
With the read-only mount open, click **File**, **Save As**.

Click OK button of the next dialog box.

Browse to the proper folder to save this part file in, and enter (make sure you use your next `D#.Location number`) `D20000501.004` as the file name and click **Save**.



Take note of the new name given to mount. There are several other configurations of this mount that we don't need in this part file. Delete the design table that is embedded in the part and delete all non-active configurations.

14 Close part.
Close `D20000501.004`, saving any changes that you've made.

15 Rebuild assembly.
Closing the part takes you back to the assembly. You should still be in the **Edit Route** assembly environment

Click **Rebuild** . Notice that the `Pipe.Mount.Generic` part gets replaced with the new mount.

16 Save your work.

Creating a Custom Mount

With the generic mount, comes a very generic shape. If this is not what you want your mount to look like, delete the `Pipe Mount` and create your own. You can create a custom mount because none of the integral features of this generic part (such as route points, axis feature, and so forth), are dependant upon this feature being in the part.

1 Edit pipe mount part.
Right-click on the mount (`D20000501.004`) and click **Edit Part**.

2 Delete unneeded file.
Delete `Pipe Mount` and its associated sketch.

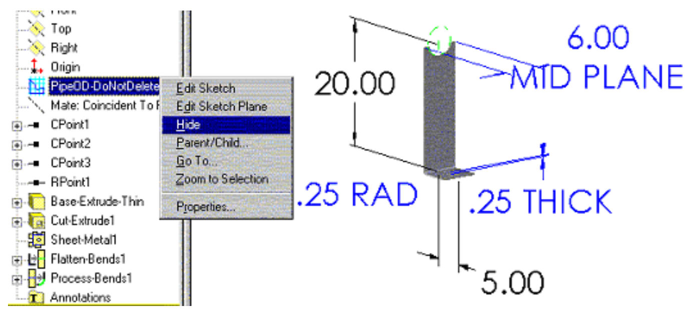
3 Start sketch.
Start a sketch on the `Right` plane of the mount. Sketch and constrain the appropriate 2D geometry as shown in the illustration.

4 Add geometric relationship.
Delete any in-context relationships that you don't want crossing sub-assemblies. Then add the dimensions.

Use a **Mid Plane** extrusion end type of 6.00 inches.

5 Add sheet metal features.
Add the **Sheet Metal** features with a.25 bend radius and holes in bottom flange, if desired.

6 Hide the sketch.
Right-click on the sketch `Pipe OD-Do Not Delete` and click **Hide.**



Forming Sub-Assemblies

After all of the piping design work is complete, you can break the route into manageable sub-assemblies. The **Form Sub-Assembly** command allows you to select components from the FeatureManager design tree and separate them into sub-assemblies.

To create a sub-assembly, you should be editing the upper level assembly — in this case `Small Layout 1`.

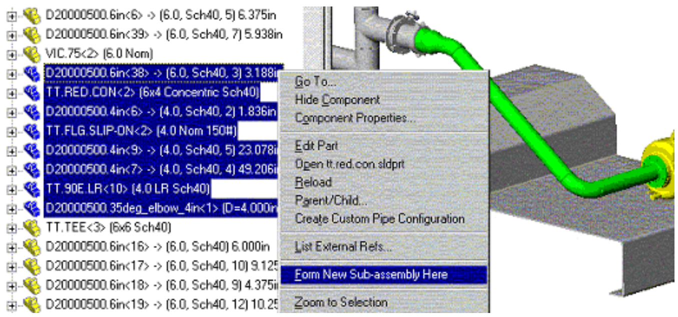
1 Determine which sub-assemblies to form.
Determine how many sub-assemblies are needed. Have those names ready. In this example, we will break this route assembly into five sub-assemblies. Name them as follows:

- D20000501
- D20000502
- D20000503
- D20000504
- D20000505

We will form the first sub-assembly, then you can do the other four on your own. The part location in the Feature Manager design tree may vary, but all of the parts should be there.

2 Select parts for sub-assembly.
From FeatureManager design tree, select all the parts between the Victaulic flex coupling (but don't include the coupling) up to, and including, the 4-inch flange. Include the mount in this sub-assembly.

3 Form sub-assembly.
With those parts highlighted, right-click in the highlighted (blue) area and click **Form New Sub-assembly Here**.

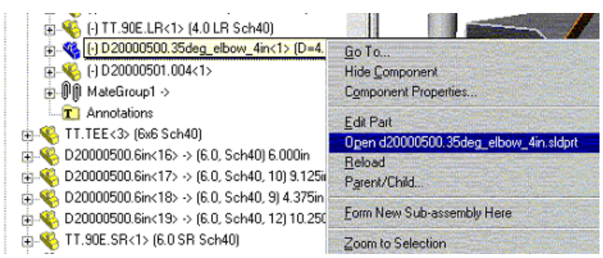


4 Save sub-assembly.
Browse to the folder that you want to save the file in, give the assembly file a name (`D20000501`), and click **Save**.

All the parts we selected are components of the sub-assembly. The mates associated to parts in this sub-assembly move also.

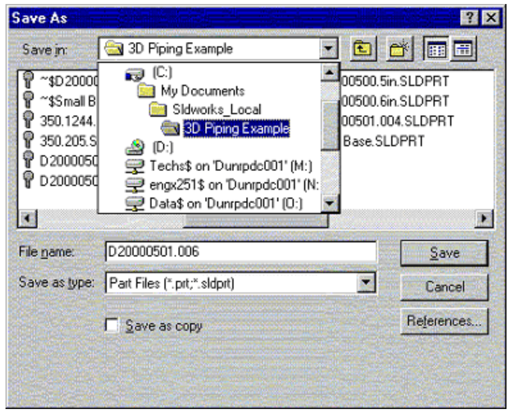
5 Name sub-assembly.
Now that we know what sub-assembly the 35 degree cut elbow is in, we can name it correctly.

