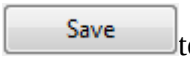
# Tutorial in US and SI Units

This tutorial will take you through the process of modeling a small piping system from start to finish and demonstrate most of the program features. New users are strongly encouraged to run through the tutorial.

The tutorial is written with the U.S. customary units shown first followed by the equivalent value in S.I. units.

# Create and Save a New Project

It is recommended that you save your new project as soon as it is created.

1. Open the software and create a new project file:
    a) Click the **New File** button  in the **Main Toolbar**, or
    b) Go to the **File Menu** and select **New**.
2. Save the file with the desired name and location:
    a) Click the **Save File** button  in the **Main Toolbar** or go to the **File Menu** and select **Save** or **Save As**.
    b) Navigate to the **Projects** folder and enter **North-South Transfer System** in the **File name** field.
    c) Click the **Save** button  to create the project file.

# Customize the User Interface

The **User Interface** consists of **Menus** located above the **FLO-Sheet**, along with **Toolbars** and **Windows** that can be displayed, hidden, or docked anywhere around the **FLO-Sheet**.

### Show and Hide a Toolbar and Window:

1. **Toolbars**
    a) In the **View Menu**, hover over **Toolbars** and select the **Toolbar** to display it (**checked**) or to hide it (**un-checked**).
2. **Windows**
    a) For a **Window** that is being displayed, it can be hidden from the **View Menu** or by clicking on  on the right side of the **Window** title bar.
    b) Alternatively, **Right Click** on any **Toolbar** or the title bar of any displayed **Window** to open a drop down list of all available **Toolbars** and **Windows**. Click on the desired **Toolbar** or **Window** to show or hide it.

### Dock and Undock a Toolbar and a Window:

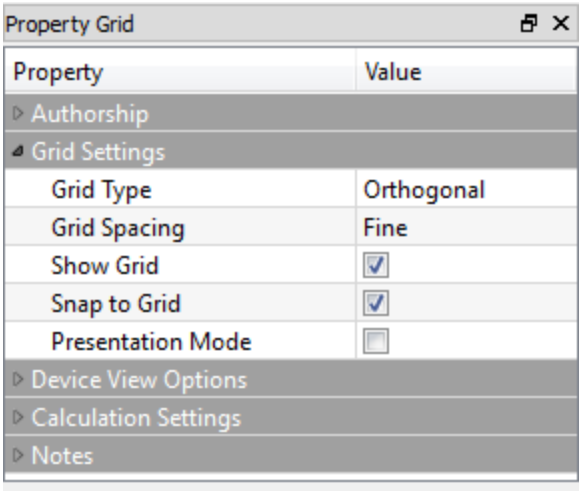
1. **Toolbars**
    a) Click and hold down the left mouse button on the dotted separator line at the beginning of a **Toolbar**.
    b) Dock the **Toolbar** by dragging it to the far right, far left, top and bottom of the workspace until a blue rectangle appears to indicate where the **Toolbar** can be docked, then release the mouse button to dock it at that location.
    c) Undock the **Toolbar** by releasing the mouse button anywhere except on the blue rectangle.

2. **Windows**
    a) Click and hold down the left mouse button on **the Message Window** title bar.
    b) Dock the **Message Window** by dragging it around the workspace until a blue rectangle appears to indicate where the **Window** can be docked, then release the mouse button to dock it at that location.
    c) Drag the **Message Window** and place it on top of the **List View Window** to create tabbed **Windows**. Click the tabs at the bottom to switch between the two **Windows**.
    d) Undock the **Message Window** by clicking on on the right side of the title bar, or by dragging and dropping the **Window** anywhere except on the blue rectangle

### Adjust the Size of an Undocked Window

1. Undock the **Message Window**.
2. Click on a corner or edge of the **Window**.
3. Drag the corner or edge until the **Window** is at the desired size.

# Set up the FLO-Sheet




There are several options to determine how the **FLO-Sheet** will look and what information will be displayed on the **FLO-Sheet**.

### Define the Grid Settings

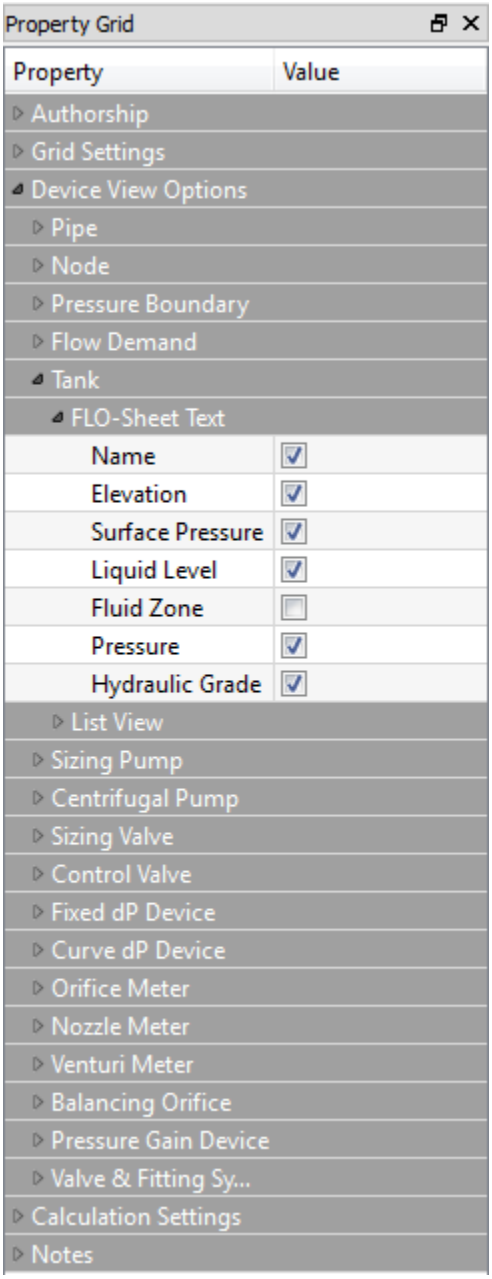
To customize how the **FLO-Sheet** will look:

1. Click on an empty space on the **FLO-Sheet**.
2. In the **Property Grid**, click the triangle beside **Grid Settings** and set the following:
    a) **Grid Type**: **Orthogonal** (the other option is Isometric)
    b) **Grid Spacing**: **Fine** (the other options are **Standard** and **Coarse**)
    c) **Show Grid**: **check** the box to show the grid
    d) **Snap to Grid**: **check** the box to snap to grid

### Set the Device View Options

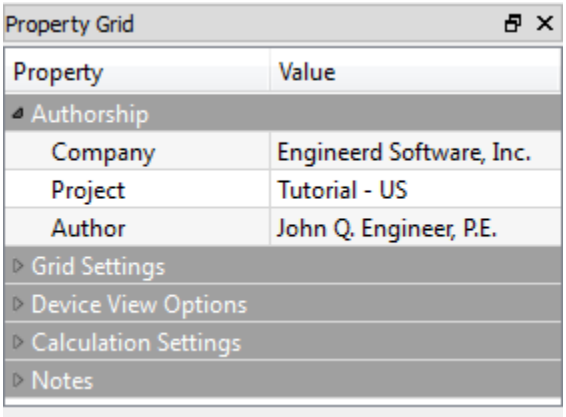
1. To customize the amount of information to display on the **FLO-Sheet** for each **Device**: Click on an empty space on the **FLO-Sheet**.

2. In the **Property Grid**, click the triangle beside **Device View Options**
    a) Expand the section for each **Device**, then expand the **FLO-Sheet Text** and check the selection box for each parameter to display.
    b) Expand the **Valve and Fittings Symbols** section and check the box beside all the types to display on the **FLO-Sheet**.



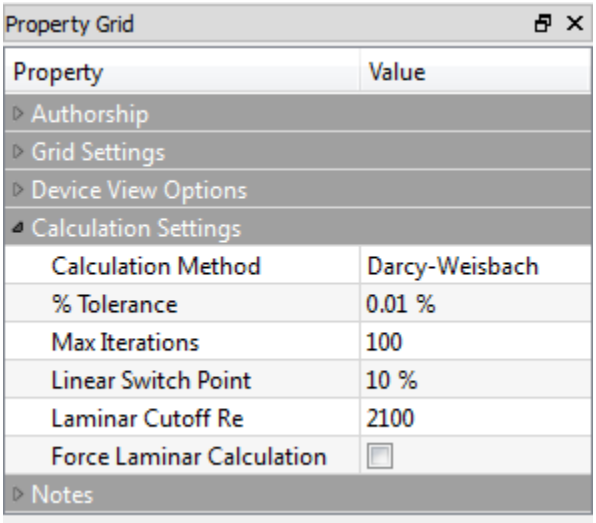

# Define the Authorship Settings

1. In the **Property Grid** for the **FLO-Sheet**, click the triangle beside **Authorship**
2. Enter your **Company** name
3. Enter the following **Project** name: **North-South Transfer System**
4. Enter **your name** as the **Author**

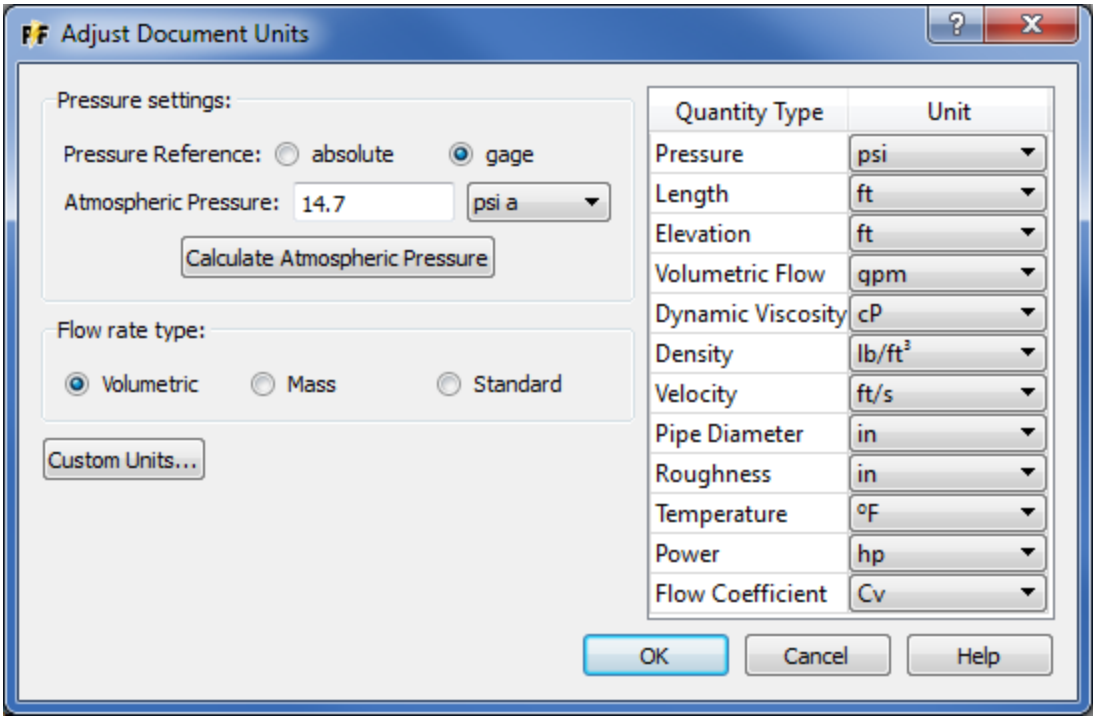



# Define Calculation Settings

1. In the **Property Grid** for the **FLO-Sheet**, click the triangle beside **Calculation Settings.**
2. For the **Calculation Method** select **Darcy-Weisbach** (other option is **Hazen-Williams**).
3. Leave the following settings at their default values. These can be changed if the software cannot converge to a solution.
    a) **% Tolerance: 0.01%**
    b) **Max Iterations: 100**
    c) **Linear Switch Point: 10%**
    d) **Laminar Cutoff Re: 2100**
    e) **Force Laminar Calculations: Unchecked**



