CADWorx

Plant Tutorial

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Preface for CADWorx Plant Tutorials

This document provides step-by-step instructions designed to get you up and running quickly with CADWorx Plant and the new features. The individual lessons that make up this tutorial provide hands-on practice using the new features in CADWorx Plant.

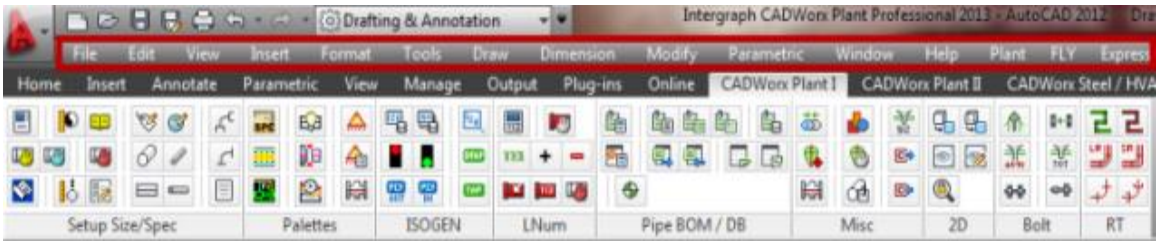
We welcome comments or suggestions about this documentation. You can send us an email at: PPMdoc@intergraph.com.

SECTION 1

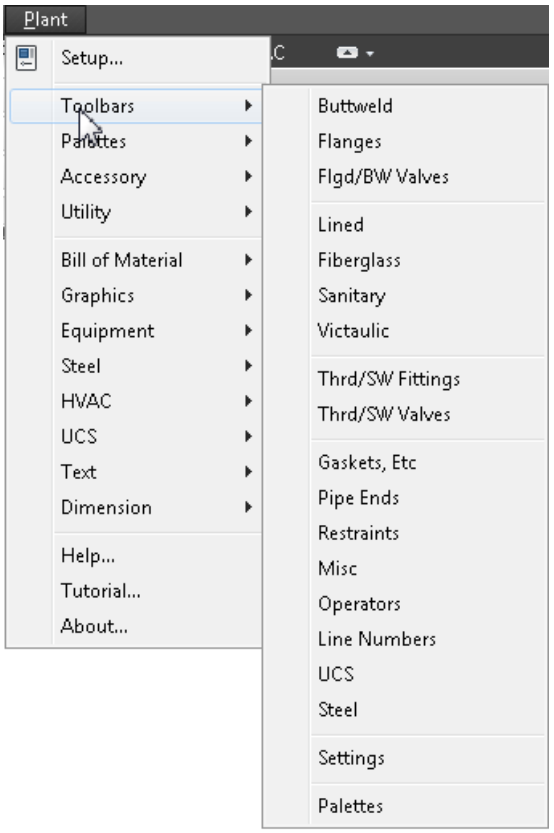
CADWorx Plant Tutorial

A solid working knowledge of AutoCAD is required to complete the lessons in this tutorial. AutoCAD is delivered with a comprehensive online Help library. To access this library, press **F1** while a menu item is highlighted. If the AutoCAD setvar **TOOLTIPS** is set to 1, CADWorx Plant displays a descriptive tooltip about the command when you rest the mouse pointer on top of a command icon.

The lessons in this tutorial make extensive use of toolbars and commands that are available only on the AutoCAD **Plant** menu. Consequently, you must have AutoCAD menus on while you are working a tutorial lesson. To turn on AutoCAD menus, type **MENUBAR** on the command line, and then type **1**. The AutoCAD menu bar is outlined in red in the figure below.



The quickest way to access CADWorx Plant toolbars is to click **Plant > Toolbars**, and then select a toolbar in the list. The toolbars displays in the drawing environment, where it remains until it is removed.



You can also dock toolbars anywhere you prefer. For information on docking toolbars, refer to the AutoCAD documentation.

All references to a menu command are accompanied by a reference to the AutoCAD command line entry. These commands are just for reference. Aliases for all these commands are listed in the *[Product Folder]*\Support\Pipe_alt.pgp file. For more information about command aliases, see *Aliases and Command Names* in the *CADWorx Plant User's Guide*. Additionally, if a command can be accessed using one of the CADWorx Plant-specific panels instructions for doing so are also included.

A set of sample drawing files, for both English and metric units, accompanies each lesson. Optionally, you can use the tutorial drawing file that corresponds to the lesson, or you can save your drawing when you complete a lesson so that it can be used in the next lesson. If you intend to use the tutorial drawing files, we recommend that you make a backup copy of the sample files so that they can be used multiple times within different sections of the tutorial.

NOTES

- Tutorial lessons can be completed using metric or English units. References to English units that appear in a lesson are immediately followed by their metric counterparts in parenthesis, such as 4" *(102mm)*.
- Throughout this tutorial, it is assumed that the working folder is *[Product Folder]*\Tutorial. If you work *CAESAR II and CADWorx data transfer* (on page 41), we recommend that you use the *[Product Folder]*\Tutorial folder as well.

Set drawing parameters

Each time you begin a new drawing you must use the **CADWorx Plant Setup** dialog box to define drawing parameters as required for the current project. For more information about setup options, see *Setup Size/Spec Panel* in the *CADWorx Plant User's Guide*.

1. Click **Setup** on the **Settings** toolbar. Alternatively, type **SETUP** on the command line, and then press ENTER.

   TIP You can also click the **CADWorx Plant I** tab, and then click **Setup** on the **Setup Size/Spec** panel.

   *The **CADWorx Plant Setup** dialog box displays.*
2. Select **3D Solids** in the **Drawing Mode** list.
3. Select **Socket** in **Fitting Mode** list.
4. You can change the colors of the **Compass**, the **Dimension,** or the **Highlight**. The default color is red. For more information on the compass, see *Compass* in the *CADWorx Plant User's Guide*.
5. Select **Configuration Settings.**

   *The **Configuration Settings** list displays.*
6. Under **Configuration Settings**, make sure **SystemMeasure** is set to **English/Inch** (Metric/Metric). For more information, see *Configuration Settings*, and *Startup Variables* in the *CADWorx Plant User's Guide*.
7. Select **Specification / Size**.

   The Set Specification and Size dialog box displays.

   TIP This dialog box can also be found by clicking **Main Size** on the **CADWorx Plant I** tab on the **Setup Size/Spec** panel. This is the same dialog box except that the name of the dialog box does not appear in the **CADWorx Plant Setup** dialog box.
8. Click **Browse**, and then navigate to the CADWorx **Spec** folder.
9. Select **Sample_Inch.prj** *(Sample_Metric.prj)* in the list of available project specs, and then click **Open**.

   *The specifications display in the **Specifications** box.*
10. Select **150** (***150_mm***).

    *The sizes display in the **Main / Reduction size** box.*
11. Select **4"** (***100***) under **M** for the main and then **2"** (***50***) under **R** for the reduction.

    A check-mark appears next to each selected size.
12. Select **Configuration Layers**.

    The layers list displays.

    NOTE These are the colors and linetypes for the components.
13. Leave the layers as they are.
14. Select **Piping Rules**.

    *The **Piping Rules** list displays.*
15. Select **Apply Weld Insertion Rule**, and then change the list to **Automatic - Buttweld and Socket Weld**.
16. Select **Apply Gasket Insertion Rule**, and then change the list to **Automatic**.
17. Select **Apply Bolt Insertion Rule**, and then change the list to **Automatic**.

18. Click **Apply & Close** on the **CADWorx Plant Setup** dialog box.
19. Click **File** > **Save As**.
20. Type **Plant1** in the **File name** box, and then click **Save**.

    By default, the file is saved in AutoCAD 2010 Drawing (.dwg) format.*

NOTES

- To modify nominal size data, click **Main Size** on the **Settings** toolbar.
- To modify the assigned specification, click **Specification** on the **Settings** toolbar.
- You can also access **Main Size** on the **Setup Size/Spec** panel when the **CADWorx Plant I** tab is selected.

Route Modeling

1. Do one of the following:
   - Open the Plant1.dwg drawing file that you saved in *Set drawing parameters* (on page 9).
   - Open the *[Product Folder]*\Plant\Tutorial\Tutor1.dwg file delivered with the software.

     NOTE To create a drawing based on metric units, open the *[Product Folder]*\Plant\Tutorial\Tutor1m.dwg file.
2. Click the **View** tab, and then select **SW Isometric** in the **Views** palette.
3. Click the **CADWorx Plant I** tab, and then click **Auto Route** on the **RT** panel. Alternatively, type **AUTOROUTE** or **ROUTE** on the command line, and then press ENTER.

   Specify start point or [Alignment/Reference] <last point>.
4. Type **0,0** on the command line, and then press ENTER.

   Pick end point or [component List /Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].
5. Type **@48,0** *(***@1219,0***)*, and then press ENTER.

   TIP English units are mentioned first followed by their metric counterpart in parentheses. Only type in one set of units: @48,0 or @1219,0.
6. Right-click and select **component List**.

   The components list displays.
7. Select **Elbows** > **[4"** (***100***)**] ELL, 90 LR S/STD, ASTM A234 GR WPB**.

   Pick other direction.
8. Move the cursor to the left (going north in the screen) so the elbow is pointing in that direction, and then click. There is more pipe showing than is needed, but the next step will clean this up.
9. Right-click and select **component List**.

   The components list displays.
10. Select **Flanges** > **[4"** (***100***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.

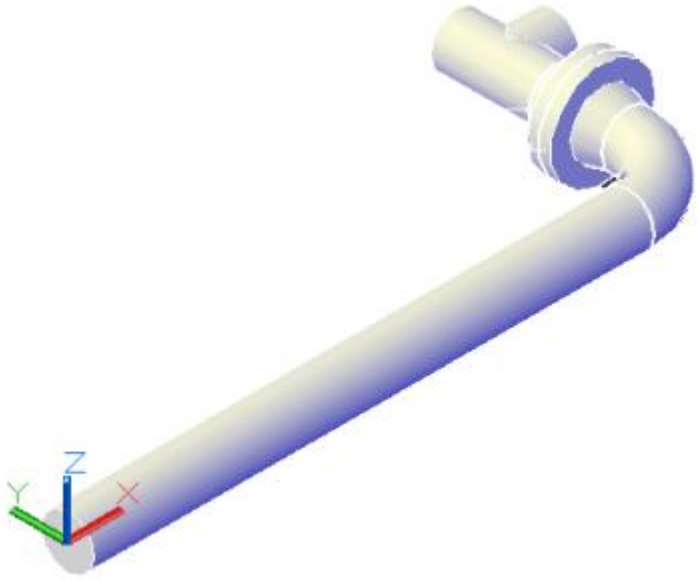
    The flange is placed.
11. Right-click and select **component List**.

    The components list displays.
12. Select **Flanges** > **[4"** (***100***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.

    The flange, bolt, and gasket are placed.
13. Right-click and select **component List**.

    The components list displays.

14. Select **Tees** > **[4"** (***100***)**] TEE, STR. S/STD, ASTM A-234 GR WPB**.
    Pick direction or [Change connection].
15. Select a direction to the east, and then click.
    Your drawing should resemble the drawing below. Conceptual view was used for clarity. Your drawing appears in wireframe.

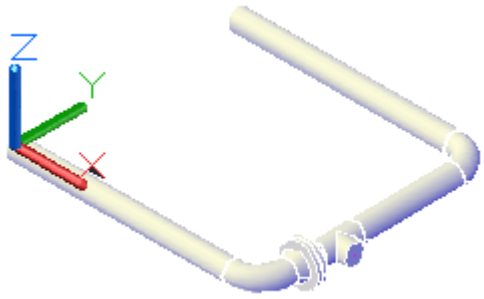


    The tee is placed, and the program places you back at the north point of the model on the other end of the tee.

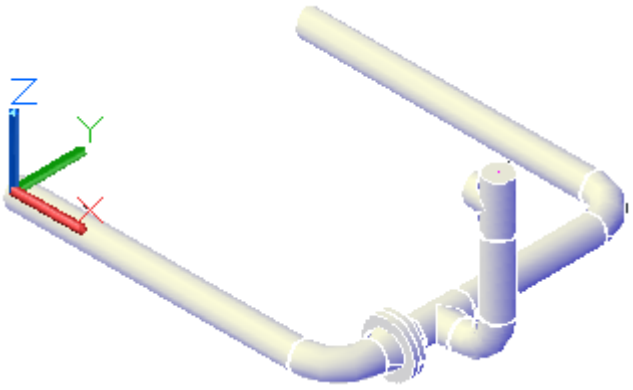
You will now continue routing to finish the model.

1. Type **22** (***558***) on the command line, and then press ENTER.
   The piping shows up coming off the main of the tee.
2. Right-click and select **component List**.
   The components list displays.
3. Select **Elbows** > **[4"** (***100***)**] ELL, 90 LR S/STD, ASTM A234 GR WPB**.
   Pick other direction.
4. Select the direction to the west to parallel this elbow with the previous elbow.
   The elbow is placed.
5. Type **.X** on the command line, and then press ENTER.
   of.
6. Select the end of the first pipe line that you drew.
   of (need YZ).
7. Click west of the new pipe, and then press ENTER.
8. Click the **View** tab, and then select **SE Isometric** in the **Views** palette.

Your drawing should look like this.



9. Click the branch of the tee, and then select the plus sign.
10. Right-click and select **component List**.
    The components list displays.
11. Select **Elbows** > **[4"** (***100***)**] ELL, 90 LR S/STD, ASTM A234 GR WPB**.
    Pick other direction.
12. Pick a direction upward.
    The elbow appears, and depending on where your cursor is placed, some piping might appear.
13. Type **12** ***(305)*** on the command line, and then press ENTER.
14. Right-click and select **component List**.
    The components list displays.
15. Select **Tees** > **[4"** (***100***)**] TEE, STR. S/STD, ASTM A-234 GR WPB**.
    Pick direction or [Change connection].
16. Pick a direction to the west (parallel with) the elbow below.
    Your drawing should look like this.

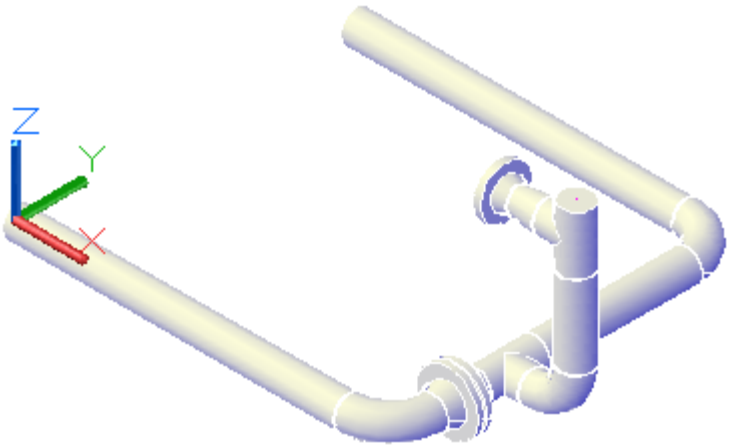


When you insert a reducing component--such as the concentric reducer--the main size and the reduction size default large to small providing the proper connection at the reducer. After inserting a reducing component the main size and the reduction size reverse. This allows placement of other components on the pipe run without having to reset the sizes.

1. Click the branch of the tee, and then select the plus sign.
2. Right-click and select **component List**.

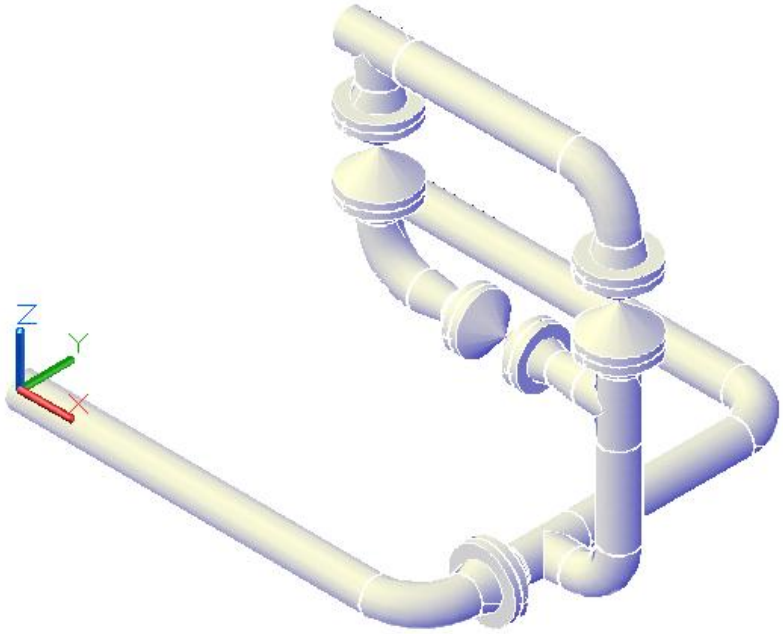
The components list displays.

3. Select **Reducers** > **Concentric Reducing Buttweld** > **[4" X 3"** (***100 X 80***)**] REDUCER, CONC S/STD, ASTM A-234 GR WPB**.
4. Right-click and select **component List**.
   The components list displays.
5. Select **Flanges** > **[3"** (***80***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.



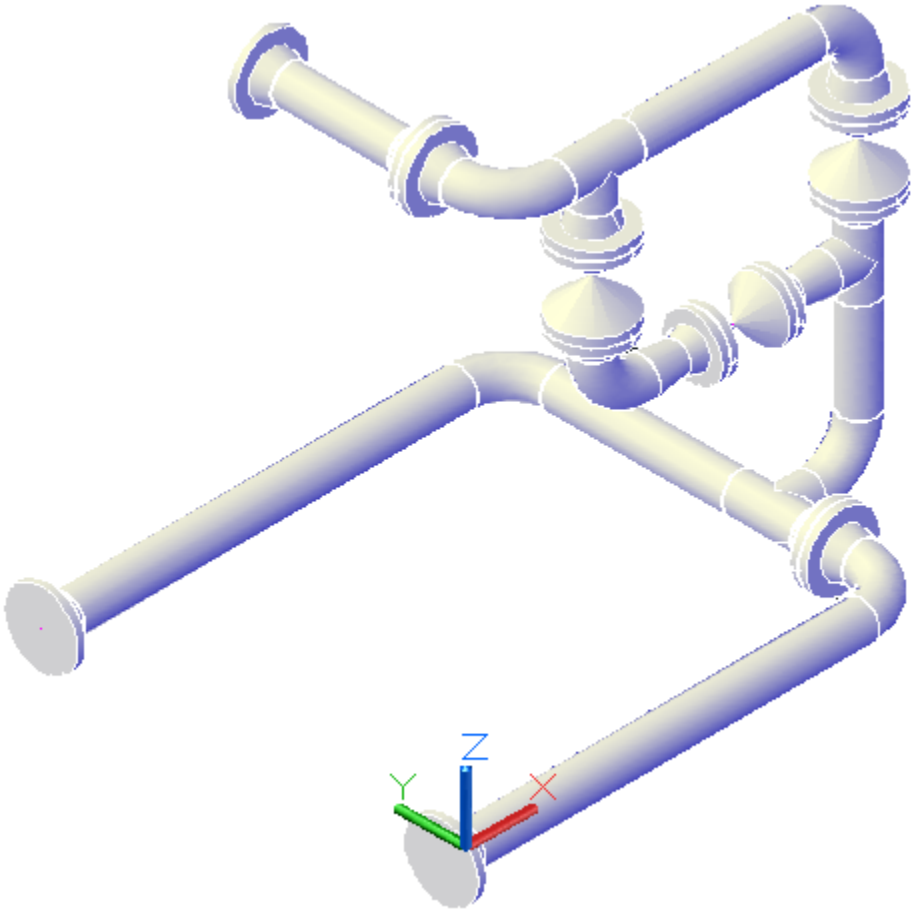
The main size is now set to 3" (80), and the reduction is set to 4" (100).

6. Continue to model the remaining components--the gate valves, elbows, and additional piping-- as shown below. The Reducer is 4" X 3" and we need to use the **Change connection** option to place the branch of the second Tee as it appears in the drawing.



NOTE To further improve the view of the drawing, use the **View > 3D Views > SW Isometric** command and give the drawing another viewpoint.