Right-click the cut elbow in FeatureManager design tree (using the name we gave it earlier) and click **Open**.



6 Save
Click **File**, **Save As**.

Browse to the proper folder to save this part file in and enter `D20000501.006` as the file name, then click **Save**.

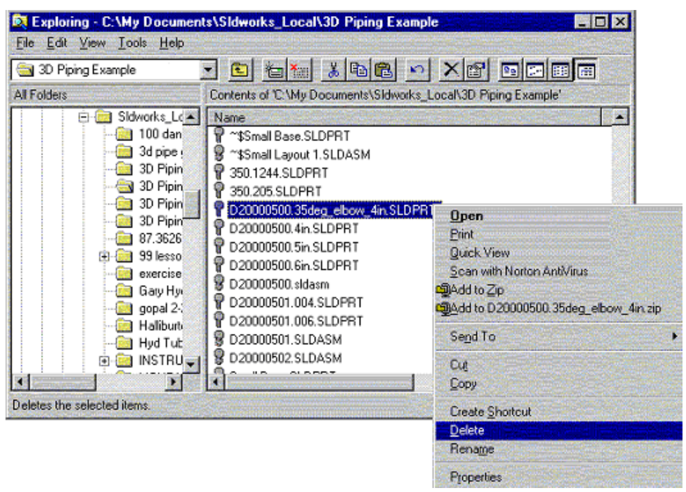


7 Close the part.
Close the `D20000501.006` part file window.

8 Rebuild the assembly.
Click **Rebuild** .

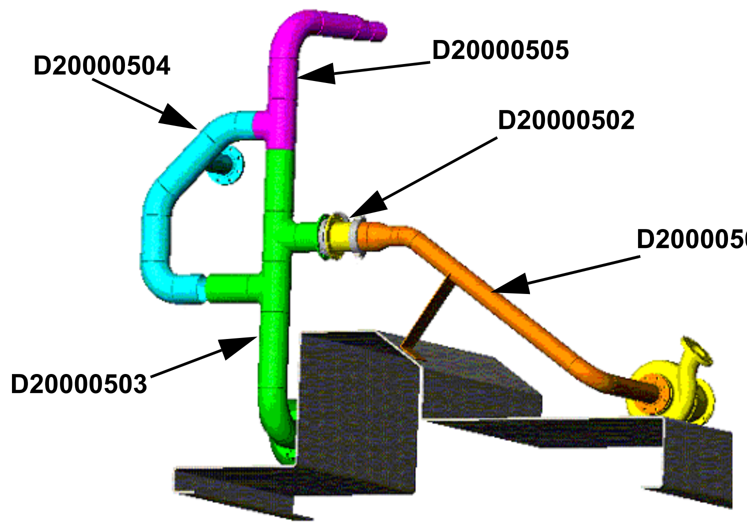
9 View the FeatureManager design tree.
Look in the Feature Manager design tree where 35 degree cut elbow is. See that part `D20000501.35DEG_ELBOW_4IN` is replaced with `D20000501.006`.

10 Delete old elbow.
Delete the old 35 degree cut elbow part file.



11 Save your work.
This sub-assembly is complete.

12 Form the other sub-assemblies.
Form the other four sub-assemblies the same way. Again, observe that the mates get moved into the proper sub-assembly. Group the parts together with the names shown in the illustration:



13 Verify sub-assemblies.
Once you're done, check that every part is in the proper sub-assembly by clicking on the sub-assembly icons (-) D20000503 in the Feature Manager design tree. When selected, all the parts that you've included in that sub-assembly will highlight green.

14 Save and close the assembly.

Adding Parts After Sub-Assemblies are Formed

Parts in the route assembly (and sub-assemblies) are controlled and positioned by the route or 3D sketch. To make changes to a sub-assembly, (such as moving, adding, or deleting parts) you must make those changes while in the 3D Sketch. When changes are made there, then associated geometry updates on the sub-assemblies, and drawings.

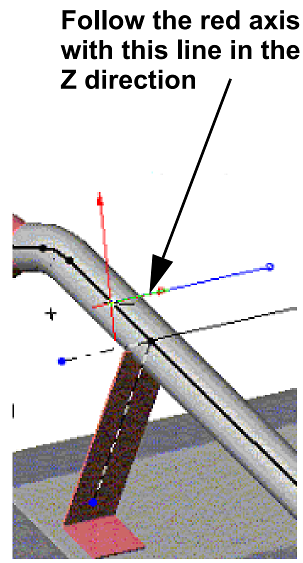
The software keeps you from making a mistake. Do the following *wrong* steps so you can see how the system deals with a user trying to make changes to a sub-assembly without going through the 3D sketch in the main route assembly.

1 Open sub-assembly.
Open sub-assembly `D20000501`.

2 View sub-assembly.
Notice the route is not present at the sub-assembly level. This keeps you from making the additions to the route and this sub-assembly.

3 Close the assembly.

4 Open upper level assembly.
Open the upper level assembly (`Small Layout 1`) to make the changes to sub-assembly `D20000501`.