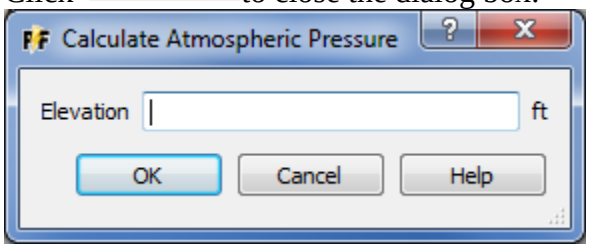

# Define the Document Units



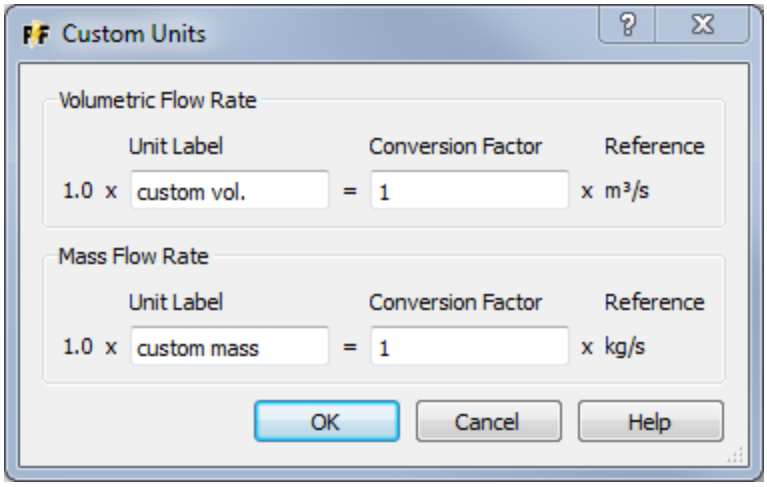

The project **Document Units** are used as the default units for the displayed results on the **FLO-Sheet**, the calculated results in the **Property Grid**, and the user-entered **Design Data** for all **Devices** in the **Property Grid**.

## Define the Document Units:

1. Click the **System Menu** and select **Units** to open the **Adjust Document Units** dialog box.
2. In the **Adjust Document Units** dialog box:
    a) Set the **Pressure Reference** to **gage**.
    b) Click Calculate Atmospheric Pressure to open the **Calculate Atmospheric Pressure** dialog box
        i. Enter an **Elevation** of **500 ft (152 m)**
        ii. Click OK to close the dialog box.



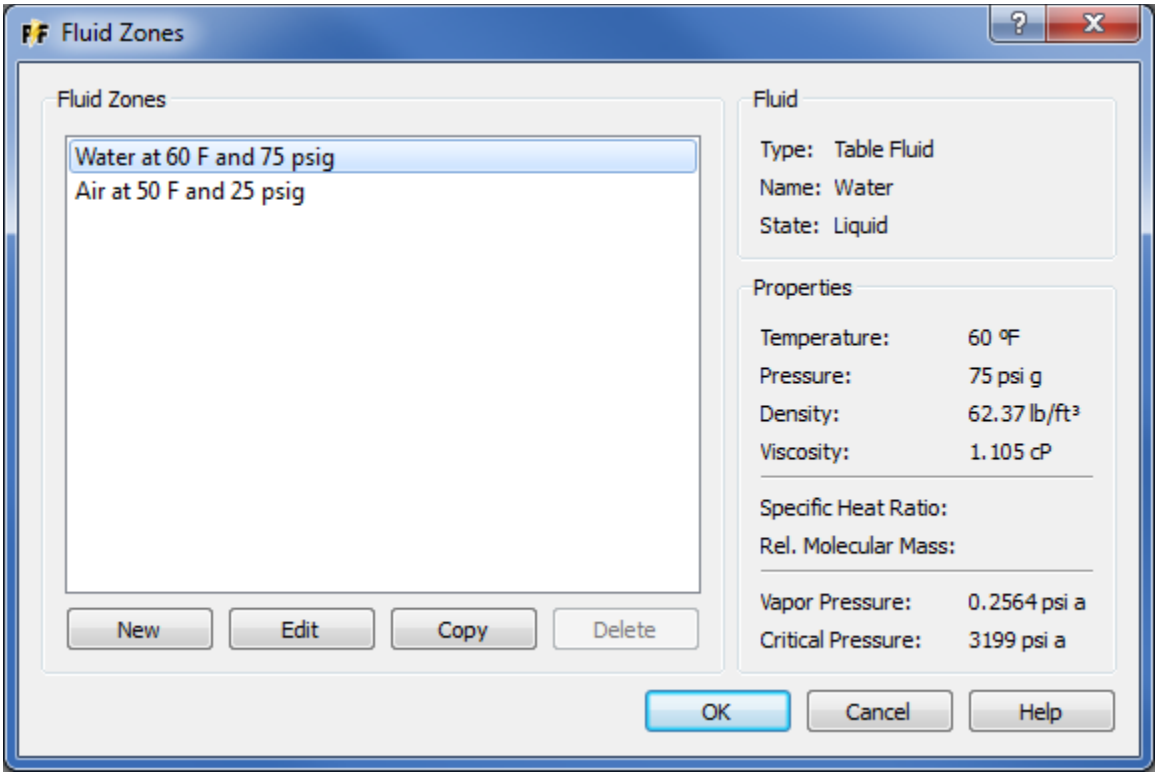

        iii. Review the calculated value of **Atmospheric Pressure**.
        iv. Over-ride this value by entering **14.7psia (101 kPa)**.
        v. Note that the unit for **Atmospheric Pressure** can be changed with the drop down menu.
    c) Select **Volumetric** as the **Flow Rate Type**




d) Click Custom Units... to open the **Custom Units** dialog box.
   i. Enter **Smoot^3/sec** in the **Unit Label** field for a custom Volumetric Flow Rate.
   ii. Enter **2.38e8** for the **Conversion Factor**.
   iii. Click OK to close the **Custom Units** dialog box.
   iv. Click the drop down menu of the **Volumetric Flow** unit and scroll down to see that the **Custom Unit** is now available to choose from.
   v. Select **gpm** for the **Volumetric Flow** rate unit.

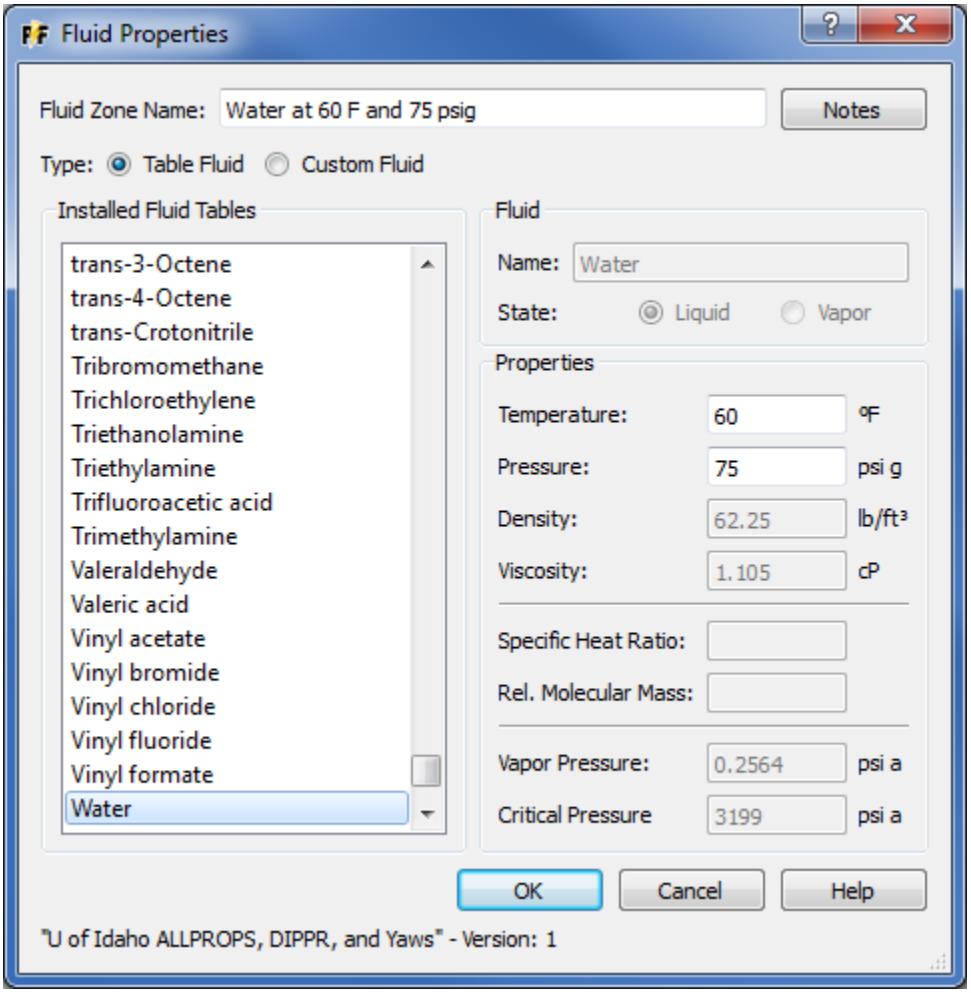
3. Click the drop down menu for a list of available units for each quantity. Accept the default units at this time.
4. Click OK to close the **Adjust Document Units** dialog box.