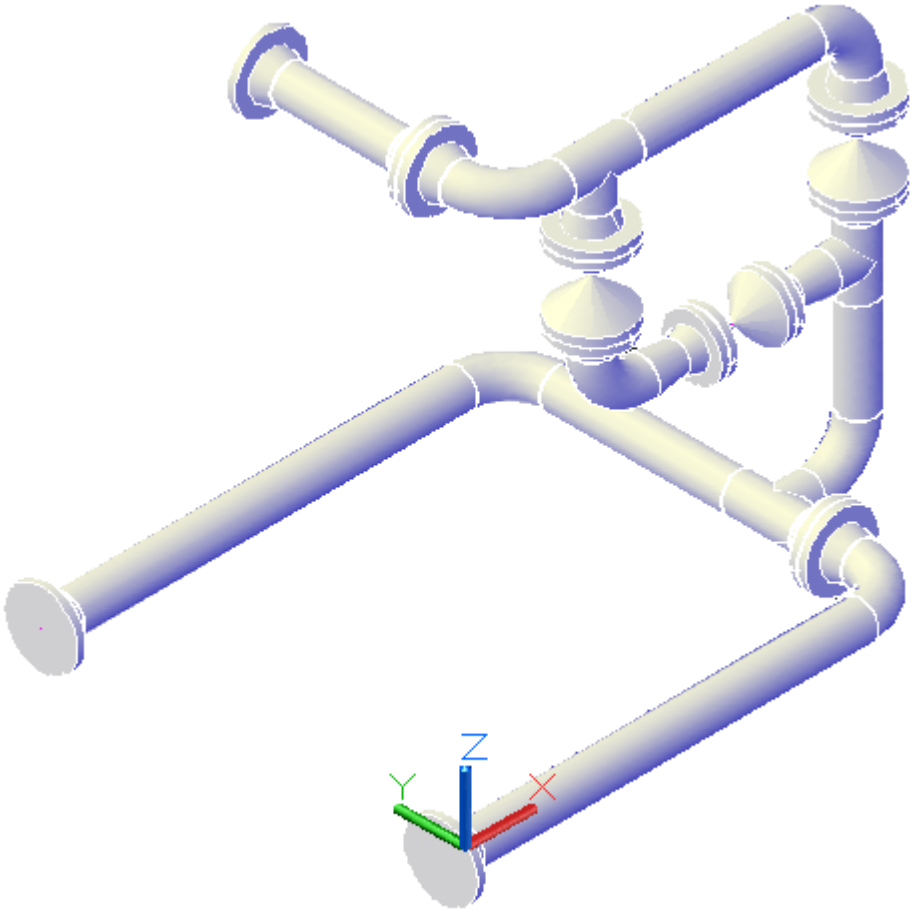
Your finished piping system in 3D space should look like this.



7. Click **Save As** . Alternatively, type **SAVEAS** on the command line, and then press ENTER.
   *The **Save Drawing As** dialog box displays.*
8. Type a name for the drawing, and then click **Save**.

Mode conversion

Mode conversion converts components from one mode to another mode or to the same mode. Mode conversion can also restore a component to its original state after the component has been modified. In *Route Modeling* (on page 10), you created the 3D Solids model shown below. In this lesson, you will convert that model to a 2D Double Line. Mode conversion also shows you where all your bolts, gaskets, and welds are located with easy viewing.



1. To define the drawing mode, click **Setup** on the **Settings** toolbar. On the **CADWorx Plant Setup** dialog box, select the required option under **Drawing Mode**. Alternatively, type **SETUP** on the command line, and then press ENTER.
2. Do one of the following:
   - Open the drawing you completed in the lesson *Route Modeling* (on page 10).
   - Open the *[Product Folder]*\Tutorial\Tutor2.dwg file delivered with the software.

     NOTE To work in metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor2m.dwg file.
3. Click the **CADWorx Plant I** tab, and then click **2D Double Line** on the **Setup Size/Spec** panel. Alternatively, type **CONVERTDOUBLE** on the command line, and then press ENTER.

   Select components to convert to double line mode.
4. Pick a point in the upper right corner of the drawing.

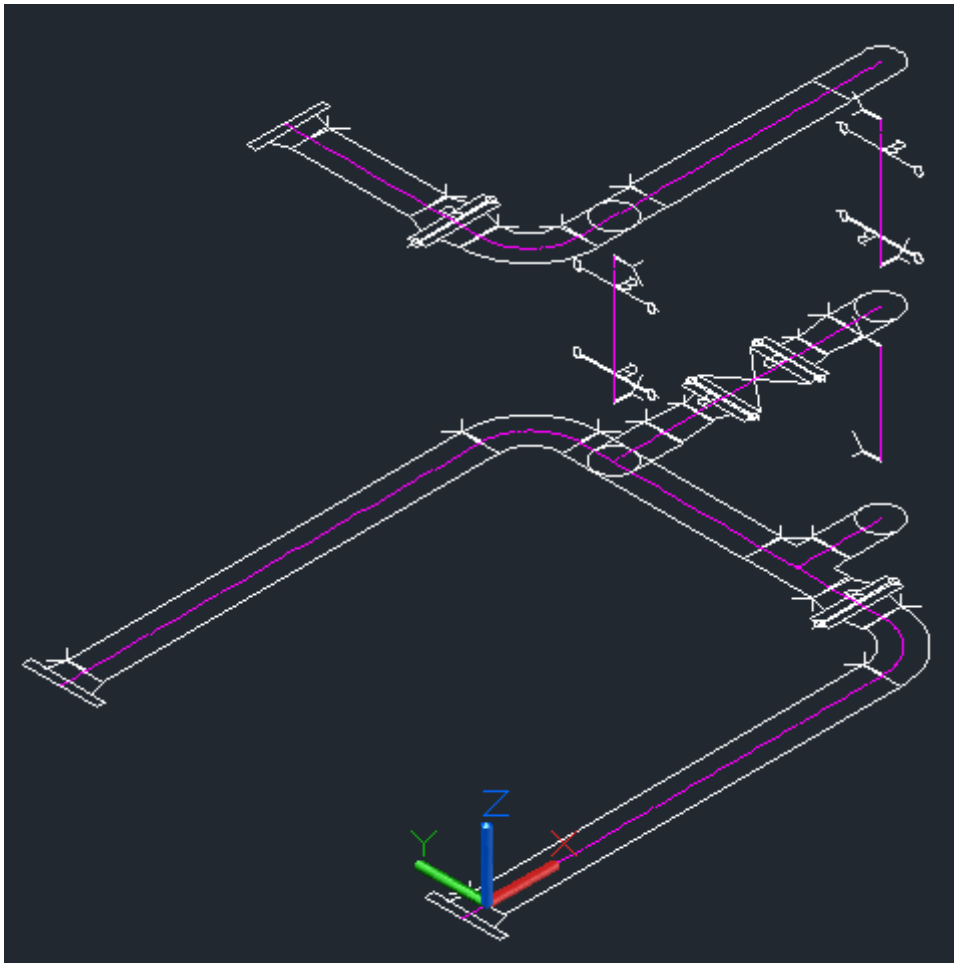
Select components to convert to double line mode: Specify opposite corner.

5. Pick a point in the opposite corner to create a select set that contains all of the model components.

   Select components to convert to double line mode: Specify opposite corner: 73 found, 73 groups.

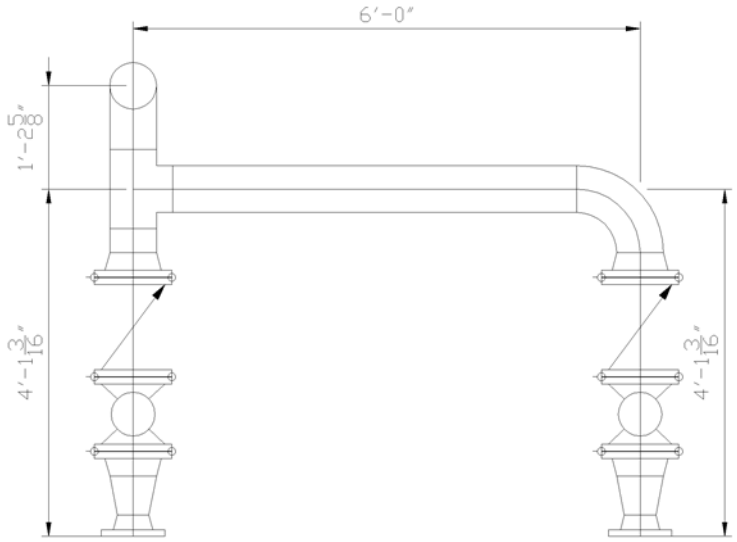
6. Press ENTER to end the selection.

   CADWorx converts the model to 2D Double Line.



Assembly Modeling

In this lesson, we will create discharge piping for two pumps using the **Assembly Manager** palette. Although we can draw the piping anywhere, we recommend that it be drawn using the **NE Isometric** view in the tutorial drawing file. Doing so allows for easy placement of the pumps. For more information on the **Assembly Manager**, see *Assembly View Palette* in the *CADWorx Plant Users Guide*.



1. Open the *[Product Folder]*\Tutorial\Tutor3.dwg file.

   NOTE To work in metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor3m.dwg file.

   The software opens the drawing, which already has a three-dimensional pipe rack outline, a set of pumps, a vessel, and a tank. We will use this same drawing to connect the assembly to the pumps when finished.

2. *Set drawing parameters* (on page 9) as defined below.
   - **M** = 4" *(100mm)*
   - **R** = 6" *(150mm)*
   - **Specification** = 150 *(150_MM)*
   - **Drawing Mode** = 3D Solids
   - **Piping Rules**
     - **Apply Gasket Insertion Rule** = Automatic
     - **Apply Bolt Insertion Rule** = Automatic
     - **Apply Weld Insertion Rule** = Automatic - Buttweld and Socket Weld

3. Click **Weld Neck** on the **Flanges** toolbar to place a 4" (100) flange. Alternatively, type **FLGW** on the command line, and then press ENTER.

   FLG, RFWN 150LB S/STD BORE, ASTM A-105

   Pick start point or [Buttweld end] <last point>.

4. Select a point in the drawing to place the flange.

   Pick direction.

5. Select a direction upward.

6. Click the newly placed flange, and then select the plus sign on the face end.

   Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].

7. Right-click and select **component List**.

   The components list displays.

8. Select **Reducers** > **Concentric Reducing Buttweld** > **[6" X 4"** (***150 X 100***)**] REDUCER, CONC S/STD, ASTM A-234 GR WPB**.

   The main and reduction sizes automatically reverse, allowing placement of the next flange without changing the pipe sizes (main size now equals 6" (150) and reduction size equals 4" (100)). Anytime a reducing component is encountered, the last size that was drawn is automatically set to the main setting (4" (100) small end to 6" (100) large end, 6" (100) large end drawn last so 6" (100) is used).

   Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].

9. Right-click and select **component List**.

   The components list displays.

10. Select **Flanges** > **[6"** (***150***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.

    Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].

11. Right-click and select **component List**.

    The components list displays.

12. Select **Flanges** > **[6"** (***150***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.

Using **ROUTE,** follow the steps below to draw a 6" (150) ball valve and a 6" (150) check valve. After drawing the 6" (150) flange onto the check valve, copy the existing piping at a distance of 6'-0" *(1829mm)* using the **Copy** command.

1. Right-click and select **component List**.

   The components list displays.

2. Select **Valve** > **Valves, Ball** > [**6"** (***150***)**] BALL VALVE, 150LB FLG**.

   Pick end point or [component List / Slope / Elevation / Plane / Reference /Fitting mode / Undo / Connect / Toggle length / Alignment].

3. Right-click and select **component List**.

   The components list displays.

4. Select **Valve** > **Valves, Check** > [**6"** (***150***)**] CHECK VALVE, 150LB FLG**.

   Pick direction or [Flip direction].

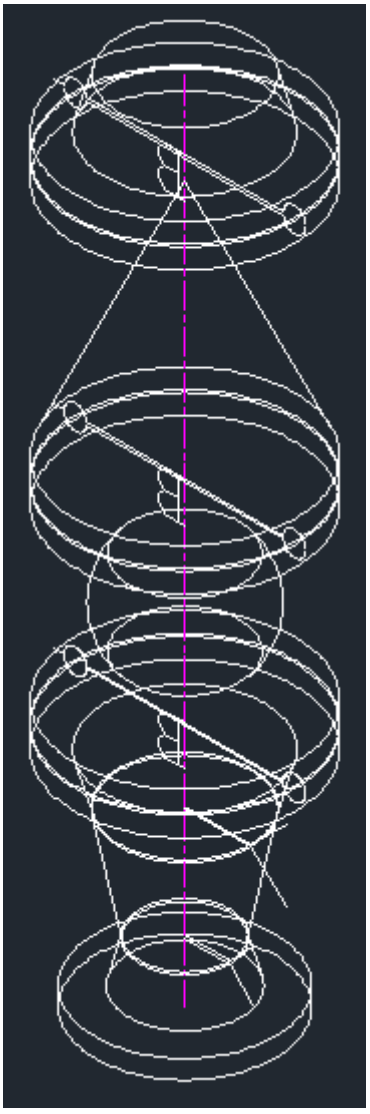
5. Pick a direction upwards.

6. Right-click and select **component List**.

   The components list displays.

7. Select **Flanges** > **[6"** (***150***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.

   Your image should look like the drawing below.



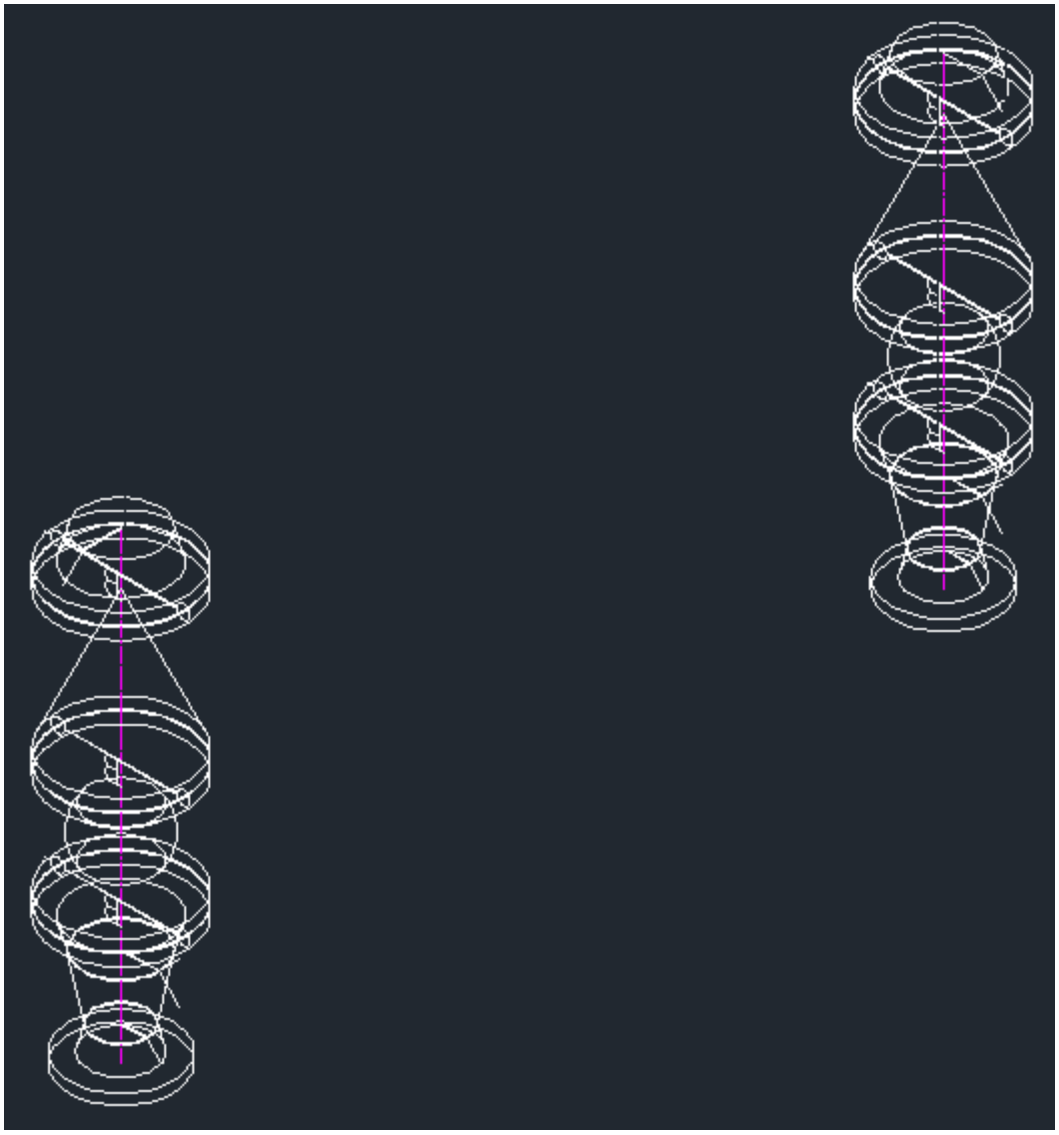
Next, we are going to *Copy the valve relay* (on page 19).

Copy the valve relay

Now we copy the valve group and place it 6' *(1829)* to the right.

1. Type **COPY** on the command line, and then press ENTER.
   Select objects; Specify opposite corner.
2. Pick a window around the components.
   Specify opposite corner: 14 found, 14 groups
   Select objects.
3. Press ENTER to finish the select set.
   Specify base point or [Displacement/mOde] <Displacement>.
4. Pick the center base of the bottom flange.
   Specify second point or [Array] <use first point as displacement>.

5. Drag the mouse to the right, type **6'** ***(1829)*** on the command line, and then press ENTER.
6. Press ENTER when finished.
   The copy appears in the drawing. Zoom out if you cannot see it.



Next, you are going to *Finish the discharge piping assembly* (on page 21).

Finish the discharge piping assembly

In this lesson we are going to finish the assembly.

1. Click the flange at the top of the copied component group, and then select the plus sign on the face end.
2. Right-click and select **component List**.

   The components list displays.
3. Select **Elbows** > **Elbows, 90** > **[6"** (***150***)**] ELL, 90%%D LR S/STD, ASTM A-234 GR WPB**.

   Pick other direction.
4. Select the direction toward the original component group to the east. Alternatively, you can click the end point of the original component to ensure the elbow is pointed in the correct direction.

   Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].
5. Type **CONNECT** on the command line, and then press ENTER.

   Pick point.
6. Select the end point of the flange on the original component group.

   Enter an option [Type/Polyline]<Polyline>.
7. Press ENTER to accept the default.

   Piping and an elbow are added to the drawing.
8. Delete the new elbow and the weld gaps associated with it on both the flange and the pipe.
9. Click the flange at the top of the original component group, and then select the plus sign on the face end.
10. Draw a piece of pipe going up.
11. Click the new pipe, and then select the plus sign in the middle.

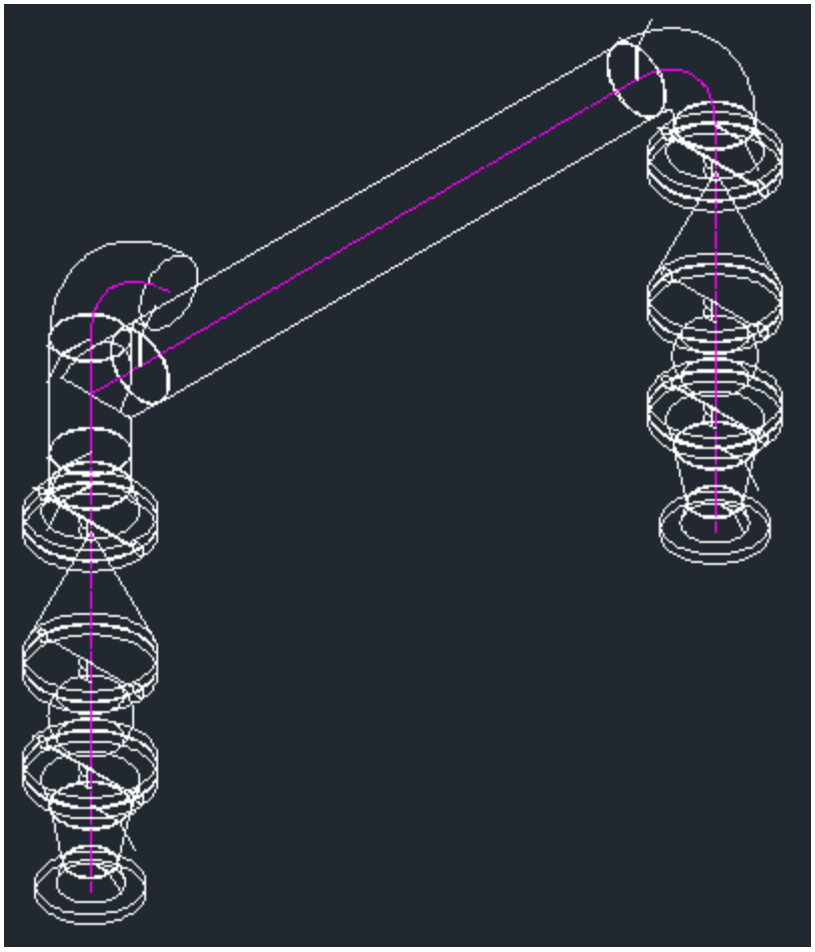
    A tee appears on the piping.
12. Select the piping end point coming from the copied component group.