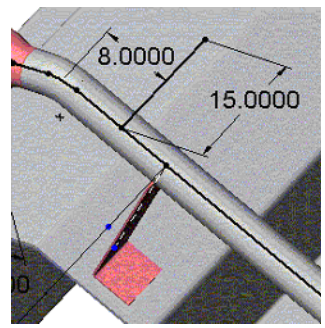


5 Edit route.
Right-click on the route and click **Edit Route**.

6 Split pipe and add line.
Select the angled 4-inch pipe, just above the bracket we put on earlier and click **Split Curve**.

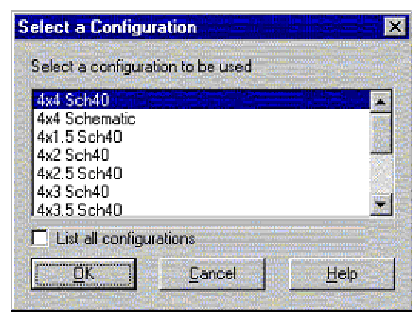
Sketch a line from that split point, in the **Z** direction (**Tab** to get the proper axis and watch the cursor feedback).

7 Add dimensions to line.
Dimension the line to position it and specify its length. Make it 15.00 inches long and positioned 8.00 inches down from the cut elbow as shown in the illustration.



8 Add tee to split point.
From the Feature Palette, drag and drop a `TT.TEE` onto the split point.

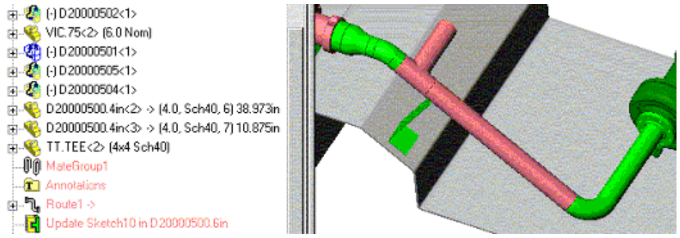
9 Configure tee.
Select the `4 x 4 Sch40` size tee from the Configuration dialog box. Click **OK**. This specifies the pipe size equal to that of the branch of the tee.



10 Resolve errors.
An error dialog box appears, just close this dialog box. These errors will be resolved once you exit the 3D sketch.

11 Exit the 3D Sketch.

12 Select sub-assembly.
From the Feature Manager design tree, select the sub-assembly `D20000501`.

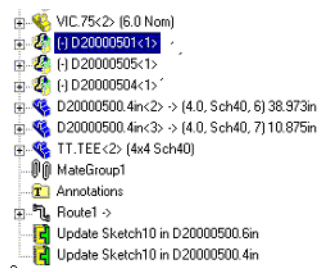


The one long angled piece of 4-inch pipe, is now short, and another piece of 4-inch pipe was added to the route (38.973 inches long, see it in FeatureManager design tree).

The new pipe, the 4-inch tee, and the 4-inch pipe that is coming out of the branch of the tee have all been added to the route assembly.

13 Add parts to sub-assembly.
Make sure that you are in the upper level assembly.

Drag and drop each of these parts into the sub-assembly `D20000501`.



14 Save and close the assembly.

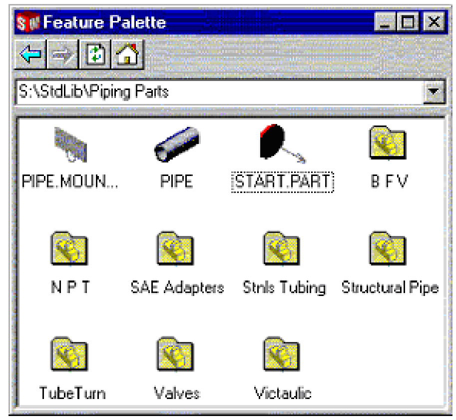
Adding Another Route Assembly

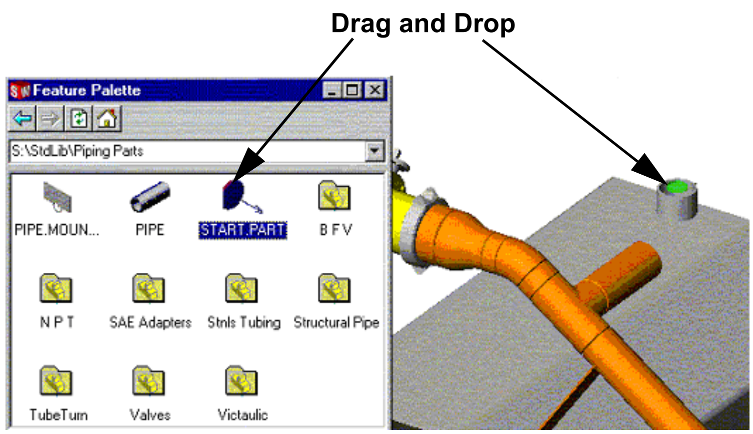
Up to this point, we have been designing a piping system all in one route assembly and breaking it up into manageable sub-assemblies. But many times, there are logical breaks in the piping systems. Examples of this might be that a unit has a suction piping system, a discharge piping system, air piping system, hydraulic piping system, and so forth.

If you see a natural break in the pipe routing system, you should consider whether or not another route assembly should be used. It is particularly recommended to create a new assembly for each bent tube.

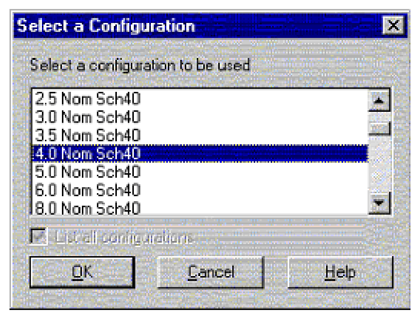
We are going to start a new route, coming out of the top of the 4-inch pipe, connected to the skid base. This route will be started with START.PART instead of a flange. Even so, the procedure is very similar. All piping parts have a route point, but the feature that allows the system to start a route by dragging from the Feature Palette, is when a part has **only** one connection point.