# Define a Fluid Zone

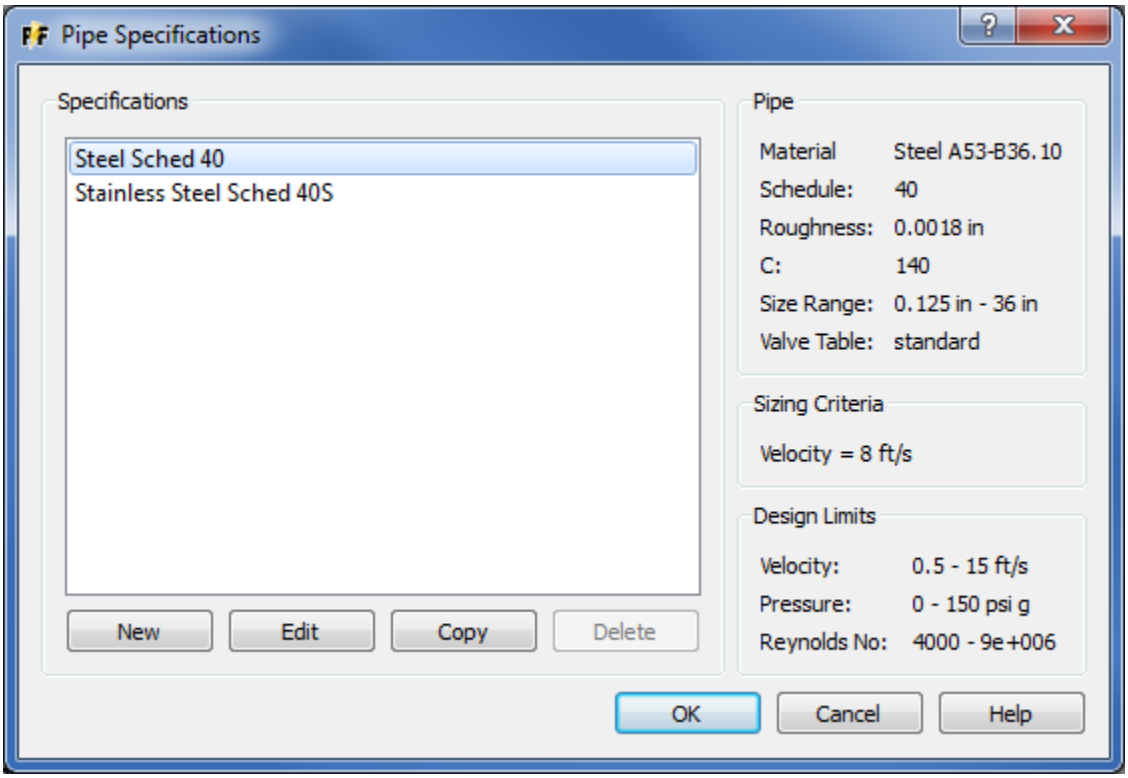


**Fluid Zones** provide key fluid property information for calculations throughout the model.

1. Click the **System Menu** and select **Fluid Zones** to open the **Fluid Zones** dialog box.
2. Click [New] to open the **Fluid Properties** dialog box.
3. Enter **Water at 60 F and 75 psig (Water at 15 C and 517 kPag)** in the **Fluid Zone Name** field.
4. Select **Table Fluid** as the **Type** (other option is **Custom Fluid**)
5. Scroll down the list of **Installed Fluid Tables** and select **Water**.
6. Specify **60** °F **(15** °C**)** in the **Temperature** field.
7. Specify **75** psig **(517** kPa**)** in the **Pressure** field.
    a) The software calculates the remaining fluid properties using the formulas in the **Fluid Table**.
    b) Note the values of **Density**, **Viscosity**, **Vapor Pressure**, and **Critical Pressure** (**Specific Heat Ratio** and **Relative Molecular Mass** are calculated if the fluid is a gas or vapor.)
8. Click [OK] to close the **Fluid Properties** dialog box
9. Click [OK] to close the **Fluid Zones** dialog box.



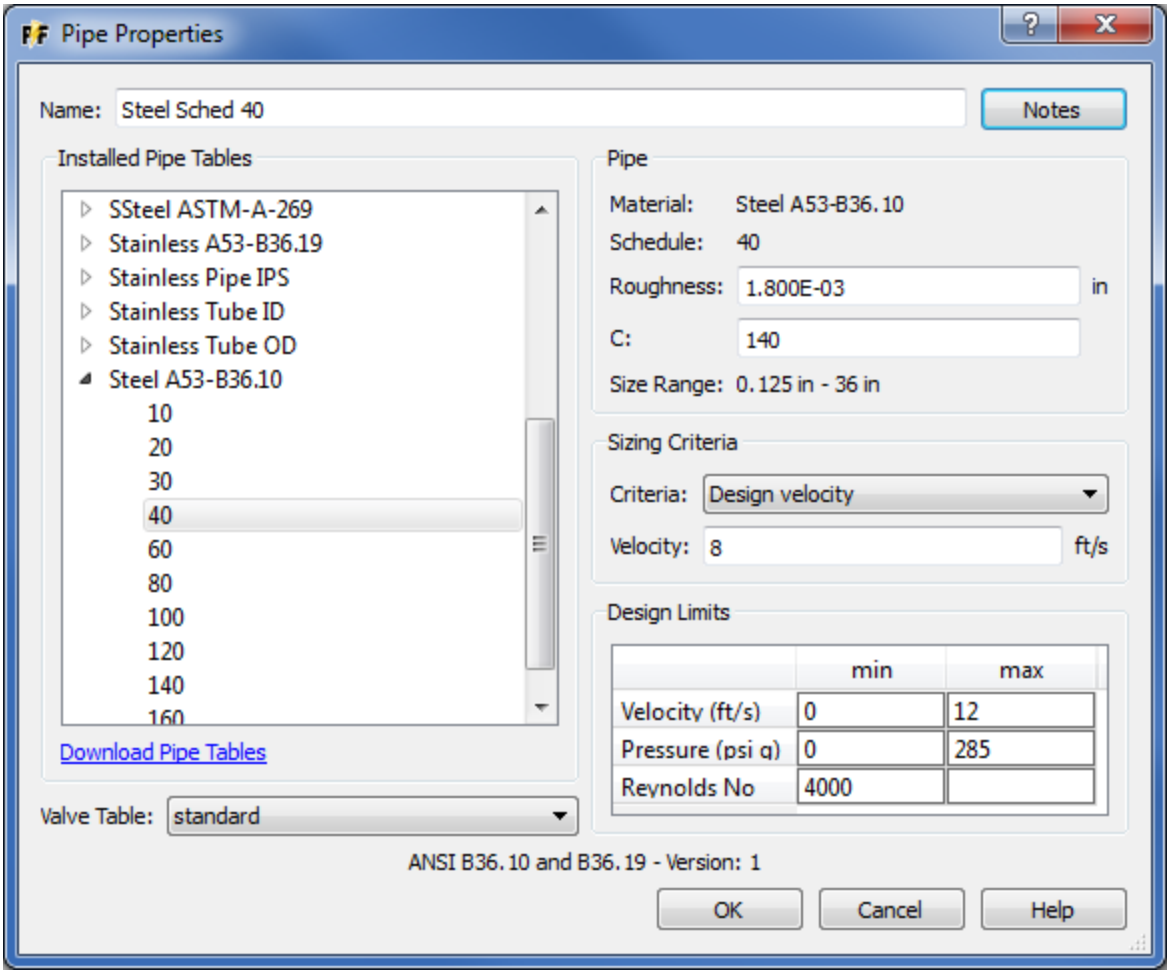
# Define a Pipe Specification




**Pipe Specifications** are used to define the pipe material, schedules, pipe sizes, the valves and fittings **Valve Table**, and the pipe **Sizing Criteria** and **Design Limits**.

1. Go to the **System** menu and click **Specifications** to open the **Pipe Specifications** dialog box.
2. Click 

to open the **Pipe Properties** dialog box.
3. Type **Steel Sched 40** in the **Name** field.
4. Scroll down the list of the **Installed Pipe Tables** and click the arrow next to **Steel A53-B36.10**.
5. Scroll down and select **40** from the list of available schedules.
    a) Note the values of **Absolute Roughness** and the **Hazen-Williams C Factor** are default values taken from the **Steel A53-B36.10 Pipe Table** and can be over-ridden if desired.
    b) Note the displayed **Size Range** for the Pipe Material and Schedules.




6. In the **Sizing Criteria** drop down list:
    a) Select **Design velocity** (other options are **Head Loss** per 100 units of length or **Pressure Drop** per 100 units of length)
    b) Specify **8** ft/sec (**2.4** m/s) in the **Velocity** field
7. In the **Design Limits** fields:
    a) Specify **0** and **12** ft/sec (**0** and **3.7** m/s) in the **Velocitymin** and **max** fields (respectively).
    b) Specify **0** and **285** psig (**0** and **1965** kPag) in the **Pressuremin** and **max** fields (respectively).
    c) Specify **4000** in the **Reynolds No min** field and leave the **max** field blank
8. Open the drop down menu for **Valve Table** and review the available **Valve Table** to choose from. Select the **StandardValve Table**.
9. Click OK to close the **Pipe Properties** dialog box.
10. Click OK to close the **Pipe Specifications** dialog box

# Draw the Piping System Model

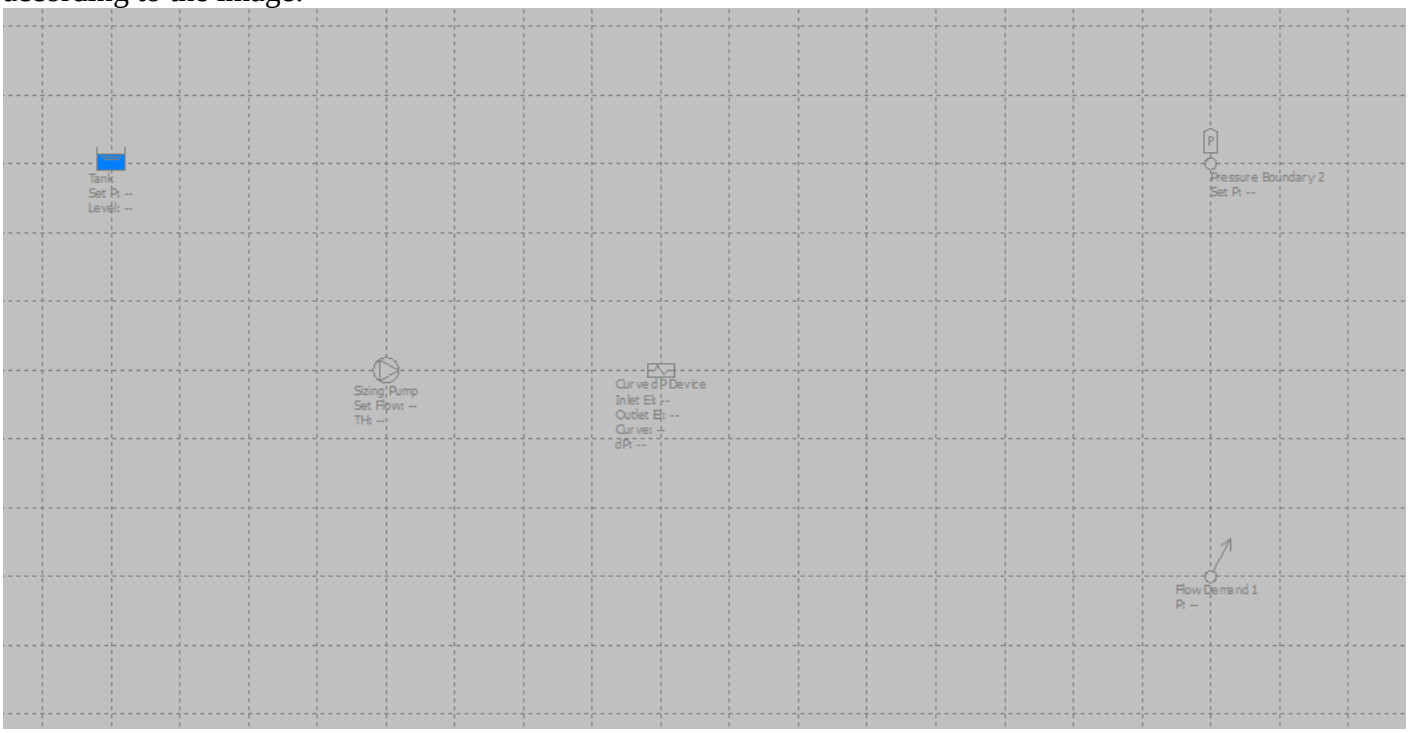
The **PIPE-FLO Interface** allows the piping system model to be drawn quickly by placing all the **Devices** on the **FLO-Sheet** without entering any **Design Data**. A default **Device Name** is automatically entered and incremented to create a unique name for each **Device.** The user can over-ride these names and enter their desired name when entering the **Design Data**.