    Pick branch end direction.
13. Select the piping end point coming from the copied component group.
14. Press ENTER to finish.
15. Click the pipe coming from the copied component group, and then select the plus sign at the end.
16. Connect the pipe to the tee.
17. Delete the extra piping from the tee, including the weld gap.
18. Click the tee, and then select the plus sign at the top.
19. Right-click and select **component List**.

    The components list displays.
20. Select **Elbows** > **Elbows, 90** > **[6"** (***150***)**] ELL, 90%%D LR S/STD, ASTM A-234 GR WPB**.

    Pick other direction.
21. Select a direction to the north, and then press ENTER to finish.

The drawing should look like this.



Now that we are finished, *Create the Assembly* (on page 22) using the **ASSEMBLYVIEW** command.

Create the Assembly

Now that the model is complete, we can add it to the Assembly Viewer.

1. Click **Assembly View** . Alternatively, type **ASSEMBLYVIEW** on the command line, and press ENTER.

   *The **CADWorx Assembly Manager** displays.*

2. Click **Create Assembly** .

   Select components.

3. Pick a window around the components.

   Select components: Specify opposite corner: 39 found, 39 groups.

4. Press ENTER to finish selection.

   Pick insert point.

5. Select the end point of the right or left bottom flange. These points will be used later to connect to a pump.

   Enter Assembly name.

6. Type **Pump Connection** on the command line, and press ENTER.

   *The Pump Connection is saved to the **Assembly Manager**, and displays in the **Assemblies** box.*

If you want to save the drawing to your drawings folder, do the following:

1. Click **Save As** . Alternatively, type **SAVEAS** on the command line, and then press ENTER.

   *The **Save Drawing As** dialog box appears.*

2. Type a name for the drawing, and click **Save**.

Continue to *Placing an assembly using the Assembly Manager* (on page 23).

Placing an assembly using the Assembly Manager

In this lesson we add the discharge piping to the pumps in the Tutor 3 model.

1. Open the *[Product Folder]*\Plant\Tutorial\Tutor3.dwg file delivered with the software. If you already have this open then skip to step 2.

   NOTE To create a drawing based on metric units, open the *[Product Folder]*\Plant\Tutorial\Tutor3m.dwg file.

2. *Set drawing parameters* (on page 9) as defined below.
   - **M** = 4" *(102mm)*
   - **R** = 6" *(152mm)*
   - **Specification** = 150 *(150_M)*
   - **Drawing Mode** = 3D Solids
3. Click the **View** tab, and then click **NE Isometric** in the **Views** panel.
4. Click **Assembly View** . Alternatively, type **ASSEMBLYVIEW** on the command line, and press ENTER.

   *The **CADWorx Assembly Manager** displays.*

5. Select **Pump Connection** from the list of available assemblies, and then click **Insert selected assembly** .

   The assembly appears in the drawing.

6. Connect the assembly with the pumping as shown in the drawing below.

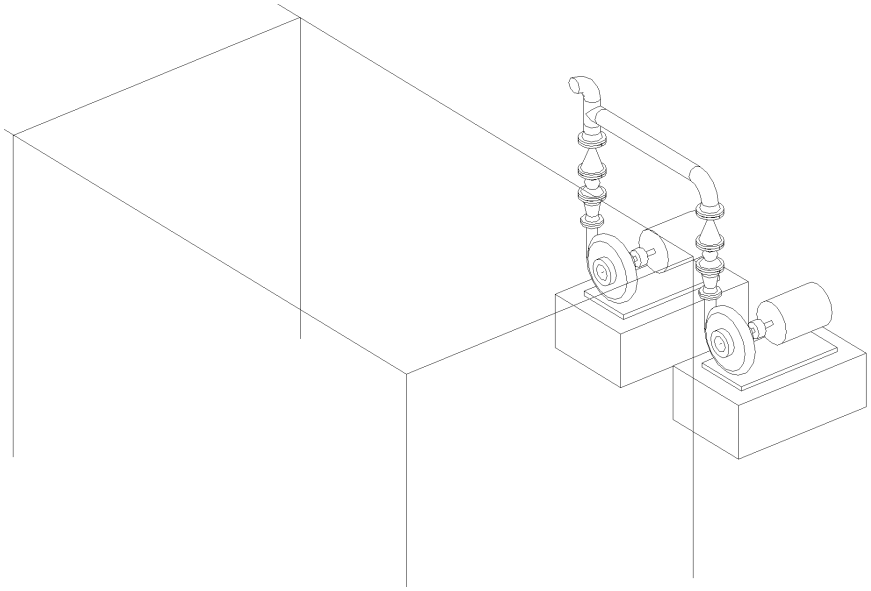
   Pick direction.

7. Select a point in the drawing above the pump.

   Pick other direction.

8. Select a direction south towards the pipe rack.

Your model should look like this.



9. Click **Save As** . Alternatively, type **SAVEAS** on the command line, and then press ENTER.
   *The **Save Drawing As** dialog box appears.*
10. Type a name for the drawing, and click **Save**.

Routing pipe on a pipe rack

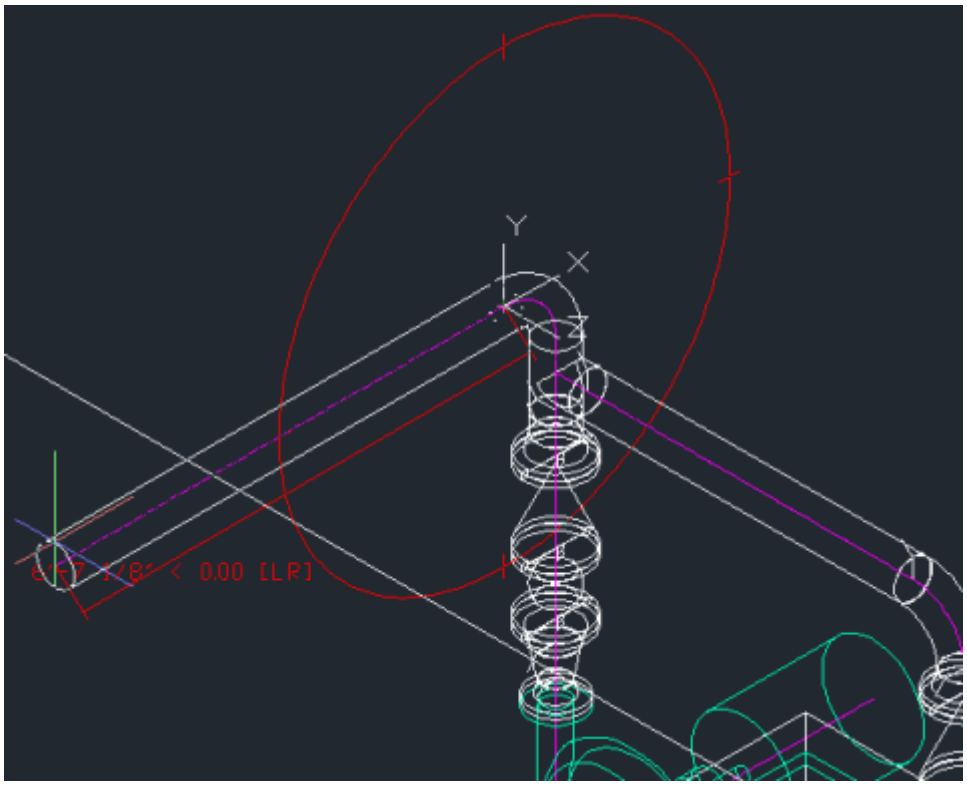
In the drawing that we use for this lesson, the T.O.S. (top of steel) of the pipe rack is at 12'-0" *(3658mm)*. From the last 6" (150) elbow on the discharge piping, more piping is required. Placement of the 6" (150) pipe must be 1'-0" *(305mm)* from the edge of the pipe rack.

1. Do one of the following:
   - Open the completed drawing from *Placing an assembly using the Assembly Manager* (on page 23).
   - Open the *[Product Folder]*\Tutorial\Tutor5.dwg file delivered with the software.

     NOTE To work in metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor5m.dwg file.

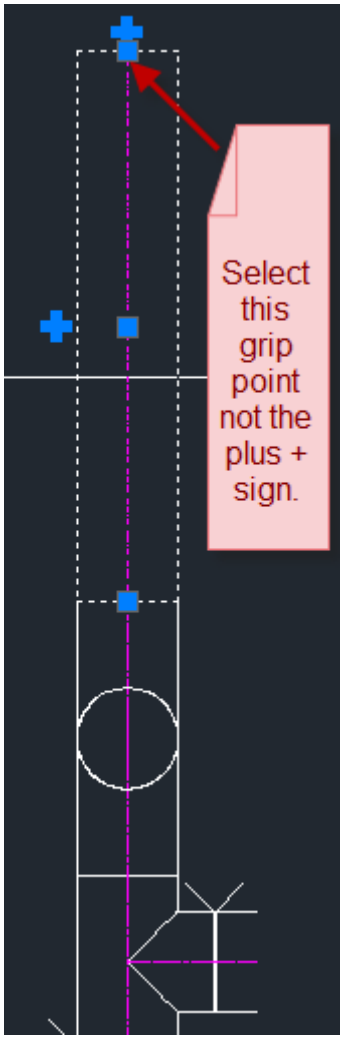
     TIP Keep the drawing in the **SE Isometric** view during these steps.

2. Click the elbow at the top of the pump connection and select the available plus sign.
   Pick end point or [component List /Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect /Toggle length / Alignment].
3. Pull the pipe towards the pipe rack and type **9'2"** (***2794mm***).

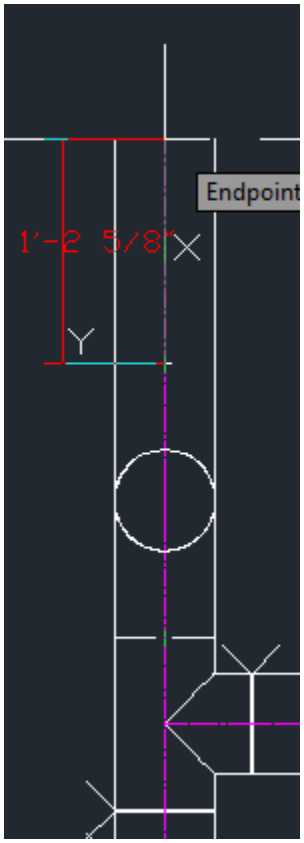


4. Right-click and select **component List**.
   The components list displays.
5. Click **Elbows** > **Elbows, 90** > **[6"** (***150***)**] ELL, 90%%D LR S/STD, ASTM A-234 GR WPB**.
   Pick other direction.
6. Select a point above the components.

7. Model the piece of pipe up from the elbow, and then press ENTER.
8. Click the **View** tab, and then select **Front** from the **Views** panel.
9. Select the pipe that extends up from the pipe rack, and then click the top grip point below the plus sign as shown below.



10. Drag the pipe down to the top of the pipe rack so that it is on the same line.



11. Click the pipe again, and then select the plus sign at the top.

    Pick end point or [component List /Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect /Toggle length / Alignment].

12. Type **A** for alignment on the command line, and then press ENTER.

    Enter an option [Center/BOP/TOP/Left/Right]<Center>.

13. Type **BOP** on the command line, and then press ENTER.

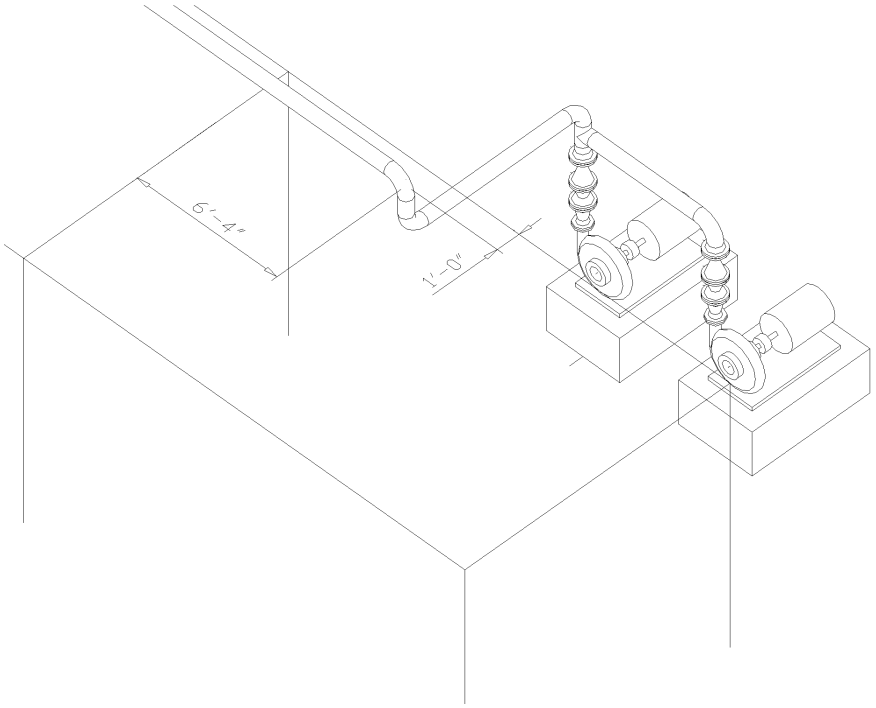
    Pick end point or [component List /Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect /Toggle length / Alignment].

14. Pull the pipe to the left, type **12'** *(3658mm)* on the command line, and then press ENTER.
15. Press ENTER to finish.

    The piping line is established.

16. Click **View > 3D View > Southeast** .

The drawing should look like this.



17. Click **Save As** . Alternatively, type **SAVEAS** on the command line, and then press ENTER.
    *The **Save Drawing As** dialog box appears.*
18. Type a name for the drawing, and click **Save**.

Insert components

Set up the drawing

1. Do one of the following:
   - Open the completed drawing from *Routing pipe on a pipe rack* (on page 24).
   - Open the *[Product Folder]*\Tutorial\Tutor6.dwg file delivered with the software.

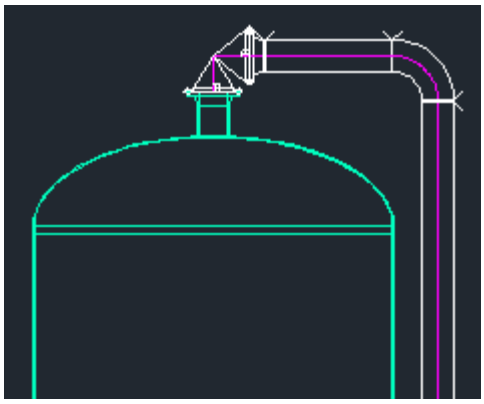
     NOTE If you are working in metric units, open the *[Product Folder]*\Plant\Tutorial\Tutor6m.dwg file.
2. *Set drawing parameters* (on page 9) as listed below:
   - **Main size** - 6" *(150mm)*
   - **Reduction size** - 8" *(200mm)*
   - **Specification** - 150 *(150_MM)*

Follow these steps to place a 6" *(150mm)* flange on the 6" *(150mm)* nozzle. We must use the **center of** object snap to select the face of the nozzle. Check that the **Apply Gasket Insertion Rule** is set to **Automatic** in **Piping Rules**.

1. Click the **View** tab, and then select **Front** from the **Views** panel.
2. Click the **CADWorx Plant I** tab, and then select **Spec View** from the **Palettes** panel.
   *The **CADWorx Spec View** palette displays.*
3. Click **Valves, Relief** > **RELIEF VALVE, 150LB X 150LB FLG [2-24** (***50-600***)**]**.

Pick point or [Length/Pressure/Relief/Corner].

4. Select the center point of the nozzle on top of the vertical vessel.
   Pick relief end direction.
5. Pick a direction to the right towards the pipe rack.
6. Select the previously placed relief valve, and then select the plus sign on the relief end.
7. Right-click and select **component List**.
   The component list displays.
8. Click **Flanges** > **[8"** (***200***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.
   Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].
9. Drag the piping to the right (towards the pipe rack), and then type **2'10.25"** ***(869.95)*** on the command line.
10. Right-click and select **component List**.
    The component list displays.
11. Click **Elbows** > **Elbows, 90** > **8"** (***200***) **ELL, 90%%D LR S/STD, ASTM A-234 GR WPB**.
    Pick other direction.
12. Drag downwards along the vertical vessel, click, and then type **61'-5.875"** ***(18742.03)***.
    The drawing should look like this.



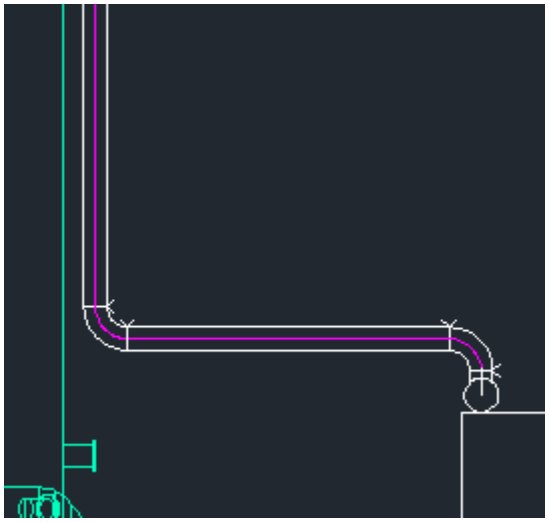
13. Right-click and select **component List**.
    The component list displays.
14. Click **Elbows** > **Elbows, 90** > **8"** (***200***) **ELL, 90%%D LR S/STD, ASTM A-234 GR WPB**.
    Pick other direction.
15. Drag towards the pipe rack away from the vertical vessel, click, and then type **10'5"** ***(3175)***.
16. Right-click and select **component List**.
    The component list displays.
17. Click **Elbows** > **Elbows, 90** > **8"** (***200***) **ELL, 90%%D LR S/STD, ASTM A-234 GR WPB**.
    Pick other direction.
18. Select a direction downward towards the pipe rack, and then click.
19. Change the view to **SE Isometric** .
20. Select the previously placed elbow, and then select the plus sign on the bottom.
21. Right-click and select **component List**.
    The component list displays.

22. Click **Tees** > **Tee Reducing Buttweld** > **[12"x8"(*300-200*)] TEE, REDUCING S/STD, ASTM A-234 GR WPB**.

    Pick direction or [Change connection].
23. Type **CHANGE** on the command line, and then press ENTER.