1 Add START.PART to assembly.
Drag and drop `START.PART` from the Feature Palette to the top edge of the 4-inch pipe. Pause over the edge, to allow `START.PART` to snap into place.





2 Configure START.PART.
Select the configuration of `START.PART` you want. This sets the default pipe and elbow size of this route assembly. Click **OK**.



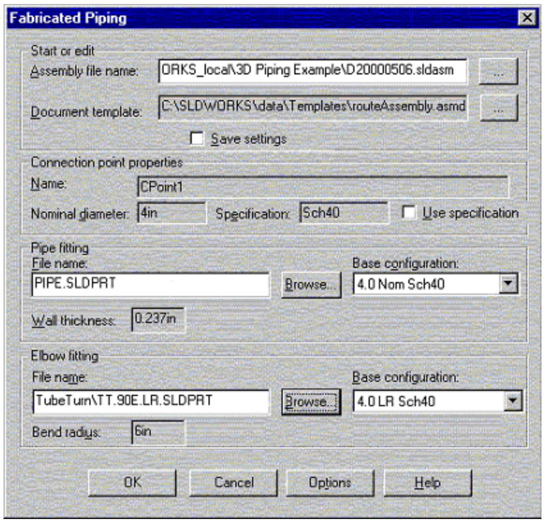
3 Name the new route assembly.
Name the route assembly. The next number in our series should be `D20000506`.

4 Specify the default pipe fitting.
Browse and select the path for the **Default Pipe Fitting**. The proper path is always `C:StdLib\Piping Parts\Pipe`, although the path gets abbreviated in the dialog box.

Make sure the base configuration (schedule of pipe) is correct.

5 Specify the default elbow fitting.
Browse and select the path for the **Default Elbow Fitting**. This path also gets abbreviated when entered in the dialog box.

Make sure the base configuration (schedule of pipe) is correct.



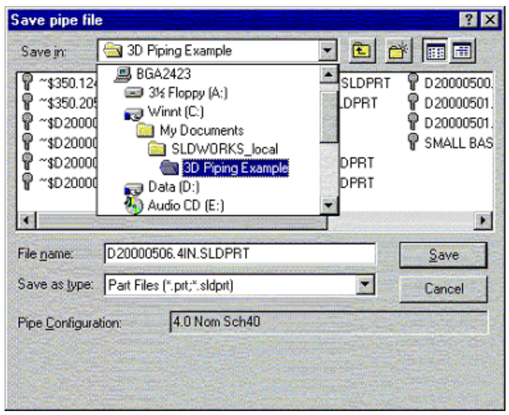
6 Accept options.
Click **OK** to accept the **Fabricated Piping** options.

7 Resolve errors.
Error markers may appear on the mates. These errors will be resolved once you exit the 3D sketch.

Because of the mate references on `START.PART`, there is only one more degree of motion left for you to mate the `START.PART` into the assembly.

8 Exit the 3D Sketch.

9 Name pipe sections.
The system prompts you to give a name to the straight pipe sections that will be in this route assembly.



Type in your “`route assembly D# dot pipe size`” in the **File name** field and before you click **Save**, make sure you are in the proper directory.

Take note of a couple of things as you exit the 3D sketch. The mating errors go away and piping parts that are in your route all appear in the FeatureManager design tree now.

10 Add geometric relations.
To stop the rotation of `START.PART`, add a Parallel relationship between the `Front` plane of the `START.PART` and the `Front` plane of the `Small Layout 1` assembly (the upper level).

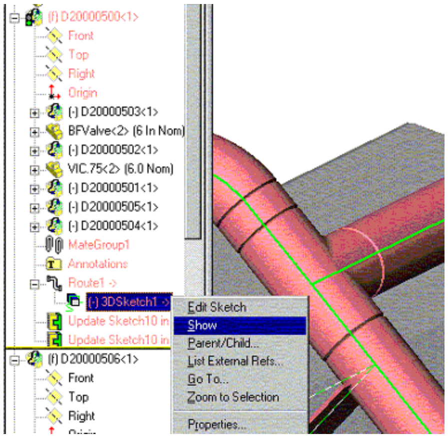
Notice that the minus sign is now removed from the START.PART in FeatureManager design tree, indicating that it is fully defined.

We were able to add this mate because the part that initializes a route assembly (in this case, START.PART) that is not driven by the route. All other parts in the route are positioned by the 3D sketch. If it were driven by the route, it would not be possible to add mates to position it.

11 Hide START.PART graphics.
Right-click START.PART and click **Hide Component**.

We needed START.PART to initiate the route, but — graphically — we don't want it showing up.

12 Show connection point.
Expand the route feature of the route assembly we made earlier, D20000500.

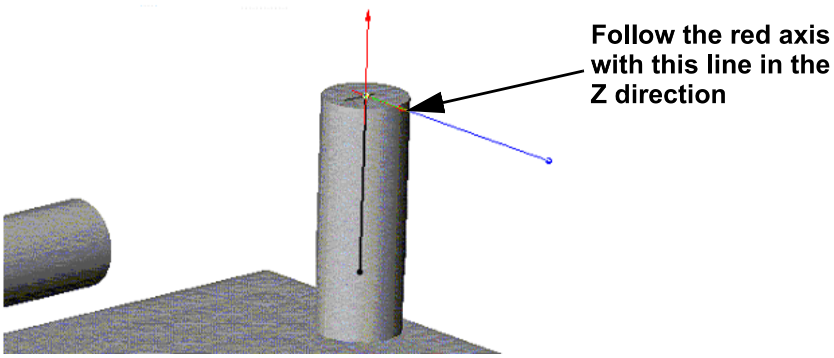


Right-click on the 3D Sketch that we made earlier, right-click connection point, and click **Show**. We will need to relate geometry to this sketch later.