Alternatively, the user can draw each **Device** and then immediately enter the **Design Data** before adding another **Device**.

The following procedure will have you draw the system first, then go back and enter the **Design Data**.

# Place All Devices on the FLO-Sheet

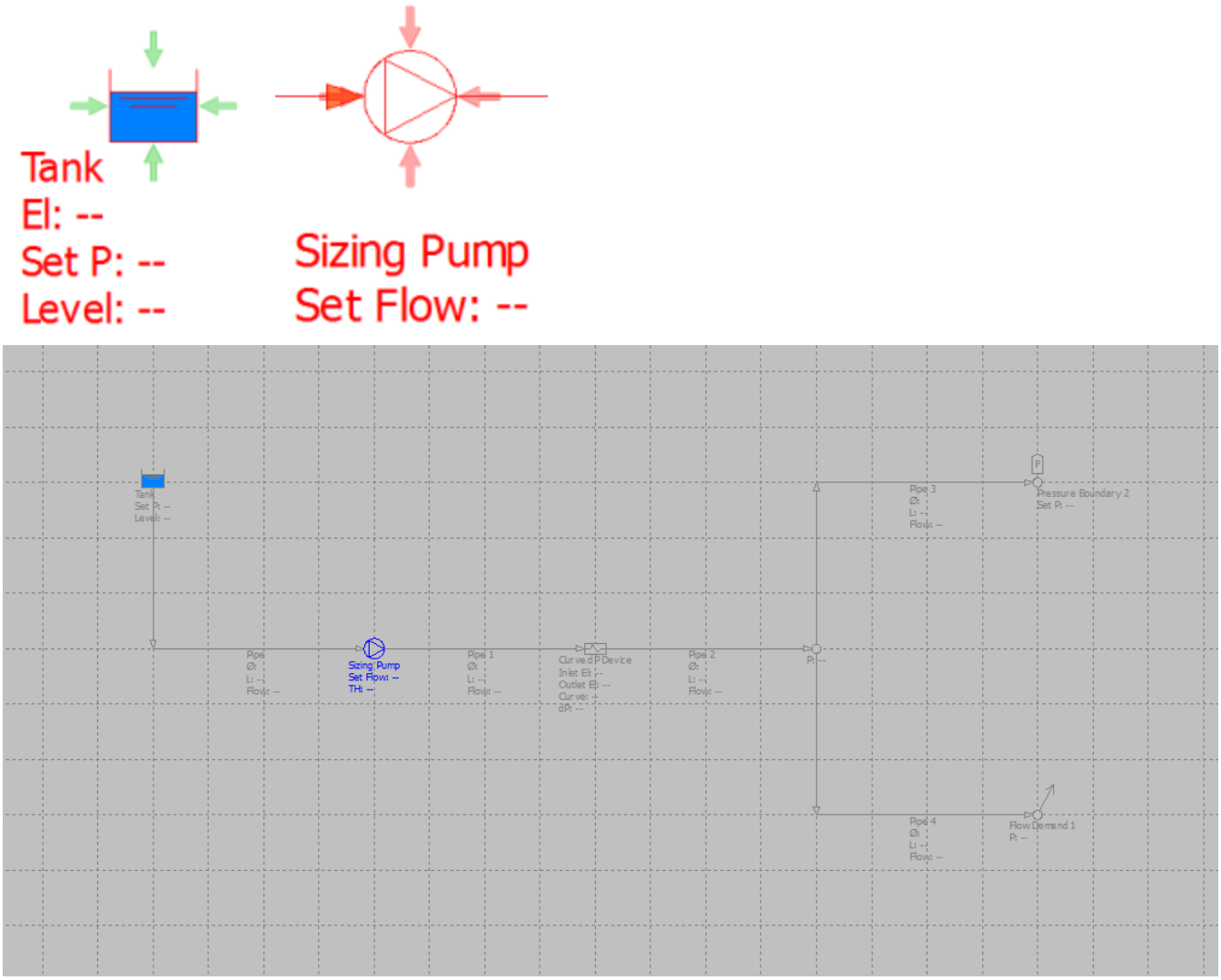
Select the following **Devices** in the **Toolbox** window and place them on the **FLO-Sheet** roughly located according to the image:




1. A **Tank** on the **FLO-Sheet** Tank
2. A **Sizing Pump** Sizing Pump
3. A **Curve dP Device** Curve dP Device
4. A **Flow Demand** Flow Demand
5. A **Pressure Boundary** Pressure Boundary

# Connect Devices Together with Pipelines

Select **Pipe** Pipe in the **Toolbox** window and connect each **Device** together according to the drawing. When the cursor hovers over a **Device** the **Cursor Indicator** changes to four green arrows pointing in toward the **Device** if the connection is allowed, and to four red arrows if the **Device** already has the maximum number of pipes connected and the connection is not allowed.




1. **Tank** to **Sizing Pump**
    a) Click on the first **Tank** on the left of the **FLO-Sheet**.
    b) Move cursor down and click on a grid line to create a vertex.
    c) Move the cursor to the right and click on the **Sizing Pump** to make the connection.
2. **Sizing Pump** to **Curve dP Device**
    a) Click on the **Sizing Pump**.
    b) Move the cursor to the right and click on the **Curve dP Device** to make the connection.
3. **Curve dP Device** to the **Node**
    a) Click on the **Curve dP Device**
    b) Move cursor to the right and double click to end the **Pipe** segment and create a **Node**. (this is another way of creating **Nodes**)
4. Connect the **Node** to the **Pressure Boundary**
    a) Click on the **Node** to start another **Pipe** and move the cursor up and click on a grid line to create a vertex, then to the right and click on the **pressure boundary** to make the connection.
5. Connect the **Node** to the **Flow Demand**
    a) Click on the **Node** to start another **Pipe** and move the cursor down and click on a grid line to create a vertex, then to the right and click on the **Flow Demand** to make the connection.

**NOTE:** The **Devices** and **Pipes** can be moved around by selecting them and dragging them to the desired position.

**NOTE:** Vertices can be added by holding down the **SHIFT** key and clicking on a **Pipe**. The vertex can be moved to the desired location.

# Enter Device Design Data

**Design Data** can be entered individually or multiple **Devices** can be selected and **Group Edited** to quickly enter redundant data.

The following steps will guide you through entering the **Design Data** individually and using the **Group Edit** feature.

# Enter the Tank Design Data

The **Tank Design Data** includes key parameters and the **Fluid Zone** to use if there are two or more pipes are connected with different **Fluid Zones**.

## Design the Supply Tank

1. Select the **Tank** on the left.
2. In the **Property Grid**:
    a) In the **Name** field, enter **Supply Tank**.
    b) In the **Elevation** field, enter **20 ft (6 m)**.
    c) In the **Surface Pressure** field, enter **0 psig (0 kPa)**.
    d) In the **Liquid Level** field, enter **10 ft (3 m)**.
    e) In the **Fluid Zone** field, leave the value set to **<Undefined>** (this only needs to be specified if the connected **Pipes** have different **Fluid Zones**).
    f) Click in the **Penetrations** field to activate the ... button to open the **Tank Penetrations** dialog box.
        i. leave the **Height** set at the default value of **0 ft (0 m)**.
        ii. Click OK to close the dialog box.
3. Notice that the **Supply Tank** text is now black, indicating that it is fully designed.

# Enter the Sizing Pump Design Data

The **Sizing Pump** is used to calculate the **Total Head** of a **Pump** based on the hydraulic characteristics of the entire piping system.

### Design the Sizing Pump

1. Select the **Sizing Pump**.
2. In the **Property Grid**:
    a) In the **Name** field, enter **Transfer Pump**.
    b) In the **Suction Elevation** field, enter **0 ft (0 m)**.
    c) In the **Discharge Elevation** field, enter **0 ft (0 m)**.
    d) In the **Flow Rate** field, enter **800 gpm (182 $m^3$/h)**

# Enter the Curve dP Device Design Data

The **Curve dP Device** is used to model equipment that has a **Pressure Drop** that varies with the **Flow Rate** through the equipment, which is the case for many **Devices** such as heat exchangers, filters, screens, etc. The **Curve dP Device** has a **Curve Data Estimator** to generate a second order curve based on a single user-entered **Flow Rate** and **Pressure Drop** data point.

### Design the Curve dP Device

1. Select the **Curve dP Device**.
2. In the **Property Grid**:
    a) In the **Name** field, enter **Duplex Strainer**.
    b) In the **Inlet Elevation** field, enter **75 ft (23 m)**.
    c) In the **Outlet Elevation** field, enter **75 ft (23 m)**.
    d) Click in the **Curve** field to activate the [...] button. Click on [...] to open the **Curve Data** dialog box.
        i. The default units displayed come from the **Document Units** but can be changed if desired.
        ii. In the **Description** field, enter **Clean Strainer**.
        iii. Click on the [Estimate Curve Data...] to open the **Curve Data Estimator** dialog box.
            1. In the **Static Pressure Drop** field, enter **0 psi (0 kPa)**.
            2. In the **Flow Rate** field, enter **800 gpm (182 $m^3$/h)**.
            3. In the **Pressure Drop** field, enter **10 psi (69 kPa)**.
            4. For the **Maximum Flow Rate**, enter **1600 gpm (363.4 $m^3$/h)**.
            5. Click [OK] to close the **Curve Data Estimator** dialog box. Notice that the data table if filled out with values generated using a 2nd order curve.
        iv. Click [OK] to close the **Curve Data** dialog box.

# Enter the Pressure Boundary Device Design Data

The **Pressure Boundary** is used to model equipment that has a given pressure at the boundary of a system.

#### Design the Pressure Boundary Device

1. Select the **Pressure Boundary Device**
2. In the **Property Grid**:
    a) In the **Name** field, enter **Pressure Boundary**
    b) In the **Elevation** field, enter **85 ft (26m).**
    c) In the **Pressure** Field enter **75 psig (517 kpa).**

# Enter the Flow Demand Device Design Data

The **Flow Demand** is used to model equipment that has a given flow requirement at the boundary of a system. The flow requirement can be either flow into the system or flow out of the system that you are modeling.

**Design the Flow Demand Device**

1. Select the **Flow Demand Device**
2. In the **Property Grid**:
    a) In the **Name** field, enter **Flow Demand**
    b) In the **Elevation** field, enter **45 ft (14m).**
    c) In the **Flow Rate** field, enter **100 gpm (23 m3/h).**
    d) In the **Flow Type** field, select **Flow Out**.

# Enter the Node Design Data

**Nodes** provide a location for connecting multiple **Pipelines** together and for providing a point in the system where the **Pressure** is calculated, making them useful for identifying the location of pressure gages in the piping system.