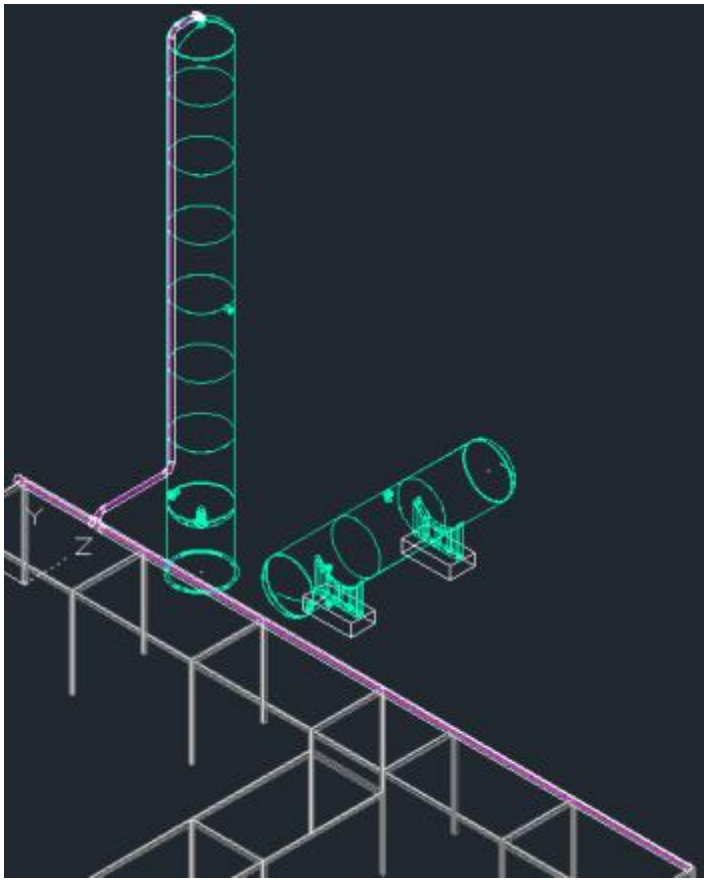
    Pick direction or [Change connection].
24. Select a direction to the right, parallel with the pipe rack.

    The drawing should look like this.

    Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].
25. Type **ALIGNMENT** on the command line, and then press ENTER.
26. Type **BOP** on the command line, and then press ENTER.
27. Drag the pipe to the end of the pipe rack on the right side, and then press ENTER to finish.
28. Click the tee again, and then select the plus sign on the left.
29. Type **ALIGNMENT** on the command line, and then press ENTER.
30. Type **BOP** on the command line, and then press ENTER.
31. Drag the piping to the left end of the pipe rack, and then press ENTER to finish.

The drawing should look like this.



Continue to *Connect the piping from the horizontal vessel to the pumps* (on page 31).

Connect the piping from the horizontal vessel to the pumps

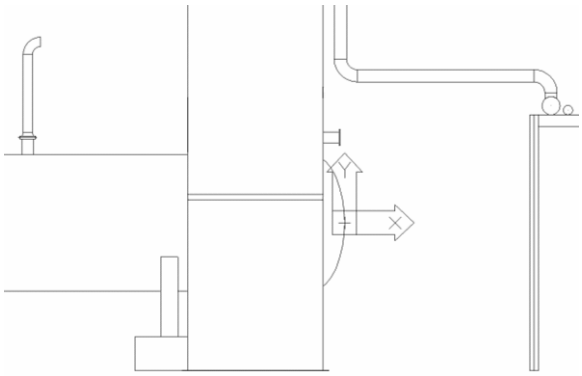
Next, the horizontal vessel requires piping all the way to the pumps that we created earlier.

1. *Set drawing parameters* (on page 9) as listed below:
   - **Main size** - 6" *(150mm)*
   - **Reduction size** - 8" *(200mm)*
   - **Specification** - 150 *(150_MM)*

Next, attach a 6" *(150mm)* pipe run to the nozzle on top of the horizontal vessel. First, we must place a flange on the nozzle and then the gasket and bolts are automatically added based on our piping rules. Draw a piece of pipe from the flange, and then another elbow, as shown in the figure below. After that, follow the steps to stretch the piping to the same BOP as the 8" *(200mm)* pipe.

1. Click the **View** tab, and then select **Front** from the **Views** panel.
2. Click the **CADWorx Plant I** tab, and then select **Spec View** from the **Palettes** panel.
   *The **CADWorx Spec View** palette displays.*
3. Click **Flanges** > **[6"** (***150***)**] FLG, RFWN 150LB S/STD BORE, ASTM A-105**.
   Pick start point or [Buttweld end] <last point>.
4. Click the center point of the nozzle on the top of the horizontal vessel.
   The flange is placed.

5. Click the newly placed flange, and then select the plus sign at the top.
   Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].
6. Pick a direction upward, and type **2'-5.8125"** (***757.24***).
   Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode/ Undo / Connect / Toggle length / Alignment].
7. Right-click and select **component List**.
   The component list displays.
8. Click **Elbows** > **Elbows, 90** > **[6"** (***150***)**] ELL, 90%%D LR S/STD, ASTM A-234 GR WPB.**
   Pick other direction.
9. Pick a direction toward the rack.
   The drawing should appear similar to the example below.

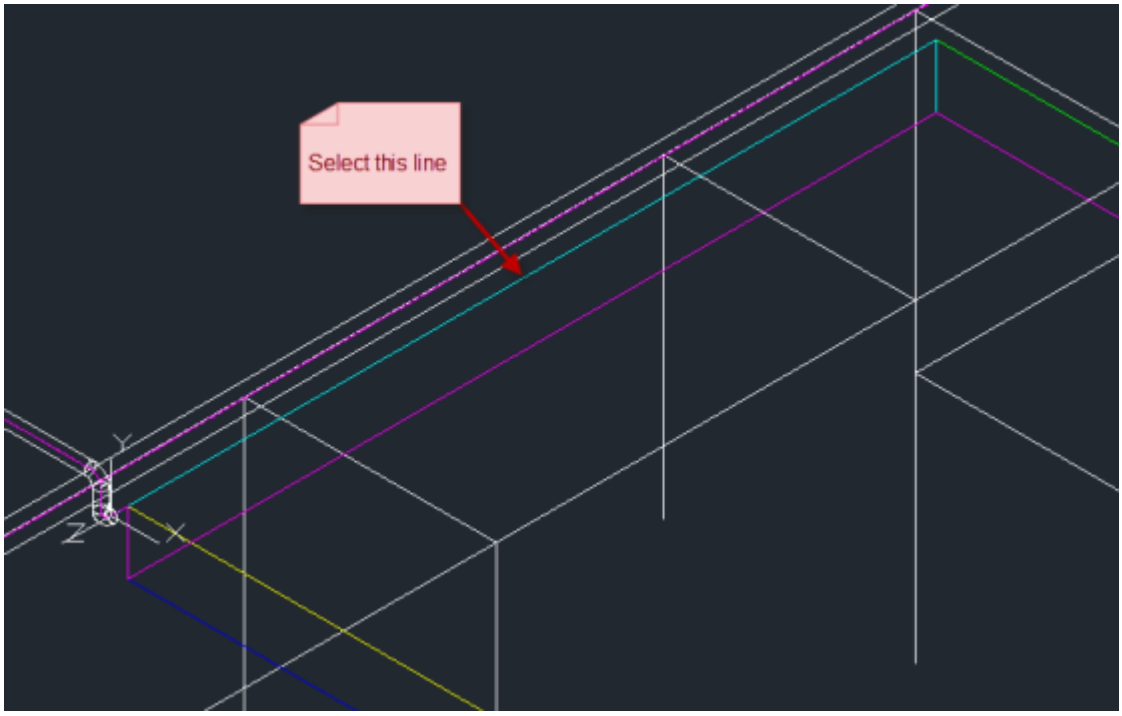


Next, finish the connection to the 6" *(150mm)* pipe on the rack.

1. Pick a direction to the right, and type **30'-7"** (***9321.8***).
2. Right-click and select **component List**.
   The component list displays.
3. Click **Elbows** > **Elbows, 90** > **[6"** (***150***)**] ELL, 90%%D SR S/STD, ASTM A-234 GR WPB.**
   Pick other direction.
4. Pick a direction downward.
5. Drag the pipe downward, and type **11.0625"** (***280.99***)**.**
6. Click the **View** tab, and then click **SE Isometric** or **NE Isometric** in the **Views** panel**.** Choose the view that best works for you.
7. Click the newly placed pipe, and then select the plus sign at the bottom.
8. Right-click and select **component List**.
   The component list displays.
9. Click **Elbows** > **Elbows, 90** > **[6"** (***150***)**] ELL, 90%%D SR S/STD, ASTM A-234 GR WPB.**
   Pick other direction.
10. Select a direction down the pipe rack, parallel with the 12" (300) piping.
    Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode / Undo / Connect / Toggle length / Alignment].
11. Type **ALIGNMENT** on the command line, and press ENTER.
    Enter an option [Center/BOP/TOP/Left/Right]<Center>.
12. Type **BOP** on the command line, and press ENTER.

Pick end point or [component List / Slope / Elevation / Plane / Reference / Fitting mode/ Undo / Connect / Toggle length / Alignment].

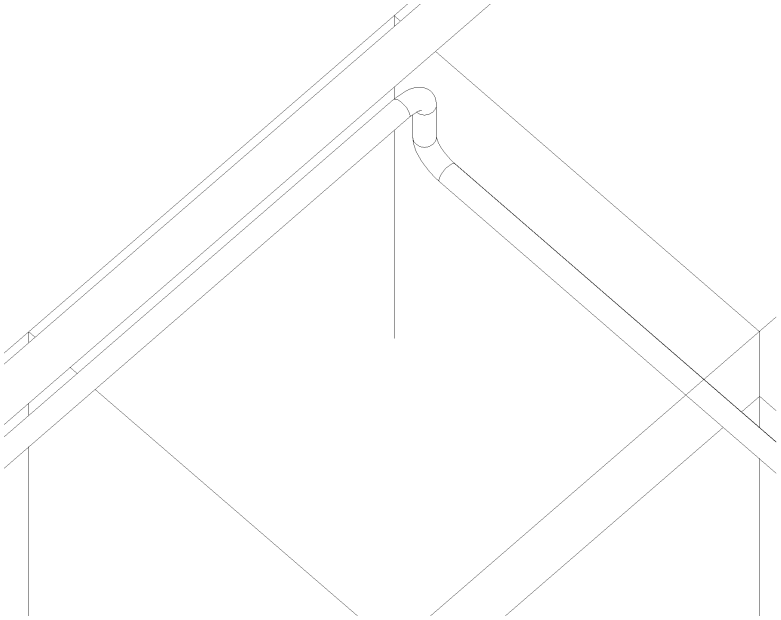
13. Type **CONNECT** on the command line, and press ENTER.
    Pick point.
14. Zoom out and select the point at the end of the 6" (150) piping coming out of the pumps.
    Do you want to shorten the length of the connecting pipe?:[Yes/No]<Yes>.
15. Type **N** on the command line, and press enter.
    Enter an option [Type/Polyline]<Polyline>.
16. Press ENTER to accept the default.
    Select the line to be routed or <Enter for finish>.
17. Select the line as displayed in the image below.



*The **Selection** box displays.*

18. Click the **3D Polyline** with the color box that is the same color as the line we selected above.
    Select the line to be routed or <Enter for finish>.
19. Press ENTER to finish.

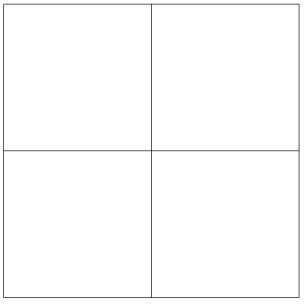
The final result should appear similar to the example below.



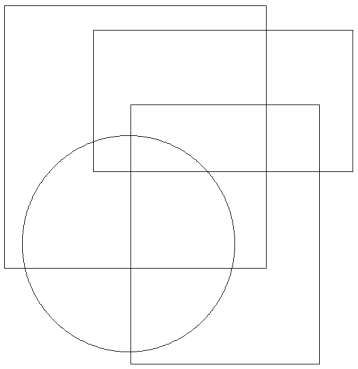
20. Click **Save As** . Alternatively, type **SAVEAS** on the command line, and then press ENTER.
    *The **Save Drawing As** dialog box appears.*
21. Type a name for the drawing, and click **Save**.

Paper space and CADWorx

Model space requires that **TILEMODE** be on, set it to 1. When working in model space, non-floating view ports are available. These non-floating view ports are created with the AutoCAD **VPORTS** command. The only way to arrange these view ports is beside each other, similar to a tiled floor. Model space is where all CADWorx orthographics, models, isometrics, and so on are created.



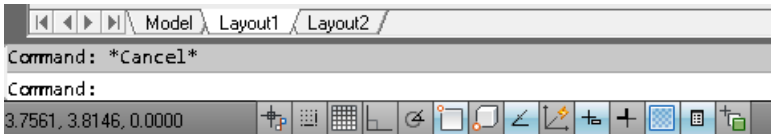
Paper space requires that TILEMODE be set to 0, which effectively turns off model space. When working in paper space, floating view ports are available. These floating view ports are created with the AutoCAD **MVIEW** command. Multiple view ports can also be created. Unlike non-floating view ports, the floating view ports created with the **VPORTS** command are able to overlap each other. Paper space is used only for annotation and borders. Although it is an entity with a layer, size, line type, and so forth, it is helpful to think of the Mview as a hole cut through the paper looking into the model space.



The **MSPACE** command displays as crosshairs while in the view port, and as a pointer when outside of the view port. While in this view port, the UCS icon appears as the traditional icon (broken pencil if out of plane).

NOTES

- You can also use the **Layout** tab at the bottom of the AutoCAD window to enter paper space. The **Model** tab allows you to enter model space.



- To create multiple layouts, right-click the **Layout** tab, and then select **New layout**. For example, you can create a separate layout for Plan, Elevation, or Section.
- **MSPACE** and **PSPACE** commands can be used only in paper space. **MSPACE** (model space) is used when entering a view port created with the **MVIEW** command. **PSPACE** (paper space) is used when exiting a view port created with the **MVIEW** command.

1. Open the *[Product Folder]*\Tutorial\Tutor7.dwg file.

   NOTE To create a drawing based on metric units, open the *[Product Folder]*\Plant\Tutorial\Tutor7m.dwg file.
2. Click **Layout1.**
3. Type **MVIEW** on the command line to create view ports with which to view the model.

   Specify corner of viewport or

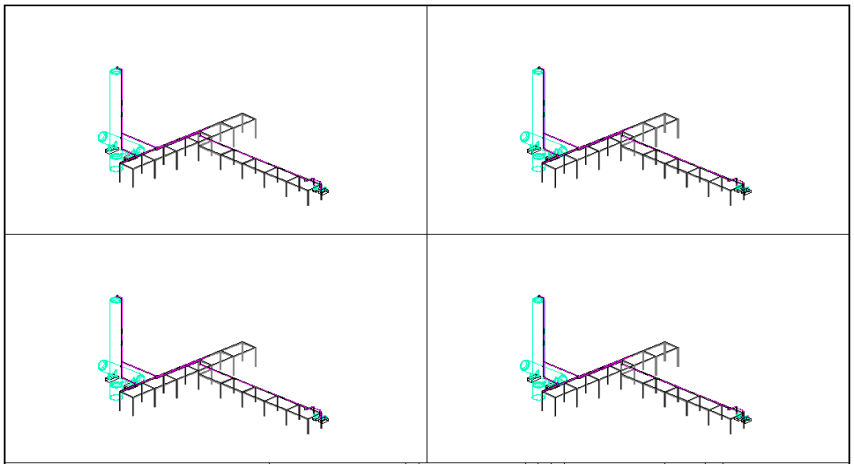
   [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/LAyer/2/3/4] <Fit>.
4. Type **4** on the command line, and then press ENTER.

   Specify first corner or [Fit] <Fit>.
5. Select the lower left corner of the inside border area.

   Specify opposite corner.
6. Select the upper right corner of the inside border area.

   Regenerating model.

Four Mviews are created. These Mviews are used to create a plan and elevations of the pumps. The upper right Mview is used for an isometric view of the pumps. The upper left is used for a plan. The lower left is a front elevation, and the lower right is a side elevation.



7. Double click the upper right corner viewport.
8. Type **DDVPOINT** on the command line, and then press ENTER.
   *The **Viewpoint Presets** dialog box appears.*
9. Type **315** in the **X Axis** box, type **30** in the **XY Plane** box, and then click **OK**.
10. Click the **View** tab, and then click **Window** on the **Navigate 2D** panel.
    Specify first corner.
11. Pick a point to the left and above the pumps.
    Specify opposite corner.
12. Pick a point to the right and below the pumps.
    The view zooms in around the pumps.
13. Click inside the upper left view port to activate it.
14. Click **View > 3D Views > Plan View > Current UCS**. Alternatively, type **PLAN** on the command line, and then select the **Current UCS** option.
15. Click **Plant > Utility > Zoom Factors > Architect > 3/8 = 1'** (metric at 1:20).
    Center point.
16. Pick a point between the two pumps.
    TIP If necessary, click **Pan** to reposition the view.
17. Click inside the lower left view port to activate it.
18. Type **DDVPOINT** on the command line, and then press ENTER.
    *The **Viewpoint Presets** dialog box appears.*
19. Type **270** in the **X Axis** box, type **0** in the **XY Plane** box, and then click **OK**. This provides a front elevation of the pumps.
20. Click **Plant > Utility > Zoom Factors > Architect > 3/8 = 1'-0"** (metric at 1:20).
    Center point.
21. Pick a point between the two pumps.
22. Click inside the last view port.
23. Type **DDVPOINT** on the command line, and then press ENTER.
    *The **Viewpoint Presets** dialog box appears.*

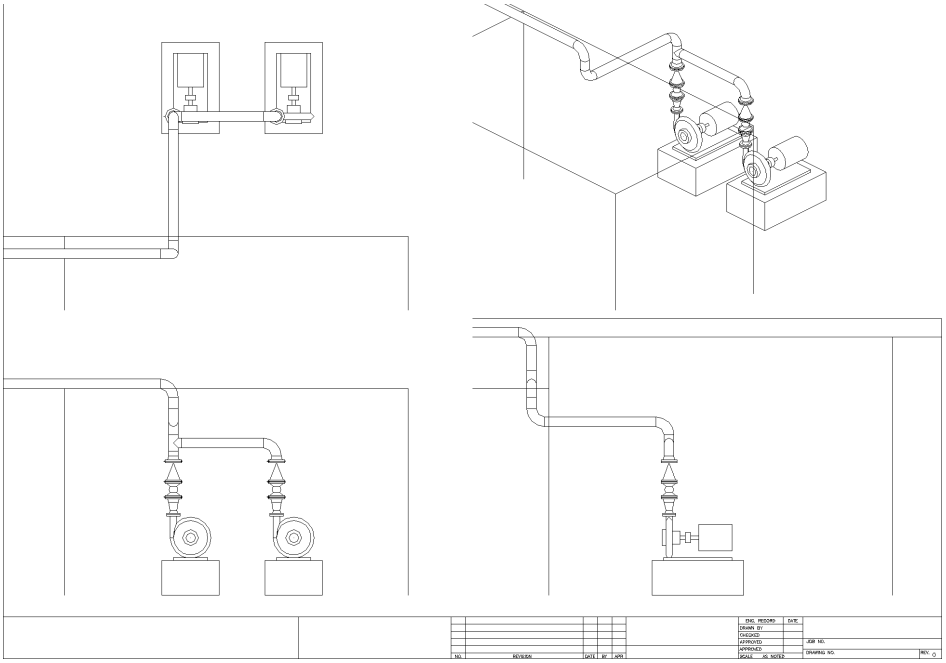
24. Type **0** in the **X Axis** box, type **0** in the **XY Plane** box, and then click **OK**.
    The view port showing the side of the pumps is adjusted.
25. Click **Plant > Utility > Zoom Factors > Architect > 3/8 = 1'-0"** (metric at 1:20).
    Center point.
26. Click a point between the pipe rack and the pumps.

NOTE You can perform dimensioning in the view port itself or within paper space. In the view port itself, set the AutoCAD setvar DIMSCALE to 0. CADWorx, and AutoCAD automatically size all the text, arrow heads, and so on. The setvar DIMASSOC handles all the scaling and sizing of the dimensions and also controls the way the dimensions are updated when objects are changed in the model space environment. For more information about this setvar, see the AutoCAD documentation.

Align views within the view ports

1. Click **Plant > Utility > MVSetup**. Alternatively, type **MVSETUP** on the command line.
   Enter an option [Align/Create/Scale viewports/Options/Title block/Undo].
2. Type **A** on the command line, and then press ENTER to select the **Align** option.
   Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo].
3. Type **H** on the command line, and then press ENTER to select the **Horizontal** alignment option.
   Specify basepoint.
4. Type **INT** on the command line, and then press ENTER.
5. Select the column on the pipe rack in the lower right view port.
   Specify point in viewport to be panned.
6. Type **INT** on the command line, and then press ENTER.
7. Select the column of the pipe rack in the lower left view port.
   Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo].
8. Type **V** on the command line, and then press ENTER to select the **Vertical** alignment option.
   Specify basepoint.
9. Type **INT** on the command line, and then press ENTER.
10. Select the upper pipe rack in the upper left view port.
    Specify point in viewport to be panned.
11. Type **INT** on the command line, and then press ENTER.
12. Select the upper pipe rack in the lower left view port.
    Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo].
13. Press ENTER to finish.
    Enter an option [Align/Create/Scale viewports/Options/Title block/Undo].
14. Press ENTER.