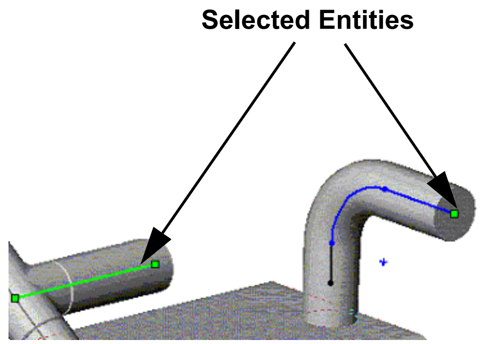
13 Edit route.
Right-click on the newly created route and click **Edit Route**.

14 Lengthen pipe.
Drag the endpoint of the pipe a little higher to lengthen the pipe. The exact length is not important at this time.

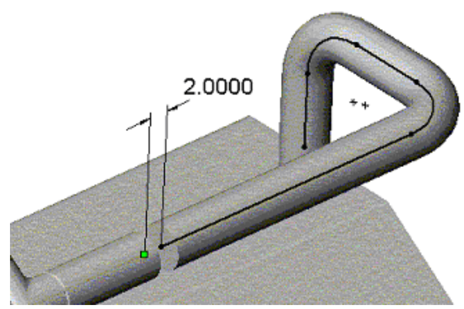
15 Add line.
From the endpoint of the vertical pipe, draw a line that is 90 degrees in the X direction (to the right). Watch the cursor feedback, making sure to stay in line with the red axis.



16 Add coincident relationship.
Select the endpoint of the line you just made and the line on the other route that you want to line up with. Add a coincident relationship between them. This makes the pipes line up horizontally and establishes the length of the last horizontal line you made.

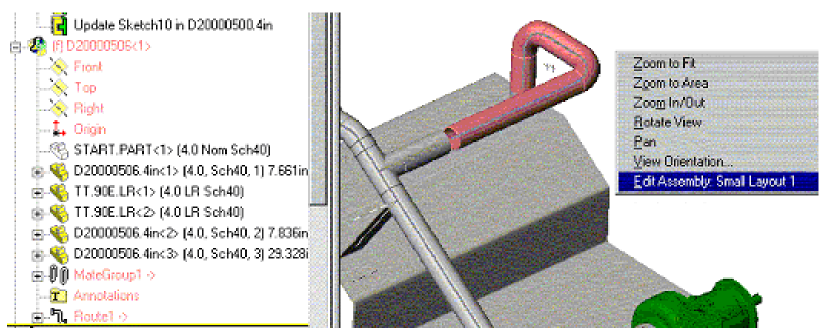


17 Complete the route.
Complete the new route by adding the final horizontal line and adding a 2.00 inch dimension between the endpoints of the two pipes.



18 Exit the 3D sketch.
Close the 3D sketch. Because there are no breaks in this route, there is no need to break it up into separate sub-assemblies.

19 Edit assembly.
Right-click in the graphics area and click **Edit Assembly:** `Small Layout 1`.



20 Hide routes.
Hide both the routes and save your work.

Lesson 5
Routing Tubing

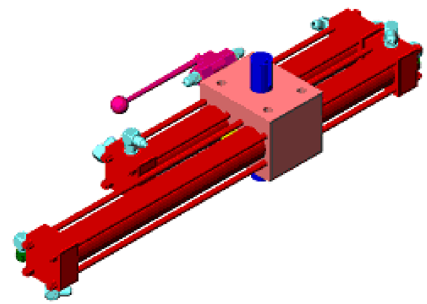
Upon successful completion of this lesson, you will understand how to route tubing.

Introduction

Routing hydraulic tubing and hose with SolidWorks is, for the most part, the same as routing welded pipe. Rather than routing the tube as a single component, tubes and hoses are actually pipe routes that contain special elbows and straight lengths of pipe that are designed specifically for routing tube. These special components can be found in `C:\StdLib\Piping Parts\Stnls Tubing`. Fittings have been created for tubing, hose, and pipe and can be found in various folders in `C:\StdLib\Piping Parts`.

In this lesson, we will use an assembly that already has the standard library fittings in place.

1 **Open assembly.**
Open assembly file `D00005358`, found in `C:\My Documents\Sldworks_Local\3D Piping - Routed Tubing`.



We will route three tubes. Since the two tubes going to the valve are the same, we will only route one of them and then copy it and mate the copy in the proper position.

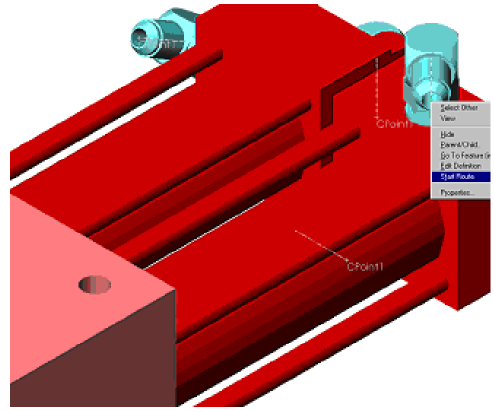
2 **View piping points.**
Click **View**, **Piping Points** to turn on all of the connection points in the various fittings.