**Design the Nodes**

1. Select the **Node** at the outlet of the **Duplex Strainer (Curve dP Device)**
2. In the **Property Grid**:
    a) In the **Name** field, enter **PG-101**.
    b) In the **Elevation** field, enter **0 ft (0 m)**.

# Enter the Pipe Design Data

The **Pipe Design Data** includes key pipe and fluid properties and the ability to add valves and fittings that are installed in the **Pipe**. There are also options to **Close** the **Pipe** to simulate a closed valve.

A **Pipe Size Calculator** is available to size the **Pipe** based on a **Sizing Criteria** that is selected in the **Pipe Specification**.

## Design the Pump Suction Pipe

1. Select the **Pipe** between the **Supply Tank** and **Transfer Pump**
2. In the **Property Grid**:
    a) In the **Name** field, enter **Suction**
    b) In the **Fluid Zone** field, select **Water at 60 F and 75 psig(Water at 15 C and 517 kPa)** from the drop down list
    c) In the **Specification** field, select **Steel Sched 40** from the drop down list
    d) In the **Size** field, click the ... button to open the **Pipe Size Calculator**
        i. In the **Design Flow Rate** field, enter **800 gpm (182 $m^3$/h)**
        ii. The **Calculated Size** that will be closest to the **Sizing Criteria** is displayed **(6 in) (or 150 mm)**.
        iii. From the drop down list, choose **8 in(200 mm)** instead since this is the pump suction piping (and we want to maximize **NPSHa**).
        iv. Click OK to close the **Pipe Size Calculator.**
    e) In the **Length** field, enter **45 ft (14 m)**
    f) In the **K (Valves & Fittings)** field, click the ... button to open the **Valve and Fittings** dialog box
        i. Expand the **Fitting** section:

1. Select **Entrance - Sharp Edged** and click Add to place it on the **Installed** list
2. Select **Elbow - Standard 90°**, enter **3** for Count, then click Add to place them on the **Installed** list

ii. Expand the **Valve** section and add 2 Gate valves individually:

1. Select **Gate - Knife**, leave the **Count** as **1**, then click Add to place it on the **Installed** list
2. Select **Gate - Knife**, leave the **Count** as **1**, then click Add to place it on the **Installed** list

iii. Click OK to close the **Valve and Fittings** dialog box.

iv. Notice the calculated **K (Valves & Fittings)** which is the sum of the **Resistance Coefficients** for all the installed **Valves and Fittings**

v. Notice that the **Suction Pipe** shows the individual gate valves, but there is only one symbol representing all 3 elbows. The symbols can be moved along the **Pipe** for graphical purposes, or using the **Device View Options** the **Valves & Fittings** can be hidden.

## Group Edit Common Fields on the Remaining Pipelines

3. Select all the **Devices** by **Left Clicking** above and to the right of the **North Tank**, holding down the **Left Click** and dragging the cursor to the bottom left of the **FLO-Sheet** (this draws a selection box around all of the **Devices**), then releasing the **Left Click**. All the **Devices** will be selected and the **Property Grid** will display an expandable list of the **Device** types.
4. In the **Property Grid**, expand the **Pipe Design** section
5. In the **Fluid Zone** field, select **Water at 60 F and 75 psig(Water at 15 C and 517 kPa)** and then click outside the field.
    a) Notice that when all selected devices have the same value, it is displayed in the field.
6. In the **Specification** field, select **Steel Sched 40**
    a) Notice that the field now says **Steel Sched 40**. This allows the **Pipe Size** to be **Group Edited** as well. We'll come back to this below.
7. **Group Edit** the **Names** of the remaining **Pipes**:
    a) With all the **Devices** selected, de-select the **Suction Pipe** by holding down the **CTRL** key then clicking on the **Suction Pipe.**
    b) In the **Name** field, enter **10-TW-1**, then **Enter**.
    c) Notice that all the selected **Pipes** take the new **Name** format, but the last number is incremented by 1 to create a unique name for each **Pipe**.
8. **Group Edit** common **Pipe Sizes**
    a) Click on an empty space on the **FLO-Sheet** to de-select all the **Devices**
    b) Hold down the **CTRL** key and select **Pipes**: **10-TW-1, 2, 3 and 4**
    c) In the **Size** field of the **Pipe Design** section of the **Property Grid**, select **6 in (150 mm)**, then **Enter**
    d) Notice that these four **Pipes** now show that they are **6 inch (150 mm)**

### Enter the Remaining Design Data for the Pipes

1. Using the table below, enter the **Lengths** and **Valves & Fittings** for the remaining **Pipes**.

| Pipe Name | Size (inches) | Length (ft) | Valves & Fittings |
|---|---|---|---|
| **10-TW-1** | 6<br>(150 mm) | 100<br>(30 m) | (1) Check Valve - Swing Check - Angled<br>(1) Valve - Gate - Knife |
| **10-TW-2** | 6<br>(150 mm) | 200<br>(60 m) | (1) Valve - Gate - Knife<br>(4) Fitting - Elbow - Long radius, r/d 1.5 (90°) |
| **10-TW-3** | 6<br>(150 mm) | 500<br>(152 m) | (1) Valve - Gate - Knife<br>(1) Fitting - Tee - Flow Thru Branch |
| **10-TW-4** | 6<br>(150 mm) | 500<br>(152 m) | (1) Valve - Gate - Knife<br>(10) Fitting - Elbow - Long radius, r/d 1.5 (90°) |

# Calculate the Model

Now that all the **Devices** in the model have the required **Design Data**, the system can be calculated and the results evaluated for each **Device** and the entire system. The results can be evaluated using the information selected for display on the **FLO-Sheet** and in the **List View Window**, but the first results that should be evaluated are any **Messages** that are generated in the **Messages Window**.

Calculate the model by either:

1. Click the **Calculate** button  on the **View Toolbar,** or
2. Select **Calculate** from the **System Menu**, or
3. Click **F5**.

To clear the results, click  again, **F5**, or select **Calculate** from the **System Menu** again.

# Evaluate Messages

There are five (5) levels of messages based on the calculated results. Level 0 messages are generated if results cannot be calculated based on how the system is designed. Level 1 messages mean the results are invalid, Level 2 indicated increased uncertainty in the results, Level 3 messages are generated if user-entered limits are exceeded, and Level 4 are informational messages about the status of devices.

To view the **Messages**:

1. If the **Message Window** is not displayed, from the **View Menu**, select **Messages** from the **Windows** drop down list.
2. The **Message Window** shows what **Device(s)** is causing the message, what the message is, and a link to a page in the **Help File** that explains the message and what can be done to resolve the problem that is creating the message.

There should only be one message generated at this point in the tutorial. This should be a **Level 4** message describing the **Number of Iterations** and the **% Deviation** of each iteration. If you have any other messages, review the previous sections of the tutorial to determine what was entered incorrectly.

# Evaluate Calculated Results

For each element in the piping system you can find both design information and calculated results on:

- the **FLO-Sheet**,
- the **Property Grid Results** section for each **Device**, or
- the **List View Window**

### Property grid

- This is the only place where design information can be added or edited
- When you point to something on the FLO-Sheet the Property Grid automatically shows the information about the selected item

### List View

- You can see the results of like elements and compare them together
- The list can be sorted by any heading
- Columns can be re-arranged or hidden

### FLO-Sheet

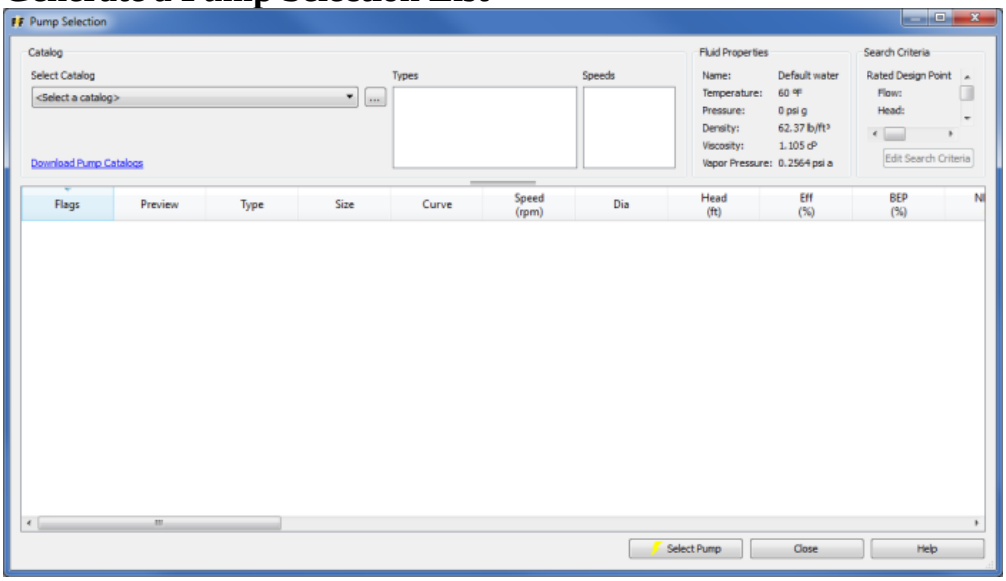
- This is to provide important information
- Best way to see the results in context of the entire system
- Information for individual items can be hidden or displayed based on the objective of the calculation
- Excellent presentation look with objective on big picture.

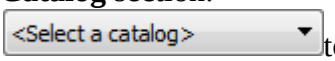
# Select a Centrifugal Pump

With the piping system model in the **Calculate Mode**, the **Total Head** and **NPSHa** results for the **Sizing Pump** can be used to select a **Centrifugal Pump** from over 95 pump manufacturer catalogs developed by Engineered Software, Inc. and supported by the software.

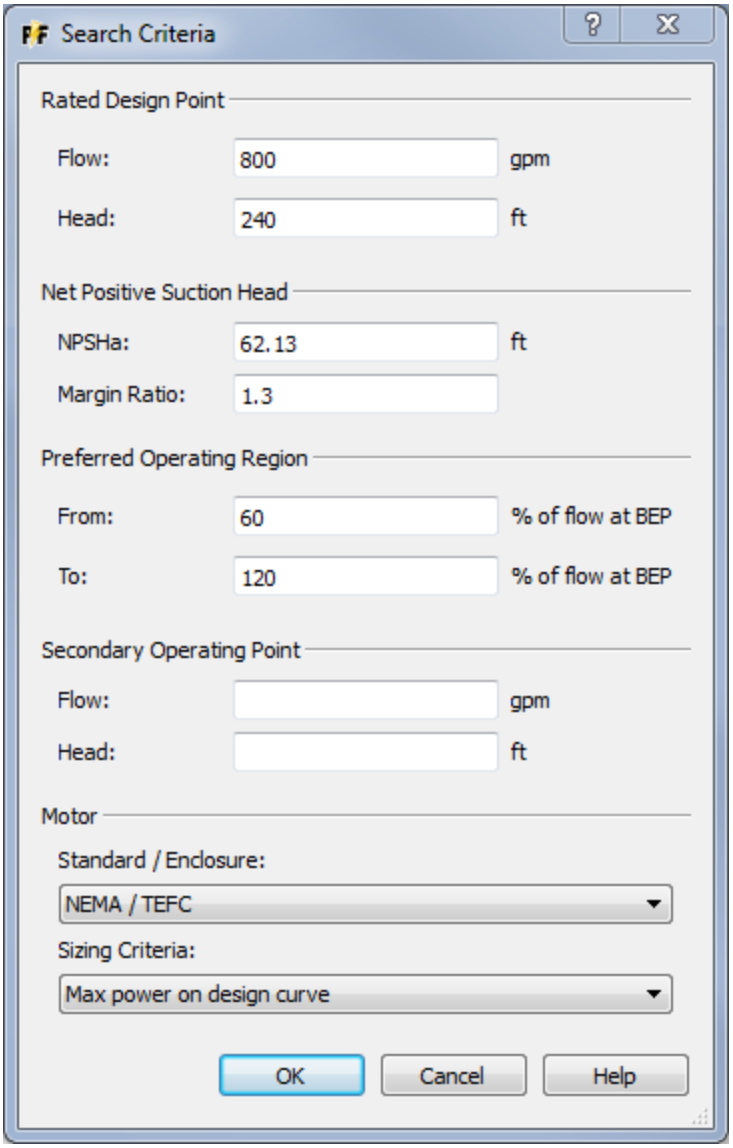
The software will step you through the selection process, automatically searching through the selected catalog. Once the pump is selected, the pump performance data is automatically imported into the model and the **Sizing Pump** is converted into a **Centrifugal Pump**.