The drawing should resemble the example below.



NOTES

- You can use the **Create** option of the **MVSETUP** command to automatically generate all the view ports and view adjustments for each one. This lesson demonstrates only the manual method.
- The most efficient way to create profiles is with the AutoCAD **SOLPROF** command. Using solids and the **SOLPROF** command allows you to create silhouettes of these items in view ports. Solids can also be plotted with hidden line removal. For example, the figure above shows the four completed views with hidden line removal. To plot this drawing, freeze the ViewL layer and set the **Paper** button active on the status bar (or type **PSPACE** on the command line, and then press ENTER).

Create a manual isometric drawing

1. Open the *[Product Folder]*\Tutorial\Tutor2.dwg file.

   NOTE If you are working in metric units, open the *[Product Folder]*\Tutorial\Tutor2m.dwg file.

   The status bar indicates that the drawing is in model space.
2. Click the **Layout** tab to place the drawing in paper space.

   *This environment shows the border. All that is now required is to activate a view port with the **MVIEW** command. Set the ViewL layer current prior to doing the steps below.*
3. Type **MVIEW** on the command line, and then press ENTER.

   Specify corner of viewport or

   [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/LAyer/2/3/4] <Fit>.
4. Type **INT** on the command line, and then press ENTER.
5. Select the lower left of the available border.

   Specify opposite corner.
6. Type **MID** on the command line, and then press ENTER.
7. Select the upper available border.

Regenerating model.

This cuts a hole through the paper showing the model in the background.

8. If the model space is not active click **Paper** on the status bar. Alternatively, double-click on an area over the view port.

   *The **Paper** button changes to **Model**.*

9. Type **DDVPOINT** on the command line, and then press ENTER.

   *The **Viewpoint Presets** dialog box appears.*

10. Type one of the following values in the **From X** box: **45**, **135**, **225**, or **315**.
11. Type **35.3** in the **XY Plane** box, and then click **OK**.

    This places the model in line with a paper space isometric rotation.

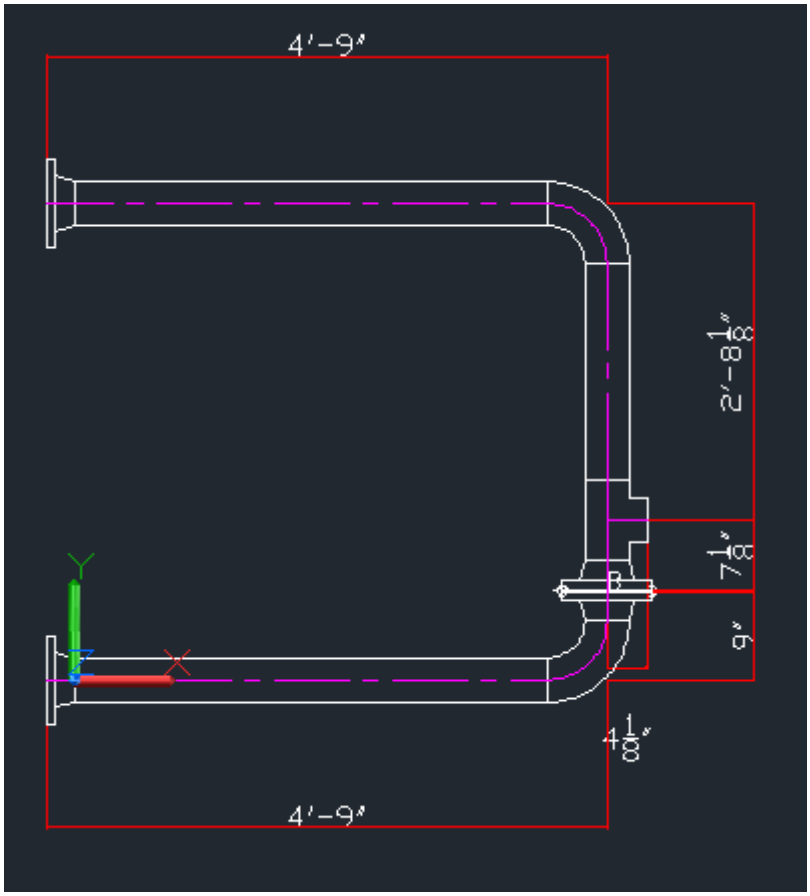
    NOTE Alternatively, you can use any of the **View > 3D Views** commands to produce the same result.

12. Use the **ZOOM,** and **PAN** commands to better position the model within the view port.
13. After you have finished positioning the view, click **Zoom Lock** on the **Settings** toolbar. Alternatively, type **ZOOMLOCK** on the command line, and then press ENTER.

    NOTE With the AutoCAD setvar **DIMSCALE** set to 0, you can easily apply dimensions. The only requirement is to position the Point and Shoot UCS to the plane that requires dimensioning. Click **Object** on the **UCS** toolbar, and then select an existing dimension to align the UCS.

14. Increase the setvar **DIMTXT** variable to 3/16" (4.7625mm) to allow the dimensions to look as though they are 1/8" (3.175mm) due to being out of the plane (30-degrees out of view).

15. Place a few dimensions, as shown below.



NOTE Placing the top dimensions requires moving the UCS. For example, click **Flat** on the **UCS** toolbar, and then pick a corner at one of the upper elbows.

16. Switch back to **Model Space**.
17. Click **Plant > Utility > Drawing Control > Elbow Centerline** to turn on the option to draw centerlines, and provide corners on the elbows to which to dimension.
18. Click **Plant > Accessory > Mode Convert > Isometric** to convert the model to isometric. Alternatively, type **CONVERTISO** on the command line, and then press ENTER.

    Select objects.
19. Pick a point in the upper left corner of the model.

    Specify opposite corner.
20. Pick a point in the lower right corner of the model.

    Select objects.

21. Press ENTER to finish the selection.

NOTES

- For dimensions that seem upside down, use the following options to place the dimensions correctly. Click **Flat** on the **UCS** toolbar, and then type **Z** on the command line to select the **Ztext** option.
- To dimension the vertical leg of the model, click **North** on the **UCS** toolbar, and then pick a point at the lower elbow.

CAESAR II and CADWorx data transfer

The CADWorx Plant import and export functions provide a seamless bi-directional link to CAESAR II and its pipe stress analysis capabilities. By selecting components in the model environment of the drawing, a CAESAR II input file can be created. This input file can be opened in CAESAR II and the model geometry modified. After the stress analysis has been performed, the input file can be transferred back into the CADWorx Plant drawing environment. During import, the software updates or adds the components that were modified in CAESAR II with a skeleton geometry that is clearly visible. Components removed in CAESAR II are deleted. Unchanged components are not modified. All CAESAR II information is then stored in the drawing providing future analysis capabilities based on the previous information (node numbers, temperature, pressure, and so forth.).

In this lesson, you use the import and export features of CAESAR II and CADWorx Plant to create a fully annotated stress isometric drawing. The major tasks in this lesson are:

- *Create a CAESAR II input file* (on page 41)
- *Modify the input file* (on page 43)
- *Import the CAESAR II input file* (on page 45)
- *Re-run the job* (on page 48)

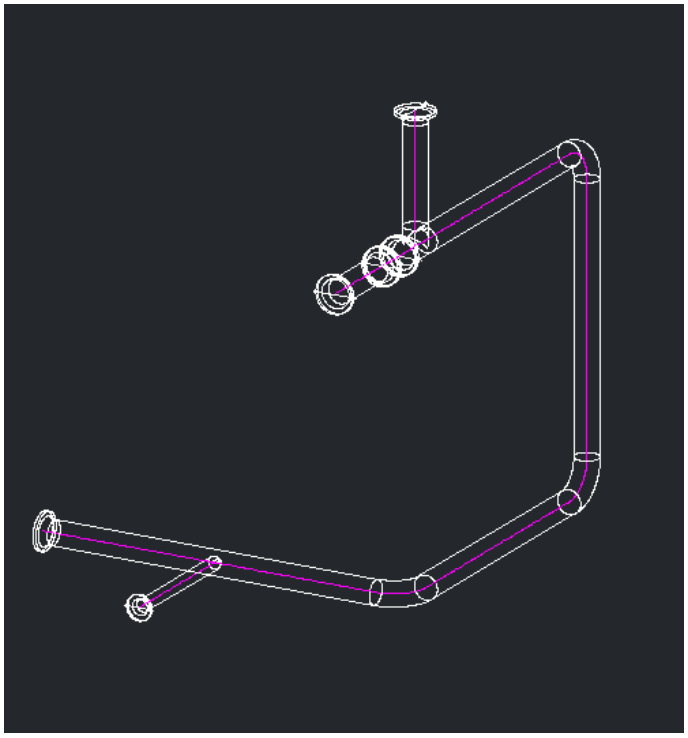
NOTE You must have CAESAR II installed on your computer to complete the tasks in this lesson.

Create a CAESAR II input file

1. Open the *[Product Folder]*\Tutorial\Tutor11.dwg file.

   NOTE To work in metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor11m.dwg file.

The drawing file that opens represents a typical system built in CADWorx Plant that is to be sent to CAESAR II.



2. Click **Plant > Accessory > CAESAR II > System Out**. Alternatively, type **C2OUT** on the command line and then press ENTER.

   TIP You can also click the **CADWorx Plant I** tab and then click **System Out** on the **Misc** panel.

   *The **CAESAR II Input File** dialog box displays.*
3. Click **Save** to save the CAESAR II input file with the default name Tutor11.

   Enter an option [Database/Line number/Select components] <Select components>.
4. Press ENTER to accept the default **Select components** option.

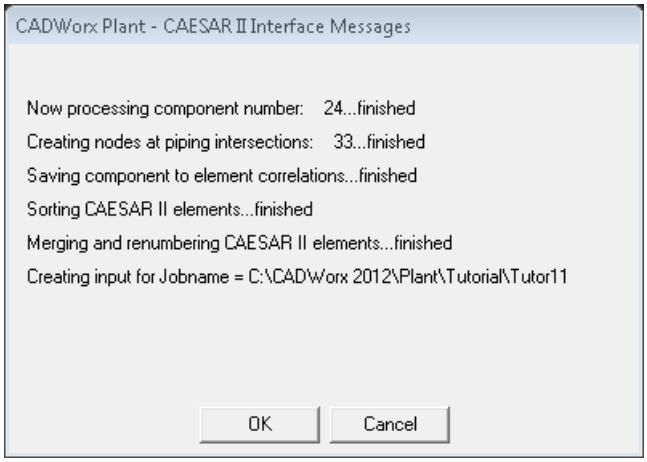
   Select objects.
5. Type **ALL** on the command line, and press ENTER.

   Select objects.
6. Press ENTER to finish selection.

   Provide start locations [Yes/No] <No>.

   NOTE This prompt allows you to control noding within CAESAR II. For more information, see "System Out" in the *CADWorx Plant User's Guide*.
7. Press ENTER to accept the default **No** option.

When processing completes, the software displays a message box similar to the example below.



8. Click **OK** to close the message box and return to the drawing.

   The piping layout has been successfully transferred to a CAESAR II input file named Tutor11._A.

Modify the input file

1. Start CAESAR II and open the *[Product Folder]*\Plant\Tutorial\Tutor11.c2 input file you created in *Create a CAESAR II input file* (on page 41).
2. Click **Input > Piping**.

   *The **Classic Input Piping** dialog box displays.*
3. Click **Options > Node Numbers**.

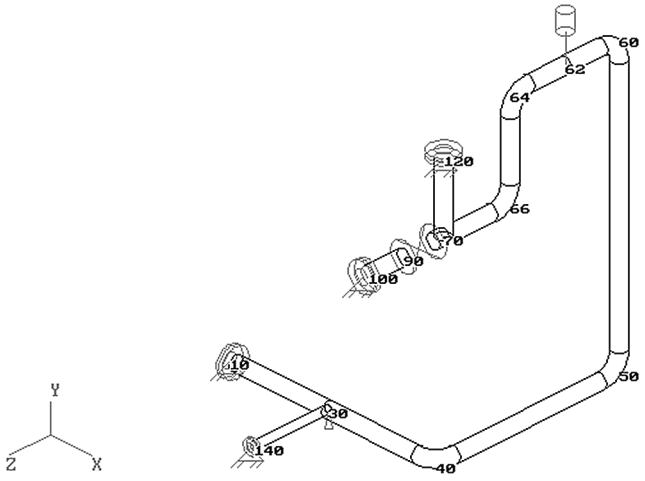
   The graphic representation of the model updates to display the same piping configuration as was in CADWorx Plant.
4. Click **First Element** and then double-click **Restraints**.
5. Type **10** in the **Node** box, and select **ANC** in the **Type** list.
6. Click **Next Element** and then double-click **Restraints**.
7. Type **30** in the **Node** box, and select **+Y** in the **Type** box.

   A 5'-0" (1524mm) expansion loop is added to the line.
8. Click **Next Element** to move to element 50-60.
9. Type **15'-0"** *(4572mm)* in the **DY** box.
10. Click **Next Element** to move to element 60-70, and then enter a single node to place a new node for the hanger.
11. Type **62** in the **To** box.
12. Type **3'-0"** *(914mm)* in the **DY** box as the distance from node 60.

    The only entries required are the new node number and the distance.
13. Double-click **Hanger**, and type **62** in the **Node** box to define a location for the new hanger.
14. Click **Next Element** to move to element 62-70.
15. Click **Model > Break**.

    *The **Break at element 62 - 70** dialog box displays.*

16. Type **64** in the **New Node Number** box, type **3'-0"** *(914mm)* in the **Distance** in box, and click **OK**.
17. Double-click **Bend** to place a bend at the end of this location.
    A long radius bend is added to the piping configuration.
18. Click **Edit > Insert** to insert another element after 62-64.
    The Insert Element dialog box appears.
19. Verify that **After** is selected, and click **OK**.
20. Type **66** in the **To** box.
21. Type **-5'-0"** *(-1524mm)* in the **DY** box.
22. Double-click **Bend** to place a bend at this location.
23. Click **Next Element** to move to element 64-70.
24. Type **66** in the **From** box.
    This connects the remaining system to the new node 66 and ties together the loop.
25. Click **Next Element** to move to element 100-110.
26. Double-click **Restraints**.
27. Type **110** in the **Node** box, and select **ANC** in the **Type** list.
28. Repeat steps 25 through 26 to place anchors on nodes 130 and 150.
29. Click **First Element**.
30. Type **250 F** *(175 C)* in the **Temp 1** box.
31. Type **300 PSI** *(1000 Kpa)* in the **Pressure 1** box.
32. Double-click **Allowable Stress**.
33. Select **B31.1** in the **Code** list, and type **20,000 PSI** *(120,000 Kpa)* in the **SC** and **SHI** boxes.
    The CAESAR II job should be similar to the example below.



NOTE Use the **Restraints**, **Anchors**, and **Hangers** commands on the **Options** menu to toggle on and off the display of the restraints, anchors and hangers.

34. Click **Start Run** to save the input and start the error checking procedure.
    The error checker software reviews the CAESAR II model and alerts you to any possible errors, inconsistencies, or noteworthy items. These items display in a grid as errors, warnings,

or notes. The total numbers of errors, warnings, or notes display in corresponding boxes above the message grid.

NOTE The error checking procedure should generate 0 errors, 0 warnings, and 2 notes. Any other result indicates an input error and should be resolved before moving on to *Import the CAESAR II input file* (on page 45).

35. Click **File > Exit** to close the **Classic Piping Input** dialog box.

Import the CAESAR II input file

Next, import the modified input file into CADWorx Plant to update the piping drawing. Some minor modifications will be required that should then be sent back to CAESAR II one more time.

1. Click **Plant > Accessory > CAESAR II > System In**. Alternatively, type **C2IN** on the command line, and then press ENTER.

   TIP We can also click the **CADWorx Plant I** tab, and then click **System In** on the **Misc** panel.

   *The **CAESAR II Input File** dialog box displays.*

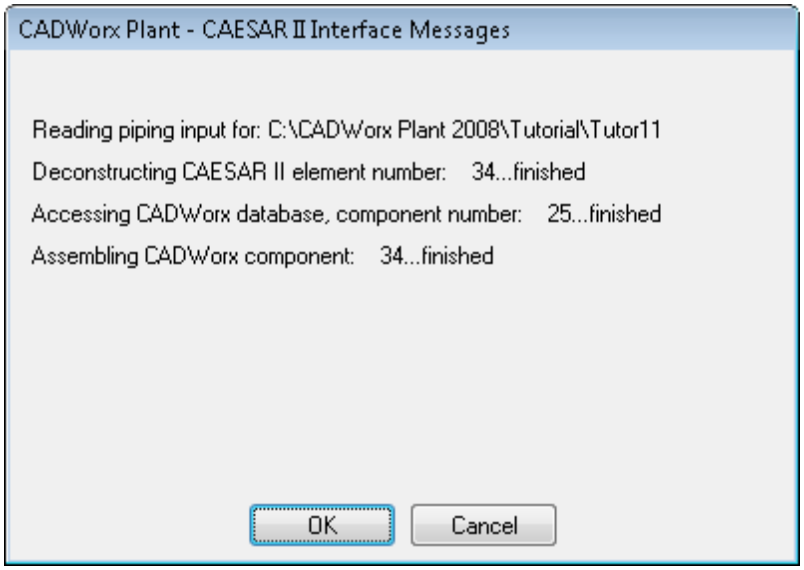
2. Navigate to the CAESAR II input file, and click **Open**.

   Please set a default Specification <150>.

3. Press ENTER to accept the default specification.

   Prepared E:\CADWORX PIPE 2002\SPEC\150.spc specification...

   NOTE The specification is not used here but is always requested by the system as a backup default.



   After reading the input file, the next prompt requests the level of intervention required. For more information, see "System In" in the CADWorx Plant User's Guide*.*

   Enter an option [Full intervention/Partial/None] <None>:

   Working component number: 34.

   NOTE The areas that have been modified are readily apparent.

4. Click **Plant > Accessory > Mode Convert > 3D Solids** to convert the entire model to 3D solids. Alternatively, type **CONVERTSOLID** on the command line, and then press ENTER.

TIP We can also click the **CADWorx Plant I** tab, and then click **3D Solids** on the **Setup Size/Spec** panel.

3D solids conversion...

Select objects.

5. Type **ALL** on the command line, and press ENTER.

   Select objects.

6. Press ENTER to finish selection process.
7. Click **North** on the **UCS** toolbar to relocate the UCS. Alternatively, type **VIEWNORTH** on the command line and then press ENTER.

   TIP We can also click the **CADWorx Plant II** tab and then click **North** on the **UCS** panel.

   [Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>.

8. Type **ENDP** on the command line, and press ENTER.
9. Select the lower end point of the vertical 8" *(200mm)* pipe to the far right.
10. Click **Plan Tee** on the **Buttweld** toolbar. Alternatively, type **TESW** on the command line and then press ENTER.

    8" (200) TEE, STR. S/STD, ASTM A-234 GR WPB

    Pick start point or [Branch/Center] <last point>.

11. Type **ENDP** on the command line, and press ENTER.
12. Select the bottom buttweld of the upper right elbow.

    Pick main end direction.

13. Type **@0,-100** on the command line and then press ENTER, or pick a direction in the drawing.

    Pick branch end direction.

14. Type **@100,0** on the command line and then press ENTER, or pick a direction in the drawing.
15. Click **Pipe** on the **Buttweld** toolbar. Alternatively, type **PIPW** on the command line, and then press ENTER.

    8" (200) PIPE, S/STD SMLS, ASTM A-106 GR B

    Pick start point or [TOP/BOP] <last point>.

16. Type **ENDP** on the command line, and press ENTER.
17. Select the branch connection of the previous drawn tee.

    Pick end point.

18. Type **@36,0** *(@914,0)* on the command line, and press ENTER.
19. Click **Weld Neck** on the **Flanges** toolbar to place the weld neck flange. Alternatively, type **FLGW** on the command line, and then press ENTER.

    8" (200) FLG, RFWN 150LB S/STD BORE, ASTM A-105

    Pick start point or [Face end] <last point>.

20. Press ENTER to accept the last point.

    Pick direction.