The connection points are used for starting the route and also to mate the opposite end of the 3D sketch to end the route.

3 **Get part number.**
Before we start the route, we need to get the next definition object number from the matrix. To generate a definition object number, click **File**, **Save As Def Num**.

Important!

Be careful not to select **Save** when the **Save As** dialog appears, because you will rename the assembly.



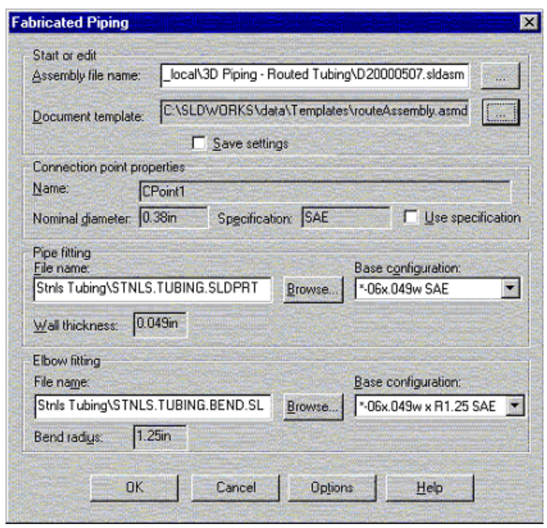
4 Start route.
Right-click the connection point on instance three of fitting `70.26732` and click **Start Route**.

5 Assign correct part name.
Highlight the default name that is assigned to the route and replace it with the new definition number in. fill out the route properties as shown below. Make sure the proper configuration is selected for each if it is applicable. The path for:

The pipe fitting is `C:\StdLib\Piping Parts\Stnls Tubing\STNLS.TUBING.SLDPRT`

The elbow is `C:\StdLib\Piping Parts\Stnls Tubing\STNLS.TUBING.BEND.SLDPRT`

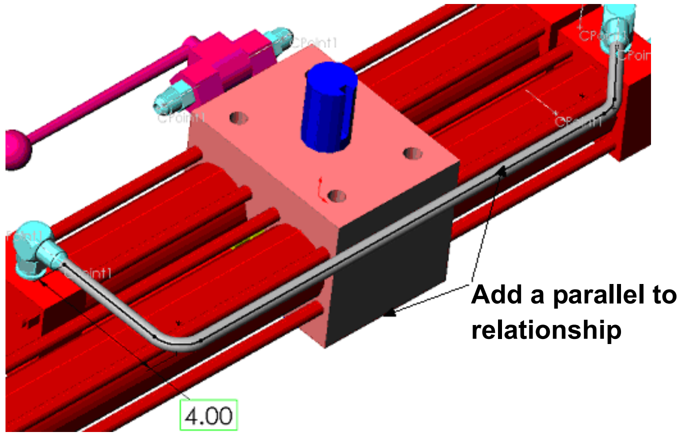
After the properties are correct, click **OK**.



6 sketch the route.

Note

At this time SolidWorks has some limitations with 3D sketches becoming over-defined. It may be necessary to delete the **Along Z** relationship from the middle (longest) sketch line and then make it parallel to a face on the actuator.



7 Add geometric relations.
Add a **Merge Points** relationship to the open ends of the lines and a **Parallel To** relationship between the route and the base assembly as shown in the illustration.

8 Add dimensions.
Add Dimensions as shown above.

The sketch is fully defined.

9 Exit the sketch.
Upon exiting the sketch, you are prompted to make a custom fitting for the bends. Name the custom fittings (bends) and the straight length of pipe. The naming convention to is similar to the naming convention used in pipe routing. Each custom fitting is named using the route assembly name 3-Digit number (although this is a continuos tube and this 3-Digit number doesn't represent a Find number at all). A straight length will be named in a similar fashion, the route assembly Name. Tube Size and SS (all CAPS) on the end of the name (as shown below). Also note that when routing hose, the hose size and HS will need to be placed on the end. If you haven't copied anything else to the clipboard, then you will be able to paste the definition number in when you save the route components and you will only have to add the text at the end. See examples below:

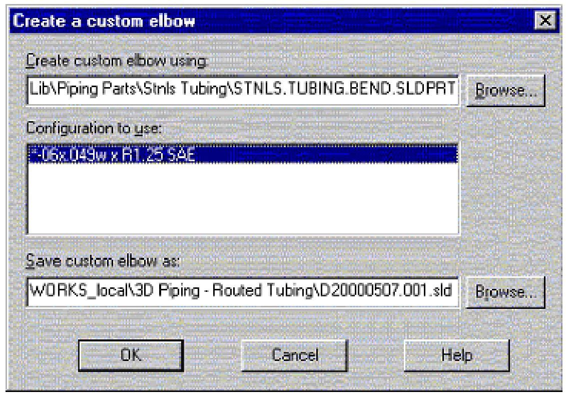
Route name…………………………………………..D20000507

Custom Fittings (bends in tube)………………….D20000507.001

D20000507.002

Straight Length Tube……………………………….. D20000507.06SS

Straight Length Hose……………………………….. D20000507.06HS



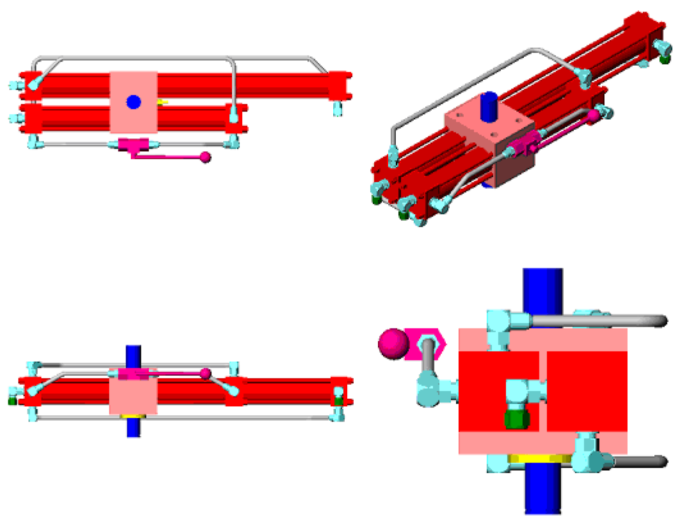
10 Edit the assembly.