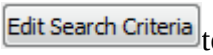
## Generate a Pump Selection List

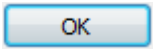



1. Verify the model is in **Calculate Mode** with the calculated **Total Head** displayed as 260.8 ft (79.5 m).
2. Right Click on the **Transfer Pump** and choose **Select Catalog Pump** from the right click menu to open the **Pump Selection** dialog box.
    a) The **Fluid Properties** section shows the key fluid properties from the **Fluid Zone** of the pump's inlet pipe.
    b) The **Search Criteria** section displays the **Flow Rate**, **Total Head**, **NPSHa**, among other criteria. This criteria can be edited once a catalog is selected.
3. In the **Select Catalog** section:
    a) Click 

to expand the list of pump catalogs that are installed on your computer.
        i. To download additional catalogs, click on the **Download Pump Catalogs** link to go the Engineered Software website.
    b) Choose **Sample Catalog.60** from the list.
        i. Notice that the **Types** and **Speeds** boxes are now populated with those available in the catalog.
        ii. Also notice that the **Edit Search Criteria** box is activated.




4. Click on the **Edit Search Criteria** button 

to open the **Search Criteria** dialog box.
    a) decrease the **Total Head** value to **240 ft** (73m) to add a design margin.
    b) Enter **62.13 ft** (19m) for NPSHa
    c) Enter a **NPSH Margin Ratio** of **1.3**
    d) Enter a **Preferred Operating Range** from **60** to **120** percent of flow at BEP.
    e) A **Secondary Operating Point** can be entered, along with a motor type and motor **Sizing Criteria**. Leave these as they are for now.
    f) Click 

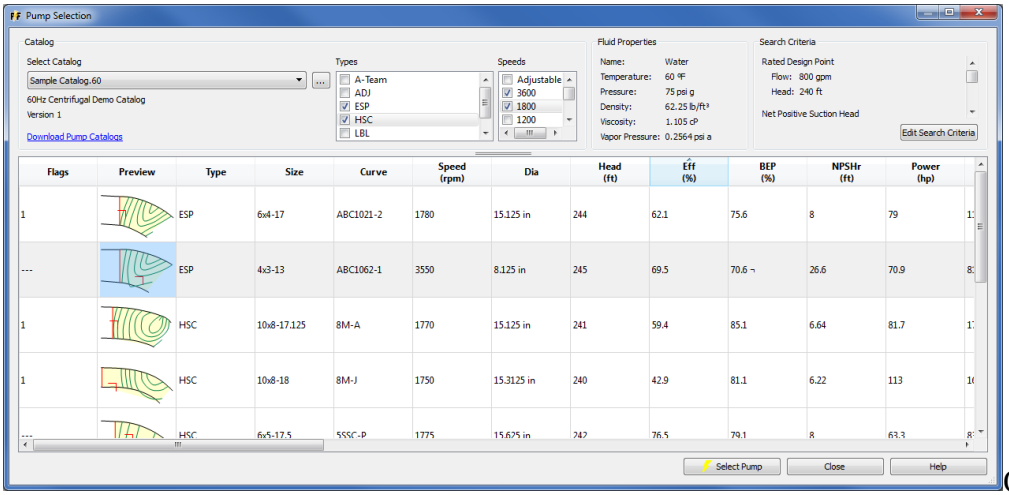
to accept the new **Search Criteria**.
5. In the **Types** section, check the box beside **ESP** and **HSC.**
6. In the **Speeds** section, check the box beside **3600** and **1800.**
7. A list of pumps that meet the **Search Criteria** is displayed, along with a thumbnail of the pump curve with the design point marked.
    a) Click on the **Eff (%)** column heading to re-order the list by pump efficiency in ascending order. Click it again to re-order the list in descending order.
8. Double click on the **ESP 4x3-13** row to open its **Pump Curve**.
    a) The pump **Speed** (3550 rpm) and recommended impeller **Diameter** (**8.125"**) (206 mm) to achieve the design point are displayed. Note that the value of **Total Head** with this diameter is **245 ft (76m)**, which is greater than the search criteria of **240 ft (73m).**

i. Click the down triangle beside the impeller diameter to reduce the size to **8"**(200 mm). Notice that the pump curve drops and the value of **Total Head** at this diameter is only **229 ft (70m)**, which is less than the required **Total Head**.
ii. Click the up triangle to increase the impeller diameter back to **8.125"** (206mm).

b) New **Flow** or **Head** values can be entered and the results displayed in the **Data Point** section. This does not change the design point used in the **Search Criteria**.

9. Click OK to close the dialog box.

### Select a Pump and Transfer Pump Data to the Model




1. Click on the **ESP 4x3-13** line to highlight the row.
2. Click Select Pump to return the pump data to the model and change the **Sizing Pump** into a **Centrifugal Pump**.
3. The **POR** and **NPSH Margin Ratio** are not brought in from the **Search Criteria**, so these have to be re-entered.
   a) Enter **60** in **POR From** field**.**
   b) Enter **120** in **POR To** field**.**
   c) Enter **1.3** in the **NPSH Margin Ratio** field.

### Evaluate Calculated Results

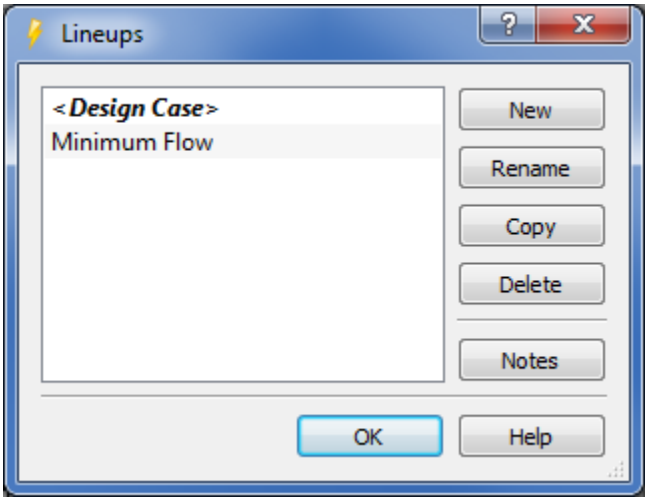
1. Notice that the flow rate through the **Transfer Pump** is now around **617 gpm** (140 m3/h), which is less than the design flow rate of **1600 gpm (363.4 m3/h)**.

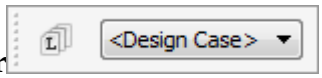
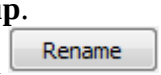
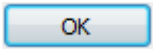
# Use Lineups to Evaluate Different Operating Conditions

Once the **Design Data** is entered for each **Device**, different operating conditions can be evaluated using the **Lineups** feature. The **Operation** and **Setting** of **Devices** can be changed, **Pipes** can be **closed**, and **Fluid Zones** can be modified.

The initial **Lineup** is called the **<Design Case>**. An unlimited number of **Lineups** can be created and **Lineups** can be re-named, copied, and deleted. **Lineup Settings** can be changed while in the **Calculate Mode**.

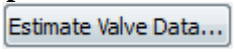
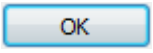
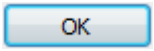
## Create a New Lineup




1. Click the **Edit Lineup** button  in the **Lineup Toolbar**  to open the **Lineups** dialog box.
2. Click  to add a **Lineup**.
3. Select the new **Lineup** and click  .
4. Rename the **Lineup** "**Minimum Flow**".
5. Click  to close the **Lineups** dialog box.

## Change Lineup Settings

Place the system in **Calculate Mode**, then:

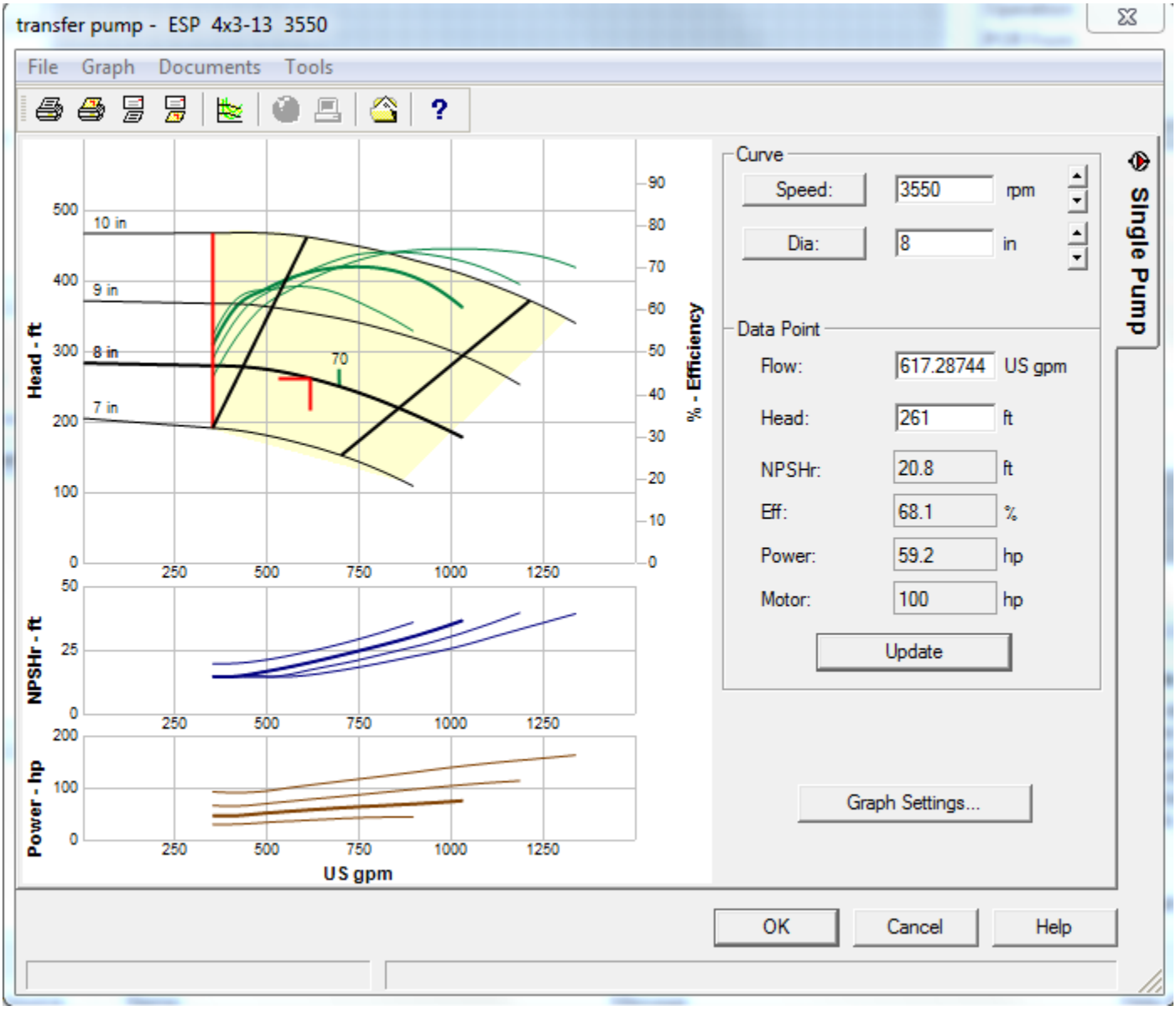
1. In the **Lineup Toolbar**, select **Minimum Flow** from the drop down menu.
2. Select the **Supply Tank** on the **FLO-Sheet** and change the **Liquid Level** to **8 ft (2.4 m)**.
3. Select the **Duplex Strainer**, then:
    a) In the **Property Grid**, click in the **Curve** field to activate the  button. Click on  to open the **Curve Data** dialog box.
    b) In the **Description** field, enter **Plugged Strainer**.
    c) Click on the  to open the **Curve Data Estimator** dialog box.
        i. In the **Static Pressure Drop** field, enter **0 psi**.
        ii. In the **Flow Rate** field, enter **400 gpm (91 $m^3$/h)**.
        iii. In the **Pressure Drop** field, enter **25 psi (172 kPa)**.
        iv. For the **Maximum Flow Rate**, enter **800 gpm (181.7 $m^3$/h)**.
        v. Click  to close the **Curve Data Estimator** dialog box.
    d) Click  to close the **Curve Data** dialog box.
4. Evaluate **Messages**
    a) With the system in **Calculate Mode**, review the **Messages** created with this operating scenario using the **Messages Window.**
        i. Pump running outside of preferred operating region.
    b) Click the **Help ID** link in the **Messages Window** to open a description of the message and what is causing it.

### Reset Lineup Settings

1. Select the **North Tank** on the **FLO-Sheet**
2. Change the **Surface Pressure** to **25 psig (172 kPa)** and the **Liquid Level** to **10 ft (3 m).**
3. In the **Lineups Values Window**, for the **Minimum Flow Lineup**, click on the **Reset** button .
4. Notice that the **Surface Pressure** and **Liquid Level** are returned to their **<Design Case>** Lineup values.

# Use Graphs to Visualize Data and Performance

There are several graphs that can be opened to visualize the hydraulic performance of various **Devices**, including **Centrifugal Pumps**, **Pipelines**, **Flow Meters**, and **Curve dP Devices**. The **System Resistance Curve** can also be graphed.

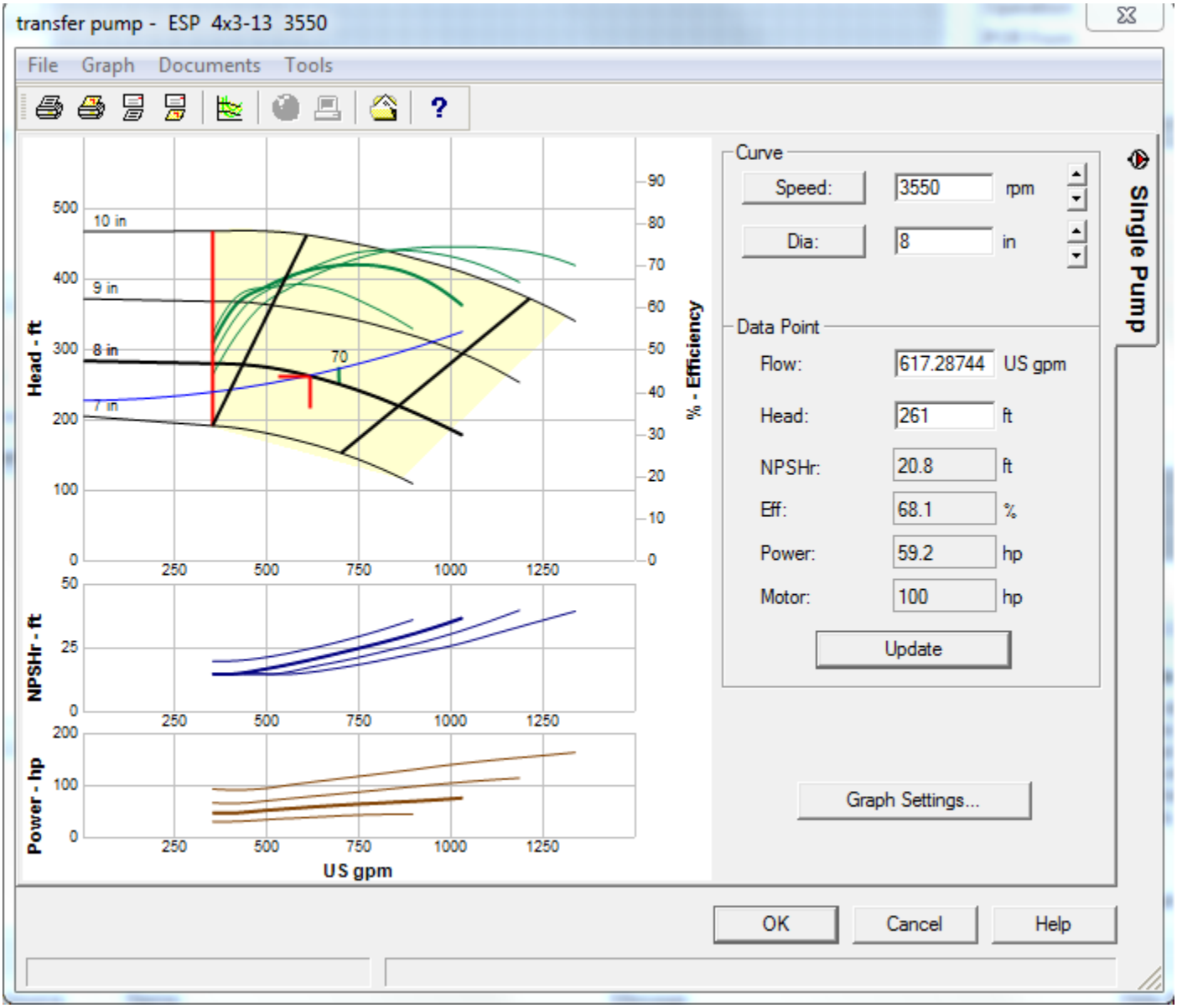



### Pump Graph

1. Right click on the **Transfer Pump** and select **Graph...** from the right click menu to open the **Pump Graph**.
2. Click on the **Graph Settings...** button to open the **Graph Display** dialog box.

1. In the **Pump Curves** section, check **Catalog**, **Envelop**, and **Design Curve**.
2. In the **Optional Curves** section, check **Efficiency** and **Power**, and either **ISO** or **Line** under each.
3. In the **Graph Elements** section, check **Grid Lines**, **Design Point**, **Design Guides**, and **Selection Window**.
4. The color of each line can be changed.
5. Click OK to close the **Graph Display** dialog box.

3. Click within the allowable operating range for the given impeller diameter to view the associated values for that **Data Point**.
4. Increase and decrease the Speed and Diameter to see the effect on the pump curve.
5. Click Cancel to close the pump graph without applying the changes to the pump performance.

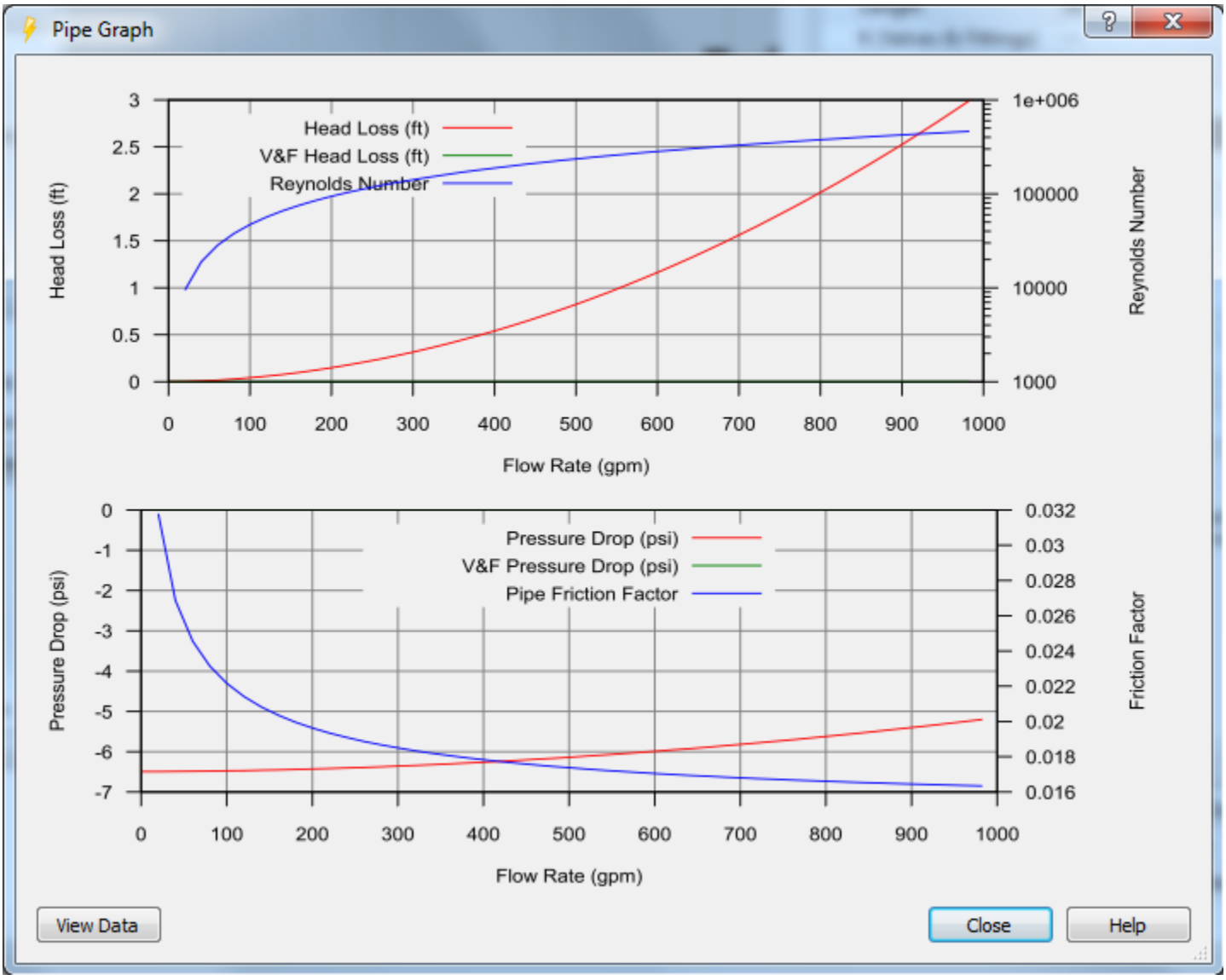



## Pump Graph with System Resistance Curve

The Pump Graph can also be printed with the System Resistance Curve.