21. Type **@100,0** on the command line and then press ENTER, or pick a direction in the drawing.

    8" (200) GASKET, 1/8" THK, 150LB

    Automatically placed...

22. Click **Spring Hanger** on the **Restraints** toolbar. Alternatively, type **SPRING** on the command line and then press ENTER.

TIP We can also click the **CADWorx Plant II** tab, and then click **Spring Hanger** on the **Restraints** panel.

Pick location on component for restraint.

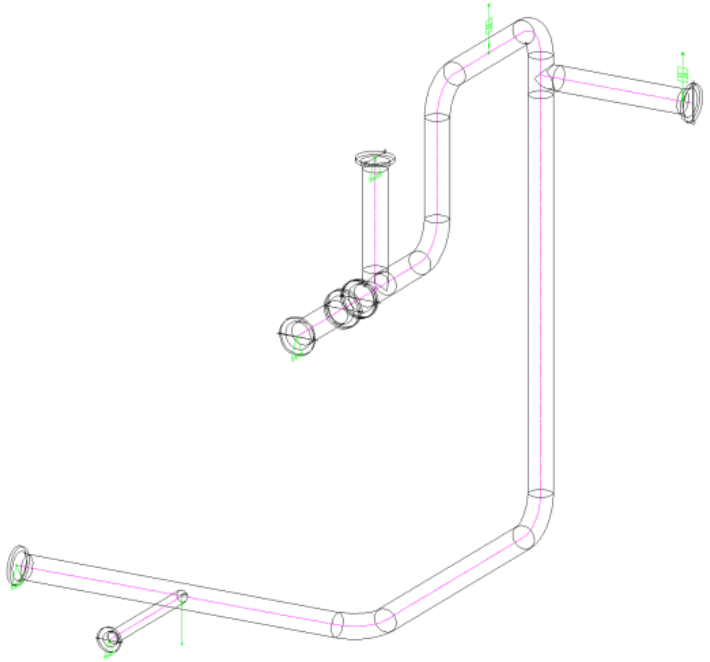
23. Type **ENDP** on the command line, and press ENTER.
24. Select the right end of the new horizontal pipe.

    Enter or pick rotation.
25. Type **@0,100** on the command line and then press ENTER, or pick an upward direction in the drawing.

    Enter depth.
26. Type **17** (***432***) on the command line, and press ENTER.

    Enter length or diameter <1'-5" (432)>.
27. Press ENTER to accept the default.

    Enter assembly identification number <17.00-3 SPRING HANGER>.
28. Press ENTER to accept the default.



29. Click **Plant > Accessory > CAESAR II > System Out**. Alternatively, type **C2OUT** on the command line, and then press ENTER.

    TIP We can also click the **CADWorx Plant I** tab, and then click **System Out** on the **Misc** panel.

    *The **CAESAR II Input File** dialog box appears.*
30. Click **Save** to accept the default input file name (Tutor11.c2)

    Enter an option [Database/Line number/Select components] <Select components>.
31. Press ENTER to accept the default **Select components** option.

    Select objects.
32. Type **ALL** on the command line, and press ENTER.

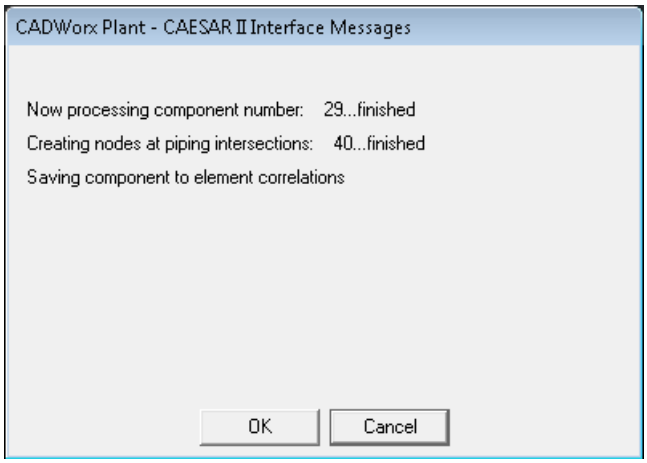
    Select objects.

33. Press ENTER to *finish the selection.*

    Provide start locations [Yes/No] <No>.

34. Press ENTER to accept the default **No** option.

    The software begins transferring the piping layout to the CAESAR II input file. When processing completes, the software displays a message box similar to the example shown below.



Re-run the job

1. Start CAESAR II, and open the *[Product Folder]*\Plant\Tutorial\Tutor11.c2 input file.
2. Click **Input > Piping**.

   *The **Classic Piping Input** dialog box appears.*

   NOTE If necessary, use the **Restraints**, **Anchors**, and **Hangers** commands on the **Options** menu to toggle on and off the display of the restraints, anchors, and hangers.
3. Click **Batch Run** to check the input data, analyze the system, and present the results when processing is complete.
4. View the results.
5. When we are finished viewing the results, click **File > Exit** to close the **Classic Piping Input** dialog box and return to the CAESAR II main window.
6. Click **File > Exit** to close CAESAR II.

Automatic Dimensioning

We can create dimensions automatically for a set of components that we select. Options to set the offset distance and whether to include flanges are also available.

NOTE When using automatic dimensioning, we must set the AutoCAD setvars as specified below:

- Set **DIMSCALE** to **0** so that the dimension text and arrows automatically size themselves. For more information, see *Paper space and CADWorx* (on page 34).
- Set **DIMASZ** to **1/8"** ***(3****mm)*. This value represents a typical plotted size. Within this environment, the dimensions should scale correctly.
- Set **DIMTXT** to **2"** (***50***mm)

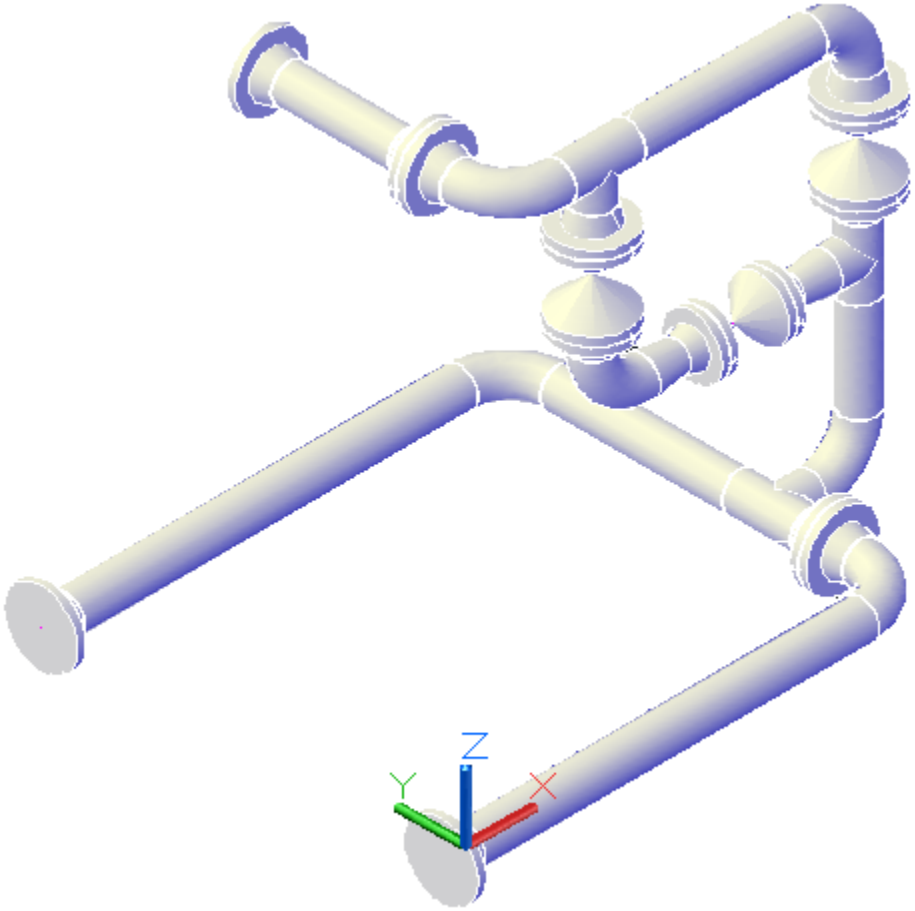
1. Open the *[Product Folder]*\Tutorial\Tutor2.dwg file.

NOTE To work in metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor2m.dwg file.

Or

2. Open the drawing we completed in *Route Modeling* (on page 10).

   The software opens the drawing, which resembles the example below.



3. Click the **CADWorx Plant I** tab, and then click **2D Double Line** on the **Setup Size/Spec Panel.**

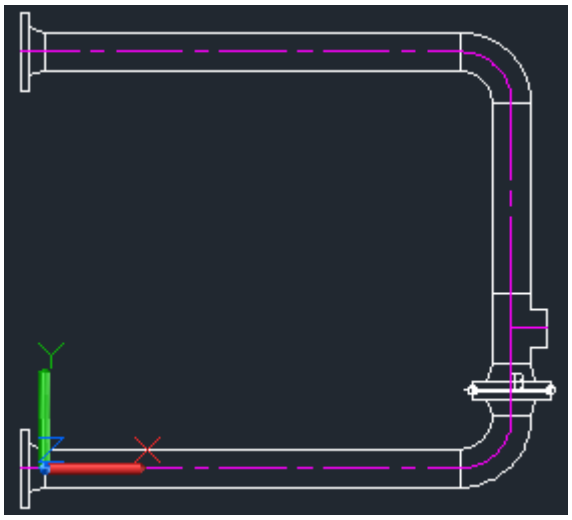
   Select components.

4. Select the entire drawing, and then press ENTER to finish.

   The entire drawing is converted to 2D double line.

5. Select everything above the u shape and extending from the bottom tee, and then delete it.
6. Click the **View** tab, and then click **Top** on the **Views** panel.

The drawing should resemble the one below.



7. Click **Plant > Dimension > Automatic**. Alternatively, type **AUTODIM** on the command line, and then press ENTER.

   TIP We can also click the **CADWorx Plant II** tab, and then click **Automatic** on the **Dimensions** panel.

   Enter an option [Flange/Offset/Selection].

8. Press ENTER for the default or select **Selection** in the popup menu to select objects to dimension.

   NOTE The **Flange** option allows dimensioning to flange faces. If this option is not selected, dimensioning is only to the corners and tees of the piping system. The **Offset** option allows you to adjust the distance from the component to the dimension line.

   Select objects:

9. Type **ALL** on the command line, and then press ENTER.

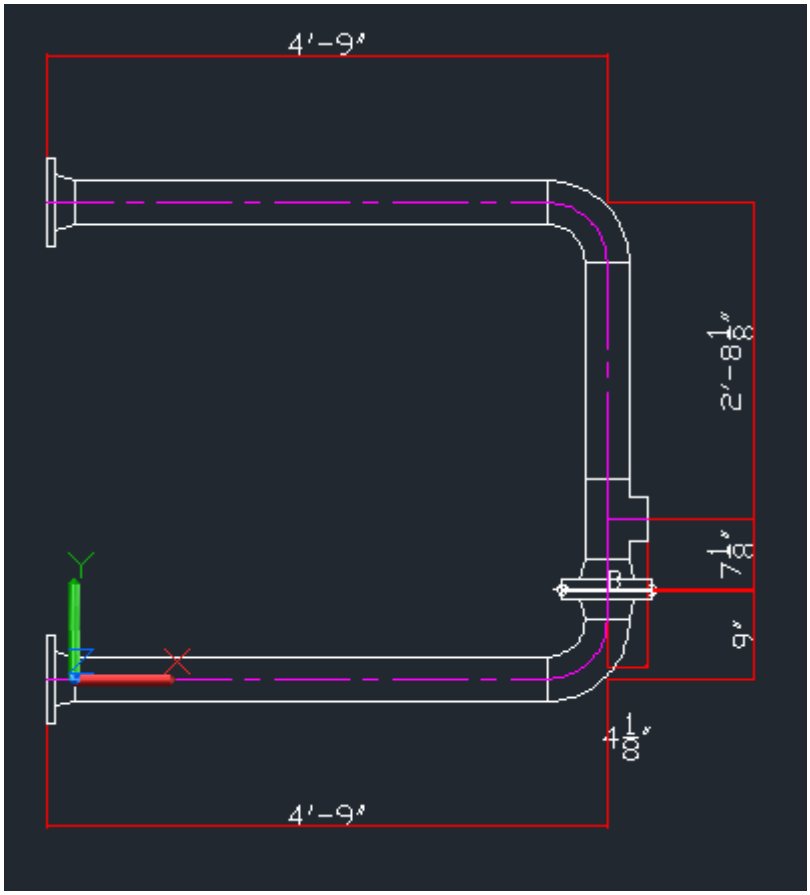
   Select objects:

10. Press ENTER to finish the select set.

    Pick centroid point of dimensions:

11. Select the open space in the middle of the U shape.

Auto Dimensioning...finished.



NOTES

- Automatic dimensioning requires that the system have at least one turn in it. If the message **Inadequate selection...aborting *Cancel*** appears, then the system is lacking an elbow or tee.
- When modifying dimensions that already exist in the drawing, the **UCS Object** command can be used to set the UCS and the dimension in the same plane. This makes it easy to stretch, grip, and correct overlapping dimensions.
- We can re-scale dimensions only if the setvar **DIMASO** is on. If the view port is re-sized by zooming, the dimensions can be updated to the correct scale.
  a. Type **DIM** on the command line, and then press ENTER.
     Dim.
  b. Type **DIMTXT** on the command line, and then press ENTER.
     Enter new value for dimension variable <3/16">.
  c. Type **0.125** *(3)* on the command line, and then press ENTER.
     Dim.
  d. Type **UPDATE** on the command line, and then press ENTER.
     Select objects.
  e. Type **ALL** on the command line, and then press ENTER.
     Select objects.

f. Press ENTER to finish the selection set

Dim.

g. Type **EXIT** on the command line, and then press ENTER.

Bill of Material (BOM)

In this lesson, we automatically place a Bill of Material schedule and all the associated tags on the drawing in paper space. For more specific Bill of Material command reference information, see *Pipe BOM/DB* in the *CADWorx Plant User's Guide*.

1. Open the *[Product Folder]*\Plant\Tutorial\Tutor10.dwg file.

   NOTE For metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor10m.dwg file

2. Click **Plant > Bill of Material > Run > Cut**. Alternatively, type **BOMCUT** on the command line, and then press ENTER.

   TIP We can also click the **CADWorx Plant I** tab, and then click **Cut** on the **Pipe BOM/DB** panel.

   Enter tag option [Counter/Manual placement/Automatic placement] <Automatic>.

3. Press ENTER to accept automatic placement.

   Select upper right corner of BOM heading.

4. Pick a point at the corner of the border.

   Enter an option [Line number/Select components] <Select components>.

5. Press ENTER to accept the default **Select components** option.

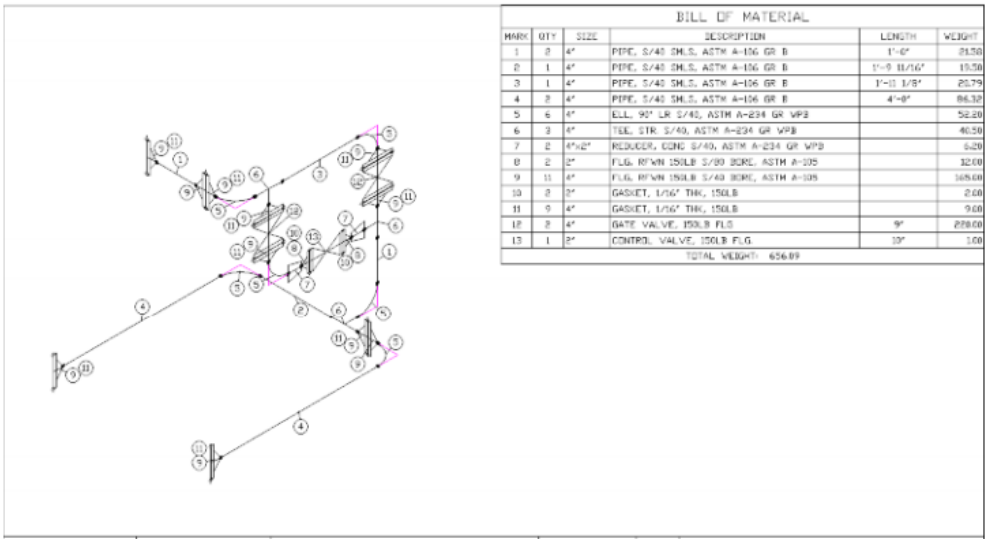
   Select objects.

6. Select all the objects in the view port.

   Select objects.

7. Press ENTER to finish.

   A Bill of Material, similar to the example below, is placed on the isometric drawing.



NOTES

- Use the **Plant > Bill of Material > Setup** command to customize the Bill of Material schedule to any style or layout that you need. Alternatively, click the **CADWorx Plant I** tab, and then click **Setup** on the **Pipe BOM/DB** panel, or type **BOMSETUP** on the command line, and then press ENTER. Using the options on the **Bill of Material Setup** dialog box, you can specify column widths, provide the schedule direction (up or down), provide user specified headings for columns, allow alignment of columns, and the selection of over 30 different entries into the schedule. You can arrange the columns in any order. For more information, see *Bill of Material Setup* in the *CADWorx Plant User's Guide*.
- You can use the system variables **BOMScheduleDirection**, **BOMScheduleSpacing**, **BOMTagGridFactor**, **BOMTagOptimize**, **BOMTagRadiusFactor**, and **BOMTagSpacingFactor** to control the Bill of Material characteristics. For more information, see *Startup Variables* in the CADWorx Plant User's Guide.
- Use the AutoCAD commands listed below to assist with the placement or relocation of material tags.
  - Click **Plant > Bill of Material > Tag > Location**, or type **TAGRELOCATE** on the command line, and then press ENTER, to relocate material tags manually. The leader to the component will also be adjusted with this function.
  - Click **Plant > Bill of Material > Tag > Insert**, or type **TAGINSERT** on the command line, and then press ENTER, to place a single graphic mark symbol (it will not be consistent with the bill of material).
  - Click **Plant > Bill of Material > Tag > Toggle**, or click **TAGTOGGLE** on the command line, and then press ENTER, to toggle on/off tags Bill of Material tags, similar to the toggle for automatic and stress isometrics.
- The Bill of Material schedule is developed and grouped together. To add lines within the Bill of Material schedule, select one of the existing lines, and copy it to the new location. Use the AutoCAD **DDEDIT** command to modify the new line item.
- The **Plant > Bill of Material > Export** command extracts the components in the drawing to a spreadsheet. Alternatively, click the **CADWorx Plant I** tab, and then click **Export** on the **Pipe BOM/DB** panel, or type **BOMEXPORT** on the command line, and then press ENTER.
- If the modeled piping view is panned slightly, the component marks do not show as correct. Use the AutoCAD **MOVE** command to adjust them, and then use the AutoCAD **PSPACE** command to return to paper space. Move the material marks by window, and then select a base point at a leader's endpoint. Next, select a second point of displacement, object snap, midpoint of, or the centerline of a component. Object snap mode selection is allowed from paper space to model space. This corrects the alignment of the material marks.

Edit components

1. Click **File > Open**, and then open the *[Product Folder]*\Tutorial\Tutor2.dwg file.

   NOTE To work in metric units of measure, open the *[Product Folder]*\Plant\Tutorial\Tutor2m.dwg file.

2. Click **Component Edit** on the **Settings** toolbar. Alternatively, type **CEDIT** on the command line, and then press ENTER.

   TIP You can also click the **CADWorx Plant I** tab, and then click **Local Edit** on the **Setup Size/Spec** panel.

   Select objects.