After naming the components in the route, right-click in the graphics area and click **Edit Assembly.** This puts you in the upper level assembly.

11 Add additional routes.

Route the rest of the tubes as shown in the illustration.

Each tube is a separate route, with the exception to the tubes on the valve.



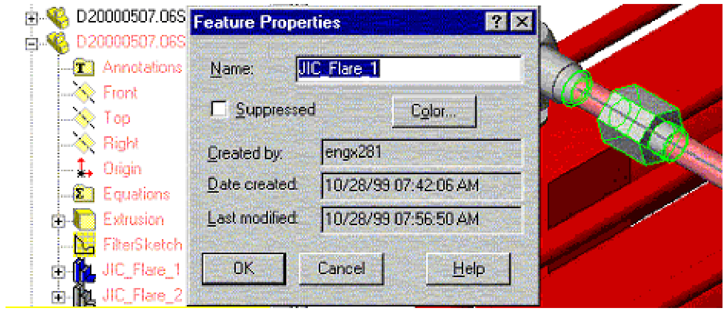
Each of the straight lengths of tube has features in them called `JIC_Flare_1` and `JIC_Flare_2`. When these are unsuppressed, the flare on the end of the tube, the sleeve, and the nut will be shown moved back on the tube so that the end of the sleeve is exactly at the start tangency for the minimum straight length. This is a good tool for checking to see if the straight length is too short.

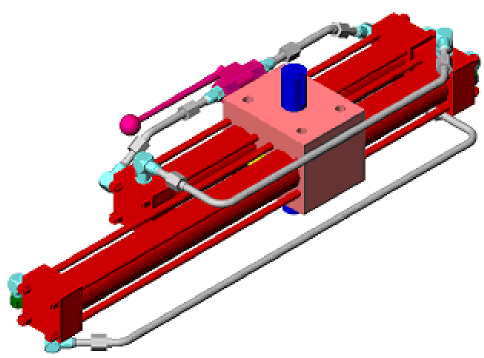
12 Create custom pipe configurations.

Right-click the tubes that need flares on their ends and click **Create Custom Pipe Configuration**.

13 Edit part.

Right-click on the same tubes and click **Edit Part** and click **Unsuppress** the `JIC_Flare_1` feature so that the tubes are as shown in the illustration.

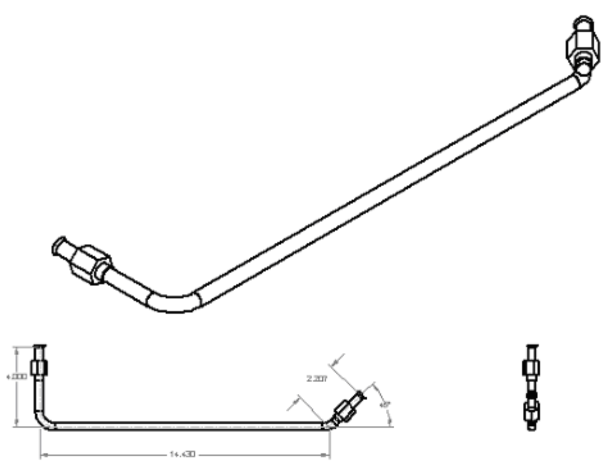




14 Hide the 3D Sketch.

15 Save the assembly.

As with any other assembly, the tube can be placed on a drawing and detailed to complete the process.



Lesson 6
Auto Mate

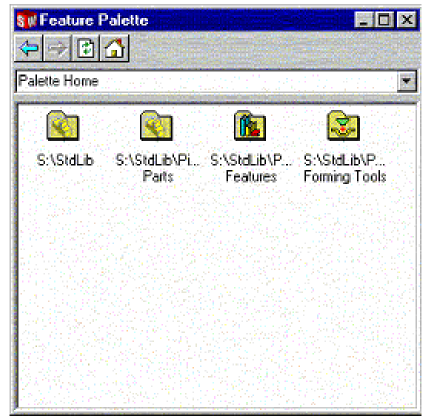
Upon successful completion of this lesson, you will understand Auto Mate Program – a quick and easy way to mate piping parts (fittings, adapters, and so forth) into an assembly.

Introduction

Auto Mate is a program that provides a quick and easy way to mate piping parts (fittings, adapters, and so forth) into an assembly.

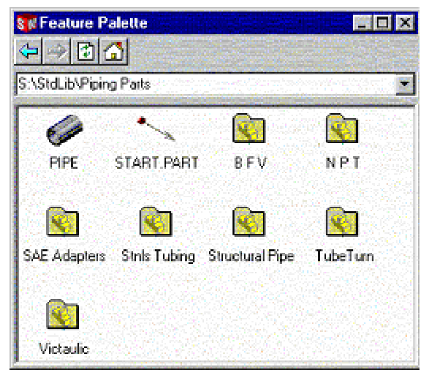
1 **Open the assembly.**
Open `Air Tank Assembly.SLDASM`, found in `C:\My Documents\Sldworks_Local\3D Piping - Automate Program`.