1. Right click on the **Transfer Pump** and select **Graph resistance curve...** from the right click menu to open the **Pump Graph**.

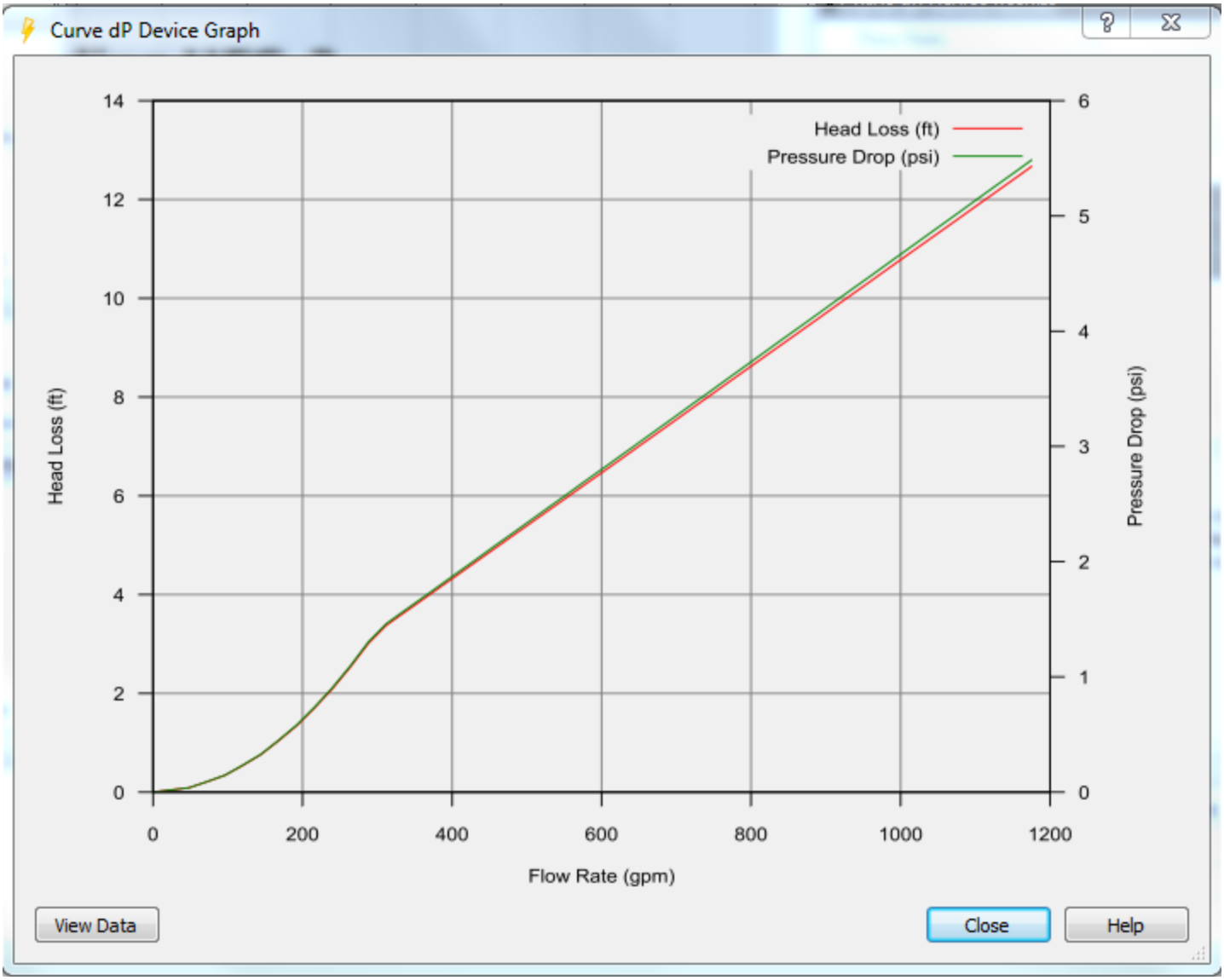
2. Click OK to close the pump graph.




## Pipeline Graph

The system must be in **Calculate Mode** to view the **Pipeline Graph**.

1. Right click on any pipeline and select **Graph** from the right click menu to open the **Pipeline Graph**.
2. Click View Data to open a text file containing the data that was generated to create the graph. This file can be saved if desired.
3. Click to close the data text file.
4. Click Close to close the **Pipeline Graph**.




**Curve dP Device Graph**

The system does not have to be in **Calculate Mode** to view the **Curve dP Device Graph**.

1. Right click on the **Duplex Strainer** and select **Graph** from the right click menu.
2. Click View Data to open a text file containing the data that was generated to create the graph. This file can be saved if desired.
3. Click to close the data text file.
4. Click Close to close the **Curve dP Device Graph**.

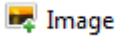
# Enhance the Presentation Value of the FLO-Sheet

There are various features that allow you to customize the look of the **FLO-Sheet** to help you highlight key information or to make your model drawing look similar to a Process Flow Diagram or Piping & Instrumentation Diagram (P&ID).

1. **Presentation Mode** can be toggled on to remove the grid and show the system on a white background.
2. **Device Symbols** can be changed and re-sized.

3. The **Text** for individual **Devices** can be shown or hidden and the font, style, size and orientation can be changed.
4. **Floating Text** can be placed anywhere on the drawing, allowing you to freely annotate various areas and objects in your system.
5. **Images** can be placed on the **FLO-Sheet**, moved and re-sized.
6. **Rectangles** can be drawn around objects and sections of the system to highlight particular pieces of equipment, operation areas, and landmarks.
7. **Notes** and **Links** can be added to each **Device** to include additional information.

# Add an Image to the FLO-Sheet

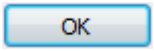
1. Click the **Image** button 

in the **Toolbox** and click on the upper middle area of the **FLO-Sheet**.
2. Navigate to the **Sample Systems** folder under **My Engineering Files>Projects**.
3. Select the **OlyPumps Logo.bmp** file and click **Open**. The image is placed on the **FLO-Sheet**.
4. The image can be moved to the desired location and re-sized.

**NOTE:** You can re-size the image by clicking and dragging the green squares on the corners and sides of the images. The corners keep the ratio of length to height locked in, while the side and top and bottom squares will stretch the image.

# Add Free Floating Text to the FLO-Sheet

1. Select the **Text** button 

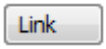
2. Click at the top center of the drawing to place a Floating Text box on the FLO-Sheet.
3. Enter **Building HGF-2** in the **Property Grid** .
4. Click  beside **Font** below the text entry box of the **Property Grid** to open the **Select a Font** dialog box.
    a) Select **Arial** as the **Font**.
    b) Select **Bold** for **Font Style.**
    c) Select **16** as the **Size**
    d) Check the **Underline** box.
5. Click 

to close the **Select a Font** dialog box.

# Add Notes and Links to Devices

Notes and Links can be added to Devices to document additional information that is not included in the Design Data fields.

### Add a Note and Link to the Duplex Strainer

1. Select the **Duplex Strainer**.
2. In the **Property Grid**, scroll down to the **Notes** section.
3. Click the **Notes** field to open the **Editor** dialog box.
    a) Click the **Edit** tab.
    b) Enter the following text: **"Manufacturer recommends cleaning strainer basket when pressure drop reaches 15 psi.**"
4. Select the word Manufacturer
5. Click 

to open the **Hyperlink** dialog box.

a) Type in the following URL: **www.eng-software.com**
b) Click OK to close the **Hyperlink** dialog box.

6. Click the **View** tab to view the **Note** and **Link**.
   a) Click the hyperlink that was created to open the web page (if you have internet access).
7. Click OK to close the **Editor** dialog box.
8. There is now an asterisk (*****) to the left of the first line of text for the **Duplex Strainer** indicating that the **Device** has a **Note**.

# Adjust Text Settings for Devices

The **Device View Options** are used to select what **Design Data** and **Calculated Results** to display on the **FLO-Sheet** for each **Device** when the text is shown.

With the **Text Settings**, unnecessary or repetitive text can be hidden individually for **Devices** or using the **Group Edit** feature, leaving just the text that the user wants to highlight. In addition, the text font, style, size, and orientation can be changed to emphasize key information.

### Hide Unnecessary or Repetitive Text

1. Select pipe **10-TW-1** at the discharge of the **Transfer Pump**.
   a) Expand the **Text Setting** section of the **Property Grid**, then un-check the **Show Text** box to hide the text for this individual pipe.

### Change Text Font, Style and Size

1. Select the **Transfer Pump**
2. In the **Text Settings** section of the **Property Grid** for the pump, click inside the Font field to activate the ... button. Click ... to open the **Select Font** dialog box.
   a) Scroll down the **Font** list and select **Arial.**
   b) Select **Bold Italic** as the **Font Style**.
   c) Select **10 pt** as the **Font Size**.
   d) Click OK to close the **Select Font** dialog box.

# Change the Device Symbols and Sizes

Once the piping system is drawn, the **Device Symbols** and **Sizes** can be adjusted if desired.

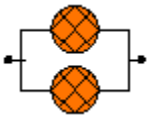
### Change Device Symbols

**Device Symbols** can be changed using the **Symbol Settings** section in the **Property Grid** for each **Device** or by right clicking on the **Device** to open the right click menu.

1.  Change the **Transfer Pump** Symbol
   a) Right click on the **Transfer Pump** and select **Change Symbol** to open the **Symbol** dialog box.

b) Select the In-Line Centrifugal Pump symbol and click OK.

2.  Change the **North Tank** symbol
   a) Right click on the **North Tank** and select **Change Symbol**.
   b) Select the Pressurized Tank symbol and click OK.
   c) Repeat for the **South Tank**.

3.  Change the **Curve dP Device** symbol

1.
   a) Right click on the **Curve dP Device** and select **Change Symbol**
   b) Select the Duplex Strainer symbol and click OK.

### Change Device Sizes

**Device Sizes** can be adjusted individually using the **Symbol Settings** section in the **Property Grid** for each **Device** or the sizes of multiple **Devices** can be changed with the **Group Edit** feature.

1. Select the **Flow demand**.
2. In the **Property Grid**, click on the triangle beside **Symbol Settings** to expand this section.
3. Click in the **Symbol Size** field to open the drop down menu and select **75%**.
4. Click on the **FLO-Sheet** or **Enter** to accept the change.

# View the FLO-Sheet in Presentation Mode

1. Click the **Presentation Mode** button  on the **View Toolbar**. The **FLO-Sheet** grid lines are removed and a white background replaces the gray background.
2. Click **Presentation Mode** again to display the **FLO-Sheet** with a gray background with grid lines.