3. Select a section of pipe, and then press ENTER.

*The **Component Edit** dialog box for the selected component type appears.*

4. Make changes as needed, and then click **OK** to save the modifications.

NOTES

- **Alpha size** is used only to define the alpha description. It does not allow the physical size of the component to be changed. There is no automatic mechanism that changes all components from one size to another.
- **Weight** allows you to change the weight per unit of pipe. For other components, the value in the **Weight** box indicates the total weight.
- **Sort sequence** is used for the Bill of Material. If the location of a component in the Bill of Material must be last, type **999** in the **Sort sequence** box.
- **Remove** allows the data inside of the component to be removed. You can use this option for marking existing piping within a new plan or piping drawing. The information is removed permanently.
- The options in the **Coordinates** section allow the length of the pipe to be altered. This does not change the physical length on the drawing, only the length that appears in the Bill of Material. If you select the **Iso** option, the **Length** box becomes available, allowing you to enter a new value. Stretching the component does not change the length recorded inside of it. If the **Iso** option is not used, and the component was stretched, the length updates automatically.
- The **Existing** option in the **Miscellaneous** section places the component in a state where it does not appear in the Bill of Materials, but it retains all of its information. This option also converts the component to the **Exist** layer (installation layer name). The **CAESAR II** option indicates whether the component has stress analysis information attached, and allows it to be removed if this box is un-checked.

Specification Editor

This tutorial demonstrates how to use the Specification Editor to create a new catalog and a new project specification. It also demonstrates how to change a delivered specification for use with a customer. The first tutorial demonstrates how to create a catalog from beginning to end. The next tutorial does the same for a project specification. Also included is a demonstration on how to update an existing project specification or catalog for a client. Lastly, we demonstrate how to use optional components, which can be toggled on or off. The Specification Editor is meant to allow companies to create specialized data for each of their clients. It is also meant to allow for component sizes that are not delivered with the specification to be added to that specification.

NOTE If you have a specification presently set in CADWorx when you open the Spec Editor, it is automatically loaded. If it is not presently set, use **Setup** on the **Setup Size/Spec** panel to set it. For more information, see *Specification and Size* in the *CADWorx Plant User's Guide*.

Create a New Catalog

In the following lesson we create a catalog from scratch for Cantor Piping. They have specific sizes that they do not want allowed in their piping ranges.

1. Open the **Spec Editor** , and click **File** > **New Catalog** .
   *The **New Catalog** dialog box displays.*
2. Type **Cantor Piping 625** in the **Name** box.
3. Select **Company Catalog** from the **Type** list.
4. Select the **Base Catalog** ellipsis, then select **Sample_Inch.cat** *(Sample_Metric.cat)***,** and click **Open**.
5. Type **Catalog for Cantor Piping products** in the Description box.
6. Check that the **Location** box is linked to the **Spec** folder.
7. If working in Metric, change the **Units** to **Metric/Metric**. If not, then leave them at **English/Inch**.
8. Click **OK**.

Now *Add a size table* (on page 55) to the new catalog.

Add a size table

1. Click the **Home** tab, and then click **Size Table** on the **Add Table** panel. Alternatively, right-click **Size Tables** in **Catalog Data,** and then select **Add Size Table** .
   *The **New Size Table** dialog box displays.*
2. Type **Size** in the **Name** box**.**
3. Type **Main sizes for components** in the **Description** box.
4. Accept the **Default Part Number**.
   NOTE The Current Units box will be grayed out and should contain the units currently associated with the open catalog.
5. Click **OK**.
   *The new size table displays in the main screen with the description we typed in at the top left. It is also added to **Catalog Data** under **Size Tables**.*

We now need to add the new sizes for all the components that will later be added to this catalog.

1. Right-click in the main window, and then click **Add Standard Size**.
   *The **Select size from template** dialog box displays.*
2. Click the **Select / deselect all** check-box to add all sizes, and then click **OK**.
   The sizes selected are displayed in the main window.

Now we will add a new size to the table.

1. Right-click in the main window, and then click **Add New Size**.
   An empty line is displayed at the bottom of the list.
2. Type **16.5000** (***425***) under **NOM**.
3. Type **17.0000** (***450***) under **OD**.
4. Type **"16 1/2"** (***"425mm"***) under **DESCRIPTION**.

5. Type **0044** under **PTN**.
6. Save the catalog by clicking **File** > **Save**.

Now *Add a material table* (on page 56) to the catalog.

Add a material table

1. Click the **Home** tab, and then click **Material Table**  on the **Add Table** panel. Alternatively, right-click **Material Tables** in **Catalog Data,** and then select **Add Material Table**  .

   *The **New Material Table** dialog box displays.*
2. Type **MAT** in the **Name** box.
3. Type **Materials table for Cantor** in the **Description** box.
4. Click **OK**.

   *The new material table displays in the main screen with the description you typed in at the top left. It is also added to **Catalog Data** under **Material Tables**.*

We now insert a row for the material used by Cantor Piping.

1. Right-click in the main window, and then click **Insert Row**.

   An empty line is displayed.
2. Type **A312** under **SPEC**.
3. Type **TPXM19** under **GRADE**.
4. Type **22Cr-13Ni-5Mn** under **COMPOSITION**.
5. Type **0.2830** under **DENSITY**.
6. Type **Caps,Elbows,Tees,Crosses,Valves,Flanges,Gaskets/Bolts/Welds** under **COMPTYPES.**
7. Type **0201** under **PTN**.
8. Save the catalog by clicking **File** > **Save**.

Now *Add a schedule table* (on page 56) to the catalog.

Add a schedule table

1. Click the **Home** tab, and then click **Schedule Table**  on the **Add Table** panel. Alternatively, right-click **Schedule Tables** in **Catalog Data,** and then select **Add Schedule Table**  .

   *The **New Schedule Table** dialog box displays.*
2. Type **SCH** in the **Name** box.
3. Accept the **Default Part Number**.
4. Type **Schedule for Cantor** in the **Description** box.
5. Click **OK**.

   *The new schedule table displays in the main screen with the description you typed in at the top left. It is also added to **Catalog Data** under **Schedule Tables**.*

Now add the schedule

1. Right-click in the main window, and then click **Add Schedule.**
   *The **Add Schedule Items** dialog box displays.*
2. Hold CTRL, and select **5S, 5, 10S, 40S, 100**, **120,** and **140** from the **Standard Items** list, and click the **Add arrow** to add them to the **Selected Items** list.
3. Click **New**  , type **130**, and then click **OK**.
   The schedule items are added to the list.
4. In the PTN box, type **001, 002, 003, 004, 005, 006, 007,** and **008** respectively for each schedule.
5. Save the catalog by clicking **File** > **Save**.

Next *Add a thickness table* (on page 57) to the catalog.

Add a thickness table

1. Click the **Home** tab, and then click **Thickness**  on the **Add Table** panel. Alternatively, right-click **Thickness Tables** in **Catalog Data,** and then select **Add Thickness Table**  .
   *The **Thickness Table** dialog box displays.*
2. Type **THICK** in the **Name** box.
3. Select **Size** from the **Size Table** list. Most likely this is the default.
4. Select **SCH** from the **Schedule Table** list. Most likely this is the default.
5. Type **Thickness Table for Cantor** in the **Description** box.
   *The new thickness table displays in the main screen with the description we typed in at the top left. It is also added to **Catalog Data** under **Thickness Tables**.*

NOTE The new table brings in all the Nominal sizes from the size table, and the schedule numbers from the schedule table.

Now add thickness amounts to 100, 120, and 140.

1. Click the block at the intersection of **100** and **24.0000** (***600***), and then type **1.5310** (***38.887***).
2. Click the block at the intersection of **120** and **10.0000** (***250***), and then type **0.5000** (***21.412***).
3. Click the block at the intersection of **140** and **8.0000** (***200***), and then type **0.8120** (***20.625***).

Now *Add an end type table* (on page 57) to the catalog. The EndType table we are going to add is for a Flange.

Add an end type table

1. Click the **Home** tab, and then click **EndType Table**  on the **Add Table** panel. Alternatively, right-click **EndType Tables** in **Catalog Data,** and then select **Add EndType Table**  .
   *The **New EndType Table** dialog box displays.*
2. Type **BW_11EN_EXA** in the **Name** box.
3. Select **Buttwel**d from the **Type** list.
4. Select **Size** from the **Size Table** list.
5. Click the **Size Table** ellipsis.
   *The **Select Size** dialog box displays.*

6. Select **2.0000, 4.0000, 6.0000** (***50***, ***100***, ***150***) from the list, and then click **OK**.
7. Type **0013** in the **Part Number** box.
8. Type **EndType Buttweld table for Cantor** in the **Description** box.
9. Click **OK**.

   The new end type table displays in the main screen with the description we typed in at the top left. It is also added to ***Catalog Data*** *under* ***End Type Tables****.*

The columns are empty so now we must input the rest of the information.

1. In the **2.0000** (***50***) row, type **1.5000** (***38.1000***) under **OAL** and **FACELEN**, and then type **6.0000** (***152.4000***) under **CFD** and **FFD**.
2. Highlight the row from **OAL** to **FFD**, right-click, and then select **Copy**.
3. Highlight the **4.0000** (***100***) row from **OAL** to **FFD**, right-click, and then select **Paste**.
4. Repeat step 3 for the **6.0000** (***150***) row.

Now *Add a category* (on page 58) to the Data Tables file. We must add a category to the Data Tables file before we can add new data tables.

Add a category

1. Click the **Home** tab, and then click **Category Table**  in the **Add Table** panel.

   Alternatively, right-click **Data Tables** in **Catalog Data,** and select **Add Category**  .

   The ***Add Category*** *dialog box displays.*
2. Click the **Category** list, and then select **Elbows**.
3. Type **Cantor Elbows** in the **Description** box.
4. Click **OK**.

   The new category is added to ***Data Tables*** *in* ***Catalog Data****.*

Now that we have added the category we can *Add a data table* (on page 58) to that category.

Add a data table

1. Click the **(+)** sign on the **Data Tables** file in **Catalog Data**.

   The ***Data Tables*** *file expands displaying the categories.*
2. Select **Elbows**.
3. Click the **Home** tab, and then click **Data Table**  on the **Add Tables** panel. Alternatively, right-click a category in **Data Tables** in **Catalog Data,** and then select **Add Data Table**  .

   The ***New Data Table*** *dialog box displays.*
4. Type **90L_W_S_STD** in the **Name** box.
5. Click the **Type** list, and then select **Short Radius Elbow**.
6. Click the **Size Table** list, and then select **Size.**
7. Click the **Size Table** ellipsis.

   The ***Select Size*** *dialog box displays.*
8. Select **2.0000, 4.0000, 6.0000** (***50***, ***100***, ***150***) from the list, and then click **OK**.

9. Click **Apply same end-type on all ends**. This option grays out all but the **Start** end type option.
10. Click the **Start** list, and select **BW_11EN_EXA**. This is the end type we created earlier.

    NOTE If this were a cross or a tee we would also be able to enter the **Left**, and **Right** end types, unless we selected **Apply same end-type on all ends**.

    CAUTION Make sure that the selected EndType tables contain data in accordance with the new component size range.
11. Type **0012** in the **Part Number** box.
12. Type **Cantor's Buttweld Elbows** in the **Description** box.
13. Click **OK**.

    *The new data table appears in the main screen with the name in the upper left corner, and in the **Catalog Data** under the category we selected.*

The columns are empty so now we must input the rest of the information.

1. In the **2.0000** (***50***) row, type **3.0000** (***80***) under **RAD**, and **90.00** under **ANG.**
2. Type **1.5000** under **WEIGHT**, and **3** under **ETS**.
3. Highlight the row from **RAD** to **ETS**, right-click, and then select **Copy**.
4. Highlight the **4.0000** (***100***) row from **RAD** to **ETS,** right-click, and then select **Paste**.
5. Repeat step 3, and 4 for the **6.0000** (***150***) row.

The New Catalog is finished. Feel free to practice this process.

Edit an existing catalog

In the following lesson, we add information to an existing catalog for a company called DJ Piping. We can take existing catalogs and save them separately for different companies that require different components.

1. Click **File** > **Open**  .
2. Click **Sample_Inch.cat,** and click **Open**.
3. Click **File** > **Save As**  .

   CAUTION If you do not save changes to a different file, then you are making changes to a delivered product. If you save these changes directly to **Sample_Inch.cat** you might have to re-install CADWorx to return your setup to default. We highly recommended that you save this using another name before continuing.
4. Type **Inch Example Tutorial**, and click **Save**. You can choose to save this outside of the Spec Folder in a folder on your computer.

If you want to practice adding new tables to an existing catalog use the steps in *Create a New Catalog* (on page 55). The following tutorial demonstrates how to edit a Size table, a Schedule table, and a Data table. We also edit the properties for some of the options.

1. Select **Size** under **Size Tables** in **Catalog Explorer**.

   *The table displays to the right of the **Catalog Explorer.***
2. Hold CTRL, and select the **2.5000, 5.0000, 16.0000,** and **38.0000** (***65***, ***125***, ***400***, ***1000***) rows.
3. Right-click on a row, and then click **Remove Row**..

   *The **Remove** dialog box displays.*
4. Click **OK**.

   *The **2.5000, 5.0000, 16.0000,** and **38.0000** (**65**, **125**, **400**, **1000**) rows are removed.*

5. Right-click **Size,** and select **Properties**. Alternatively, click the **View** tab, and then select **Properties** in the **Display** panel.

   *The **Properties** window opens.*
6. Click the word **Size** next to **Table Name**, and then delete it.
7. Type **DJ Piping Sizes Allowed**, and, then click anywhere in the screen.

   *The **Size Table** name changes.*
8. Type your name in the **Edited By** box.
9. Type **DJ Piping Requested Sizes** in the **Description** box.

Now we are going to edit a Schedule table.

1. Click **SCH** under **Schedule Tables** in **Catalog Explorer**.

   *The table displays to the right of the **Catalog Explorer.***
2. Hold CTRL, and select the **5S, 5, XXS,** and **STD** rows.
3. Right-click on a row, and then click **Remove Row**.

   *The **Remove** dialog box displays.*
4. Click **OK**.

   *The **5S, 5, XXS,** and **STD** rows are removed.*
5. Right-click **SCH,** and select **Properties**. Alternatively, click the **View** tab, and then select **Properties** in the **Display** panel. If you are doing this the same time as the last one then the **Properties** window will already be open.

   *The **Properties** window opens.*
6. Click the word **SCH** next to **Table Name**, and then delete it.
7. Type **DJ Piping Schedule,** and then click anywhere in the screen.

   *The **Size Table** name changes.*
8. Type your name in the **Edited By** box.
9. Type **DJ Piping Requested Schedule** in the **Description** box.

Now we are going to edit a Data table.

1. Click the **+** sign next to **Flanges** under **Data Tables** in **Catalog Explorer**.
2. Click the **+** sign next to **Flange Weld Neck.**
3. Double-click **FLG_W_150**.

   *The table displays to the right of the **Catalog Explorer.***
4. Hold CTRL, and select the **2.5000, 5.0000,** and **16.0000** (***65***, ***125***, ***400***) rows.
5. Right-click on a row, and then click **Remove Row**.

   *The **Remove** dialog box displays.*
6. Click **OK**.

   *The **2.5000, 5.0000,** and **16.0000** (**65**, **125**, **400**) rows are removed.*
7. Right-click **FLG_W_150,** and select **Properties**. Alternatively, click the **View** tab, and then select **Properties** in the **Display** panel. If you are doing this the same time as the last one the **Properties** window will already be open.

   *The **Properties** window opens.*
8. Click the word **FLG_W_150** next to **Table Name**, and then delete it.
9. Type **DJ Piping 150 WN Flg,** and, then click anywhere in the screen.

   *The **Size Table** name changes.*
10. Type your name in the **Edited By** box.

Create a New Project

In this lesson we create a project specification for Ezz Piping. There are certain component groups that Ezz Piping does not allow the pipers to use.

1. Open the **Spec Editor** , and click **File > New Project** .

   *The **New Project** dialog box displays.*
2. Type **Ezz Piping Spec** in the **Name** box.
3. Click the **Default Catalog** ellipsis, select **Sample_Inch.cat**, and click **Open**.
4. Select **Fabrication** from the **BOM** (Bill of Material) **type** list.
5. Click **Long Desc Format**.

   *The **Long Description Format** dialog box displays.*
6. Select **Components** from the list.
7. Select **Components Tables** from the list.
8. Use the right arrow to move COMPTYPE_CMP_TBL, and TABLENAME_CMP_TBL from **Table Fields** to **Long Desc.**
9. Click **OK** when finished.
10. Click **Part Number Format**.

    *The **Part Numbers Format** dialog box displays.*
11. Use the right arrow to move COMPONENT_PTN, and ENDTYPE_PTN from **Available Part Number Fields** to **Selected Part Number Fields**.
12. Click **OK** when finished.
13. Type **Specification settings for Ezz Piping** in the **Description** box.
14. Check the location. If you do not want to save the project to the Spec folder click the ellipsis, and save it where you want.
15. Click **OK** to create the new project.

    *The project is created, and displays under **Project Data**.*

Now we need to *Add specification to the project* (on page 61).

Add specification to the project

1. Click the **Home** tab, and then click **Add Specification** on the **Specification** panel. Alternatively, right-click the project name in the **Project Data** window, and click **Add Specification**.

   *The **New Specification** dialog box displays.*
2. Type **150_EZZ PIPE** in the **Name** box.
3. Type **5** in the **Weight Factor** box.

   NOTE The weight factor is used for adding various margins into component weights such as paint and weld weights.
4. Type the component class in the **Component Class** box.
5. Type **175 F** in the **Temperature Rating** box.
6. Type **205 PSI** in the **Pressure Rating** box.

7. Click the **Size Table** list, and then select **Size**.
8. Deselect **8.0000**, **16.0000**, and **28.0000** (***200,400,700***) sizes using the **Available Size** list.
9. Click the **Thickness Table** list, and then select **THCK**.
10. Click the **Thickness Table** ellipsis.

    The **Schedule Assignment** dialog box that displays*.*
11. Click the box to the right of **2.5000** (***65***), and then select **10S**.
12. Click the box to the right of **5.0000** (***125***), and then select **XS.**
13. Click the box to the right of **6.0000** (***150***), and then select **XXS.**
14. Click the box to the right of **14.0000** (***350***), and then select **STD.**
15. Click **OK** when finished.
16. Click the **Material Table** list, and then select **MAT**.
17. Click the **Layer Color** list to select **Blue**.
18. Type your name in the **Created By** box.
19. Type **Ezz Piping's 150 Blue Spec** the **Description** box.
20. Click **OK**.

    The new spec is added under the project name in ***Project Data****.*

We now need to *Add a component to the spec* (on page 62).

Add a component to the spec

1. Double click the spec in the **Project Data** pane.

   The spec displays in the main window with the ***New Component*** *dialog box visible.*

   NOTE Some of the information in this screen defaults to the settings from when the spec was initially set up.
2. Click the **Group** list, and then select **Caps**.
3. Click the **Type** list, and then select **Plug**.
4. Click the **Data Table** list, and then select **PLG_T_30C**.
5. Click the **Sizes** type list, and then select **5.0000** (***125***)**,** and **6.0000** (***150***).
6. Type PL-0511 in **Tag.**
7. Type **Blue Plug for Ezz Piping** in **Notes** information as needed in their respective boxes.
8. Click **Add** at the top-right hand side of the **New Component** pane.

   The component is added to the list in the main window.

Now that we have added a single component we can *Add the remaining components to the spec* (on page 63).

Add the remaining components to the spec

1. Double-click the spec in the **Project Data** pane.
   The spec displays in the main window with the **New Component** *dialog box visible.*
2. Right-click the component listing area of the screen, and then select **Add Multiple**.
   The ***Add multi component*** *dialog box displays.*

   NOTE Some of the information in this screen defaults to the settings from when the spec was initially set up.
3. In the **Component** box, select all the check boxes except **Sanitary** and **Spec Blind/Bleed Ring/Exp Joint**.
4. Click the plus **+** sign on **Caps,** and click the check-box for **Plug** to clear it.
5. Click **Apply same end-type on all ends**.
6. Click the **Start** list, and then select **BW**.
7. Click the **Main Schedule** list, and then select **STD**.
8. Click the **Branch Schedule** list, and then select **STD.**
9. Type **EZZ-Comps** in **Tag.**
10. Type **Ezz Piping does not allow for the use of Spec Blind/Bleed Rings/Exp Joints as well as Sanitary Components. All Ezz Piping's End Types are Buttweld.** in **Notes**.
11. Click **Add** at the top-right hand side of the **Add Multi Component** pane.
    The components are added to the list in the main window.

CAUTION When setting the available component size range, make sure that it corresponds with the size ranges available for each selected data table.

Now we must *Define the data tables and branches for the components* (on page 63).

Define the data tables and branches for the components

1. Click the plus **+** sign next to **Caps**, and then select the first red exclamation point.
   The line is highlighted.
2. Click the **Data Tables** list in **Edit Component** dialog box, and then select a data table.
3. Click **Apply**.
4. Repeat this procedure for the remaining red exclamation points.