2 **Set Feature Palette path.**
Verify that the Feature Palette Parts path is set to `C:\StdLib\Piping Parts`.

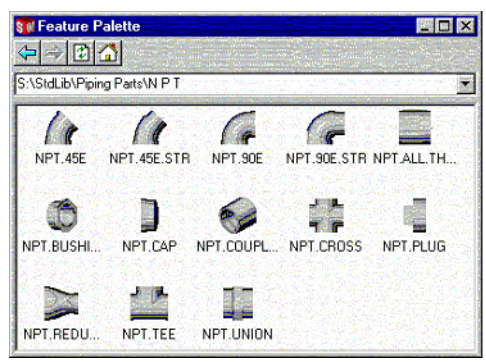


3 **Open Feature Palette folder.**
Open the folder `C:\StdLib\Piping Parts` and notice the categories of parts.

For this lesson, we will be using fittings from the `C:\StdLib\Piping Parts\NPT` folder.



4 **Select part.**
Select `NPT.45E.STR` from the Feature Palette. Drag the part into an assembly. Choose the configuration `1.0 NPT Sch40 Blk`.



5 **Mate the fitting and component.**
Mate this fitting to the component in the assembly.
This mate must be made manually as this fitting is being attached to a component in another assembly.

6 **View the elbow.**
Expand the 45° street elbow in the FeatureManager design tree.

Notice at the bottom of the elbow tree: `Pln1,Axs1,Pln2,Axs2`.

These features should be used for best results.

Select through the planes and axes to visualize which ports are represented by the planes and axes.

7 Open the air tank.
Open the `Air Tank` down to the angled coupling on the end of the tank.

Notice this coupling has the same planes/axes.

8 Mate the elbow and coupling.
Mate the Coupling Axs2 to 45° Street Ell Axs2 using Coincident.

9 Mate another elbow and coupling.
Mate the Coupling Pln2 to 45° Street Ell Pln2 using Coincident.

10 Define the rotation of the elbow.
Optional at this time. Note that the rotation of the 45° Street Ell has not been defined. Mate the Front plane of the Air Tank Assembly to the Front plane of the 45° Street Ell using 'Angle' and value of '0'.

11 Place fitting.
Select `NPT.ALL.THREAD` from the Feature Palette and drag this fitting into the assembly window and choose configuration `1.0 Sch40 Blk`.

12 Start the AutoMate application.

13 Specify file.
Click **File**, **Get Assembly Name**.

14 Select the elbow.
Select the 45° Street Ell from the FeatureManager design tree only and select Fitting 1.

Allow SolidWorks to open the fitting and retrieve the necessary data. When the data has been retrieved, SolidWorks will close the fitting. Please be patient — more ports per fitting will take longer.

15 Specify AutoMate settings.
Click **All-Thread**, **Accept Fitting 2.**

Click **Highlight Ports**. Click **Port** 2 option of **All Thread**. Click **Highlight Ports** again and notice that the opposite end of the **All Thread** is activated.

Click **Add Mate**.

Notice that the fitting may have assembled incorrectly. If it did, then use the **Undo** from the *AutoMate* application only (as this undo removes two commands).

16 Aligned option.
Use the **Aligned** option and click **Add Mate** again. The alignment conditions are the same as in SolidWorks.

17 Add fitting.
Select and drag `NPT.90E` from the Feature Palette into the assembly window. Choose configuration `1.0 NPT Sch40 Blk`.

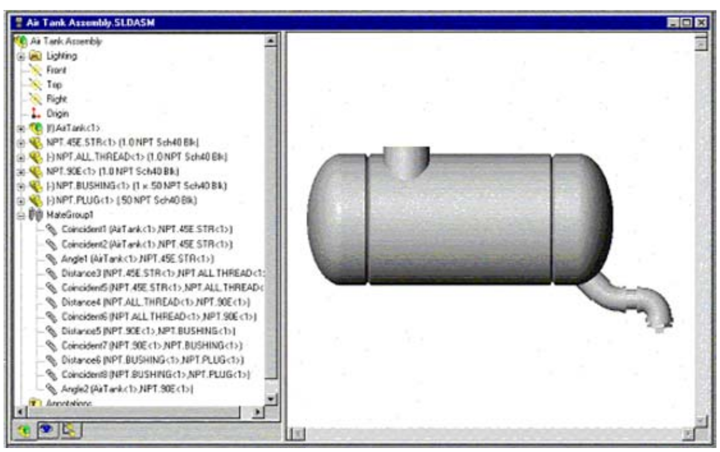
18 Add bushing.
Select and drag `NPT.BUSHING` from the Feature Palette into the assembly window. Choose configuration `1.0 x .50 NPT Sch40 Blk`.

Select and drag **NPT.PLUG** from the Feature Palette into the assembly window. Choose configuration **0.5 NPT Sch40 Blk**.

19 Select other fitting.
Select the `NPT.90E` from the assembly tree and select **Accept Fitting 1** from the AutoMate application. Allow AutoMate to retrieve the data for the `NPT.90E`.

20 Verify ports.
Verify the ports to be assembled using **Highlight Ports** and then **Add Mate**.

21 Mate rest of fittings.
Continue to assemble the remaining fittings in this manner.



Things to Remember:

Clear clears all the fitting data that has been retrieved including assembly name from local memory.

If the AutoMate application is exited, all retrieved fitting data is lost.

Rotation of fittings should be constrained by adding an angle mate either to the previous fitting or component feature per design intent.