Now we must set the branch table per Ezz Piping's request. Ezz Piping uses Reducing Tee / Cross for all sizes. They use O-Let, and Half Coupling for anything under 12" (300mm), and they use Stub-In / Set-On / OTAP for 6" (150mm), and below. For components with the same sizes both options available can be used.

1. Click the **Branch** tab at the bottom of the **Add Multi Component** pane.
   A grid displays with ***Branch Size (NPS)****, and* ***Main Size (NPS)*** *on the top, and side.*
2. Right-click the box at the intersection of **64** (***1600***), and **0.125** (***3***), and then add **Reducing Tee /Cross**.
   The grid box changes displaying that you have added a component.
3. Continue to adjust the table as needed with the information from above. SHIFT+click and CTRL+click can be used to select multiple intersections at a time.
4. Click **File** > **Save** when finished. Every time a change is made it must be saved to take affect.

Edit an existing project

In this lesson, we are updating a predefined product to match the Catalog we edited for DJ Piping. This way we now have a project and a catalog for DJ Piping based off of the original specifications.

1. Click **File** > **Open**  .
2. Click **Sample_Inch.prj,** and click **Open**.

   Click **File** > **Save As**  .

   CAUTION If you do not save changes to a different file, then you are making changes to a delivered specification. If you save these changes directly to **Sample_Inch.prj** you may have to re-install CADWorx to return your setup to default. It is highly recommended that you save this using another name before continuing.
3. Type **Inch Example Tutorial**, and click **Save**. You can choose to save this outside of the Spec Folder in a folder on your computer.

If you want to practice adding new specs to an existing project use the steps in *Create a New Project* (on page 61). The following tutorial demonstrates how to edit the 150 Spec, and the 600 Spec. We also edit the properties for some of the options.

1. Select **150** under **Sample Inch** in **Project Explorer**.

   *The table displays to the right of the **Project Explorer.***
2. Hold CTRL and select the **Spec Blind/Bleed Ring/Exp Joint** and **O-Lets.**
3. Right click and then click **Remove**.

   *The **Remove** dialog box displays.*
4. Click **OK**.

   *The **Spec Blind/Bleed Ring/Exp Joint** and **O-Lets** groups are removed.*
5. Right-click **150,** and select **Properties**. Alternatively, click the **View** tab, and then select **Properties** in the **Display** panel.

   *The **Properties** window opens.*
6. Click the number **150** next to **Specification Name**, and then delete it.
7. Type **DJ Piping 150 Specs**, and then click anywhere in the screen.

   *The **Specification Name** changes.*
8. Click the **Color** list, and select **Green**.

   *The **Apply to all components** dialog box displays.*
9. Click **Yes**.
10. Click the **Default Flange** list, and then select **Flange Weld Neck.**
11. Type your name in the **Edited By** box.
12. Type **DJ Piping Requested Spec Changes** in the **Description** box.

Now we are going to edit the 600 Spec.

1. Select **600** under **Sample Inch** in **Project Explorer**.

   *The table displays to the right of the **Project Explorer.***
2. Hold CTRL and then select the **Spec Blind/Bleed Ring/Exp Joint, Couplings, Caps, Unions,** and **O-Lets.**
3. Right-click and then click **Remove**.

   *The **Remove** dialog box displays.*

4. Click **OK**.

   *The **Spec Blind/Bleed Ring/Exp Joint, Couplings, Caps, Unions,** and **O-Lets** groups are removed.*
5. Right-click **600,** and select **Properties**. Alternatively, click the **View** tab, and then select **Properties** in the **Display** panel.

   *The **Properties** window opens.*
6. Click the number **600** next to **Specification Name**, and then delete it.
7. Type **DJ Piping 600 Specs**, and, then click anywhere in the screen.

   *The **Specification Name** changes.*
8. Click the **Color** list, and then select **Cyan**.

   *The **Apply to all components** dialog box displays.*
9. Click **Yes**.
10. Click the **Default Flange** list, and then select **Flange Weld Neck.**
11. Type your name in the **Edited By** box.
12. Type **DJ Piping Requested Spec Changes** in the **Description** box.
13. Click **File** > **Save**.

Now change the 300 Spec per the following DJ Piping information

- 300 spec does not use **Caps, Unions,** and **O-Lets.**
- 300 is requested by DJ Piping.
- 300 uses a red color for its components.
- 300 has a Flange Weld Neck as its default.
- 300 description is **DJ Piping Requested Spec Changes.**

Optional components

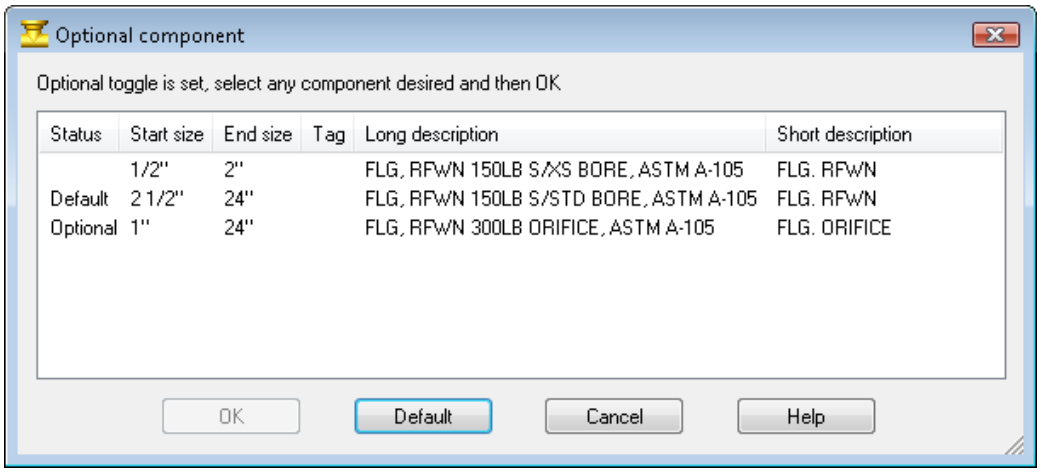
To make a component an optional component do the following.

1. Open the **Spec Editor**  .
2. Open the **Inch Example Tutorial** project.
3. Click the **150** spec, and then select a component.

   *The **Edit Component** dialog box opens.*
4. Click the **Optional Component** check-box, and then click **Apply**.

NOTES

- As long as the **Specification Option Toggle** is on, a dialog box similar to the example below opens when you place 300# orifice flanges.

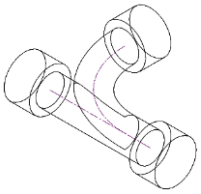


- **Specification Option Toggle** also provides the ability to place a schedule 80 8" (200mm) pipe schedule STD is the 8" default. Thus, making the 8" *(200mm)* pipe schedule 80.
- There are many other useful functions within the Specification Editor that are not discussed in this lesson. For more information about all of the available functions, see the *Specification Editor* in the *CADWorx Plant User's Guide* or any topic in the *CADWorx Spec Editor Users Guide*

User Shapes

In this lesson, we create a typical user shape and add it to a specification. CADWorx automatically creates all the data files and blocks that are required. You can use the Tutor12.dwg *(Tutor12m.dwg)* file delivered with the software, or create the drawing shown below using the following guidelines.

- Use the **ROUTE** command to draw the main body of the swept tee.
- Use the **Adjustable Bend** command on the **Buttweld** tool bar to draw the swept part. Draw both parts at 4" (100mm), and then use the AutoCAD **UNION** command to join them together.
- Use **Pipe** command (**PIPW**) to draw the hubs with the size set to 6" (150mm).
- Make sure **PICKSTYLE** is turned off, and the centerlines are erased.



NOTE The size of this component is 4" (100mm). It is a typical PVC swept tee for plumbing purposes.

Create the component

1. Click **Create**  . Alternatively, type **USERCREATE** on the command line, and press ENTER.
   *The **Define User Shape** dialog box displays.*
2. Type **SWEPT TEE** in the **Name** box.
3. Select user shape number **1** in the **Number** list.
4. Select **150** in the **Specification** list.
5. Assign the user shape to **300** using the **Assign to Spec** list.
6. Click the **Save to catalog** check box.
   NOTE This is the default setting.
7. Select **Block** from the **Type** list.
8. Select **4** *(100mm)* in the **Main Size** list.
9. Click **Select 3D Solid**.
   Select 3D solid representation.
10. Click a point in the upper left of the drawing, drag the mouse, and then click in the lower right of the drawing to select the entire user shape. Press ENTER to finish the selection.
    Pick insert point.
11. Select the lower right part of the swept tee for the insertion point on the component.
12. Click **Define Connections**.
    Pick direction.
13. Select the opposite direction away from the component.
    Set End Type for this connection: [Buttweld/Socket/Threaded/Flanges]<Butt weld End Type>.
14. Press ENTER to select the default.
    Pick connection 1 or <Enter to finish>.
15. Pick the point at the upper left connection of the main run of the same component.
    Pick direction.
16. Select the opposite direction away from the component.
    Set End Type for this connection: [Buttweld/Socket/Threaded/Flanges]<Butt weld End Type>.
17. Press ENTER to select the default.
    Pick connection 2 or <Enter to finish>.
18. Pick the point at the branch connection of the component.
    Pick direction.
19. Select the opposite direction away from the component.
    Set End Type for this connection: [Buttweld/Socket/Threaded/Flanges]<Butt weld End Type>.
20. Press ENTER to select the default.
    Pick connection 3 or <Enter to finish>.
21. Press ENTER to finish.
    *The **Define User Shape** dialog box displays.*
    TIP Connection points indicate where other equipment or components attach to the nonsymmetrical user shape. These points are very important when exporting the model to an automatic isometric and affect the scaling if not properly selected.
22. Select **Misc** in the **BOM Type** list.

23. Select **White** in the **Color** list.
24. The layer should display as **150** in the **Layer** box.
25. Type **0.2838** *(**7.8417**)* in the **Density** box.
26. Type **15** *(**6.8**)* in the **Weight** box.
27. Type **1'-11.8125** (***605***) in the **Length** box.
28. Select **Optional Component**.
29. Type **Swept Tee** in the **Long** box. The long description is used within the data file.
30. Type **SWTEE** in the **Short** box.
31. Type **SWT-001** in the **Tag** box.

    NOTE The software automatically defines the next five options.
32. Click **OK**.

    The software creates the user shape, and adds it to the specification.

    NOTE *If the reducing option had been used, the two blocks would have names such as SWEPTTEE[4.0x3.0-2D].dwg.*

Draw the component

For any user shape, there are two options that insert it into the drawing. You can use the **User 1** through **User 5** commands on the **Misc** toolbar or you can select the user shape in the Spec View palette under **User Shapes**.

1. Click **Main Size** . Alternatively, type **SETSPEC,** or **SETSIZE** on the command line, and then press ENTER.

   *The **Set Specification and Size** dialog box appears.*
2. If a project is already selected skip to step 3. If the **Project** box is empty, click **Browse**, select **Sample_Inch.prj**, and then click **Open**.
3. Select **150** under **Specifications**, and **4** (***100***) under **M** for main size.

   NOTE If you are working in Metric units, open the 150MM specification file.
4. Click and hold **Create** on the **Misc** toolbar, and then select **User 1** on the fly-out menu.

   Pick point or [Change connection]<last point>.
5. Pick a point in the drawing.

   NOTE The next set of prompts allow you to rotate the component, mirror it in the X-, Y-, or Z-axis, and roll it up, down, or at any angle of your choosing. As such, the component can be placed in the drawing in virtually any position.

   Pick rotation or [Up/Down/Roll/mirrorX/mirrorY/mirrorZ].
6. Type **X** on the command line, and press ENTER to select the **mirrorX** option.

   Pick rotation or [Up/Down/Roll/mirrorX/mirrorY/mirrorZ].
7. Pick a rotation.

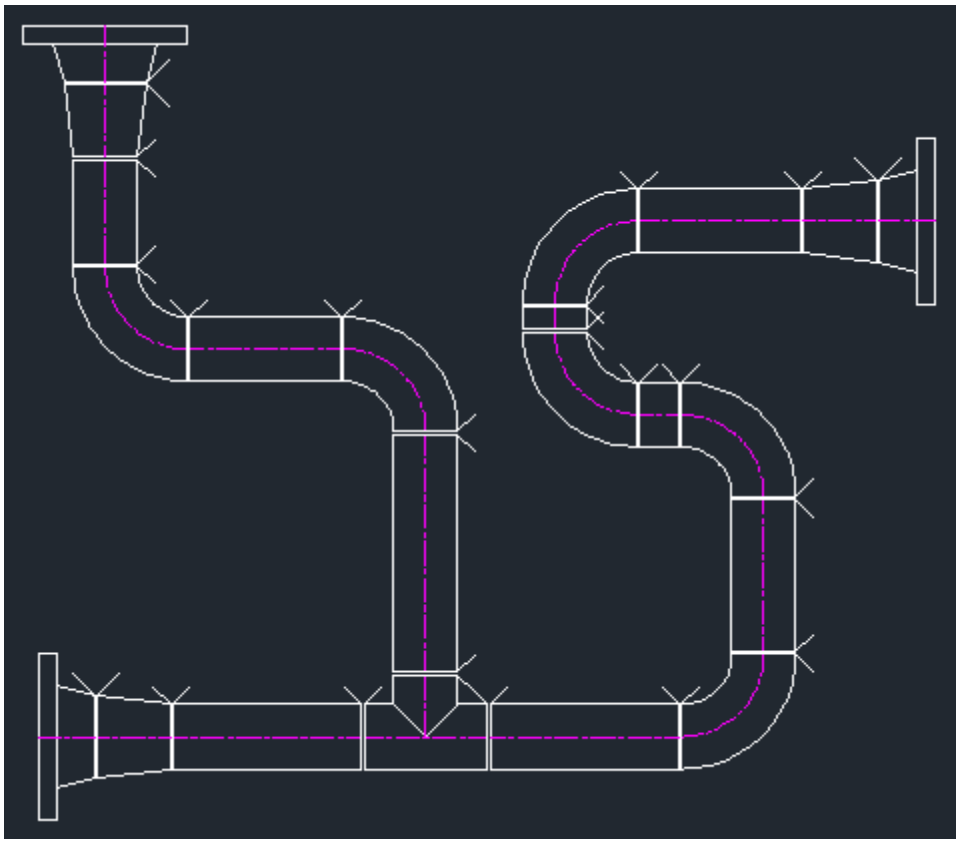
   The user shape is placed in the drawing.

Change the size of a model

The following tutorial demonstrates how to change the size of a model. Draw a pipe run with three connections. Each connection end has a reducer and a flange. One of the connections should

extend from the middle of the pipe using a tee. The Specification for the drawing is 150, the main size is 4" (100), and the reducing size is 3" (80).

1. Use the **ROUTE** command to draw a pipe run similar to the one below.

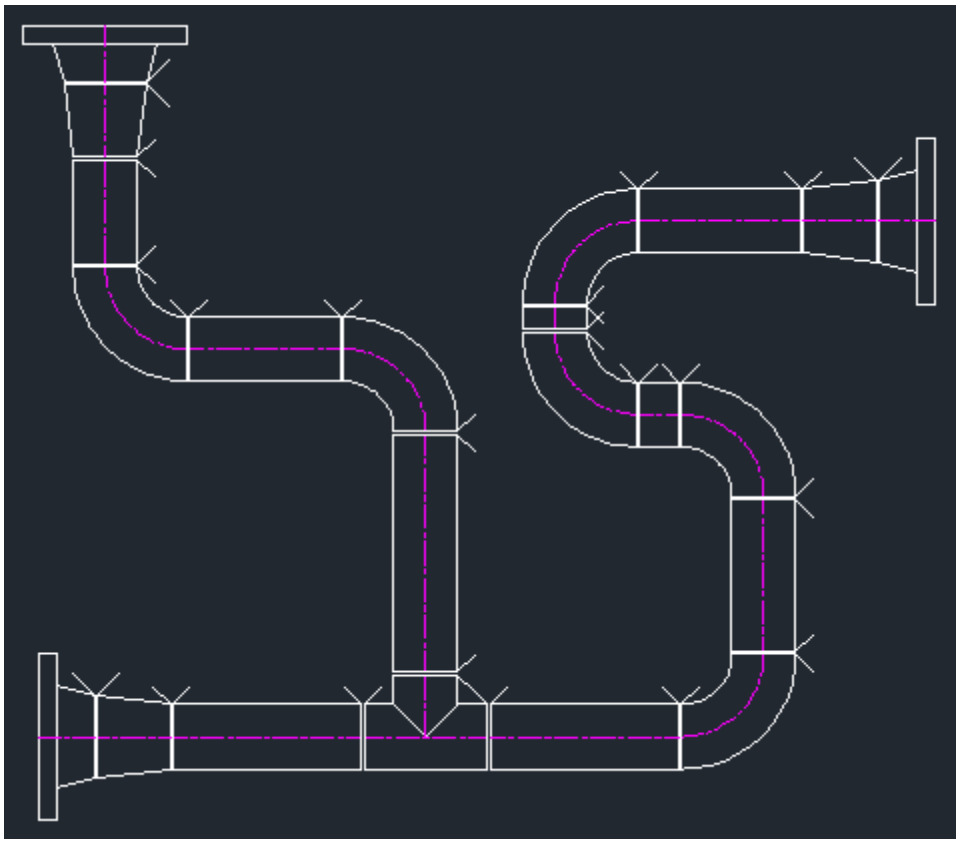


2. Click the **CADWorx Plant I** tab, and then click **Change Size** on the **Setup Size/Spec** panel. Alternatively, type **CHANGESIZE** on the command line, and then press ENTER.
   Select mode[Automatic/Manual]<Automatic>.
3. Press ENTER for automatic.
   Select line to change.
4. Select the line to the right of the tee.
   The size list displays.
5. Select **2"** (50).
   Select branches to change size or <Enter to finish>.
6. Select the bottom tee, and then press ENTER to finish.
   Enter an option [Change component/Reducing/None].
7. Type **C** on the command line, and then press ENTER.
8. Repeat step 7 for the other two reducers.
   Select a component to keep its position or [Enter to finish].
9. Press ENTER to finish.
   The drawing changes size.

Change the specification of a model

The following tutorial demonstrates how to change the spec of a model we create. Draw a pipe run with three connections. Each connection end has a reducer and a flange. One of the connections should extend from the middle of the pipe using a tee. The Specification for the drawing is 150, the main size is 4" (100), and the reducing size is 3" (80).

1. Use the **ROUTE** command to draw a pipe run similar to the one below.



2. Click the **CADWorx Plant I** tab, and then click **Change Spec** on the **Setup Size/Spec** panel. Alternatively, type **CHANGESPEC** on the command line, and then press ENTER.

   Select mode[Automatic/Manual]<Automatic>.

3. Press ENTER for automatic.

   Select line to change.

4. Select the line above the tee.

   The spec box displays.

5. Select **300**.

   Select a component to keep its position or <Enter to finish>.

6. Press ENTER to finish.

   The line above the tee changes colors indicating the spec has changed.

Having changed the relief line to 300 we are now going to change the main line to 600

1. Type **CHANGESPEC** on the command line, and press ENTER.

Select mode[Automatic/Manual]<Automatic>.

2. Type **M** for manual.
   Select components.
3. Select all the components on the main line, and then press ENTER when finished.
   The spec box displays.
4. Select **600**.
   Select a component to keep its position or <Enter to finish>.
5. Press ENTER to finish.
   The spec changes on the main line.

Change the spec of a model with branches using Automatic

1. Click the **CADWorx Plant I** tab, and then click **Change Spec** on the **Setup Size/Spec** panel. Alternatively, type **CHANGESPEC** on the command line, and then press ENTER.
   Select mode[Automatic/Manual]<Automatic>.
2. Press ENTER for automatic.
   Select a piping line.
3. Select a pipe line.
   The spec box displays.
4. Select a specification.
   Select branches to change spec or <Enter to finish>.
   NOTE If you decide not to select the branches in step 5, the spec still changes on the branching components but does not change on the branches themselves.
5. Select the branches.
   Select a component to keep its position or <Enter to finish>.
6. Select a component or press ENTER to finish.
   NOTE If you selected a component you are prompted again, and need to repeat step 5.
   The selected system changes specification up to any reducer locations.

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