IDEAS Simulation Software

IDEAS GOLD TUTORIAL

ENGINEERED SUCCESS

Contact: ANDRITZ Inc.
Division: Automation & Digitalization
13560 Morris Road, Suite 1250
Alpharetta, GA 30004, US
Phone: +1 770 640 2500
Fax: +1 770 640 2557
E-mail: simulation.support@andritz.com
andritz.com

TABLE OF CONTENTS

TABLE OF FIGURES

SECTION 1. INTRODUCTION

The IDEAS™* Gold Tutorial has been designed to give the user an overview of the key concepts of the IDEAS simulation software for dynamic modeling. Step-by-step exploration of these concepts allows the user to build confidence and capability in their knowledge of the IDEAS Gold (fully dynamic) objects. Formal IDEAS software training provides guided instruction and further exploration of the IDEAS modeling environment to take the new user to a higher level of expertise and capability. The various levels of the basic IDEAS application are named and described as:

- IDEAS Bronze: Steady-state modeling (also called 'macro' level modeling)
- IDEAS Silver: Partially dynamic modeling (includes tank residence time and controller dynamics)
- IDEAS Gold: Fully dynamic modeling (includes full pressure-flow dynamics)

1.1 CREATING A FOLDER

On the hard drive of your PC, create a new folder (e.g., IDEAS Tutorial) that will be used whenever any file is saved throughout this tutorial.

1.2 THE PROBLEM

The IDEAS Simulation Software is used in a wide variety of industries including general chemical, oil and gas, pulp and paper, oil sands, mining and minerals, and pharmaceuticals. In this tutorial, the user will model a simple dynamic process containing a cylindrical tank, a centrifugal pump, and a control valve. Such a system would exist in any industry.

The key points of this tutorial are:

- Selecting an appropriate pump and a valve to get the desired flow out of the tank
- Designing a control system to avoid damage to the pump when the tank runs dry
- Tuning the controllers
- Changing the system to carry components other than water
- Examining Excess Properties

Even though this is a simple process, there are many details to be determined before equipment purchase can be done. One of your experienced colleagues advises you to use IDEAS to model this process and get the required answers. This tutorial is a step-by-step guide for the modeling process.

* Trademark of the ANDRITZ GROUP.

SECTION 2. STARTING IDEAS

Section Concepts:

- Launching IDEAS
- Opening Libraries

2.1 STARTING THE IDEAS APPLICATION

First, locate the IDEAS application using the Windows Start Menu. From the Windows Start Menu, choose All Programs → Andritz → IDEAS.exe (this path may differ slightly depending on the installation). Launch the application by clicking on the IDEAS icon. Alternatively, the user can navigate to C:\Andritz\IDEAS 2024 to launch the application using "IDEAS.exe" shortcut.

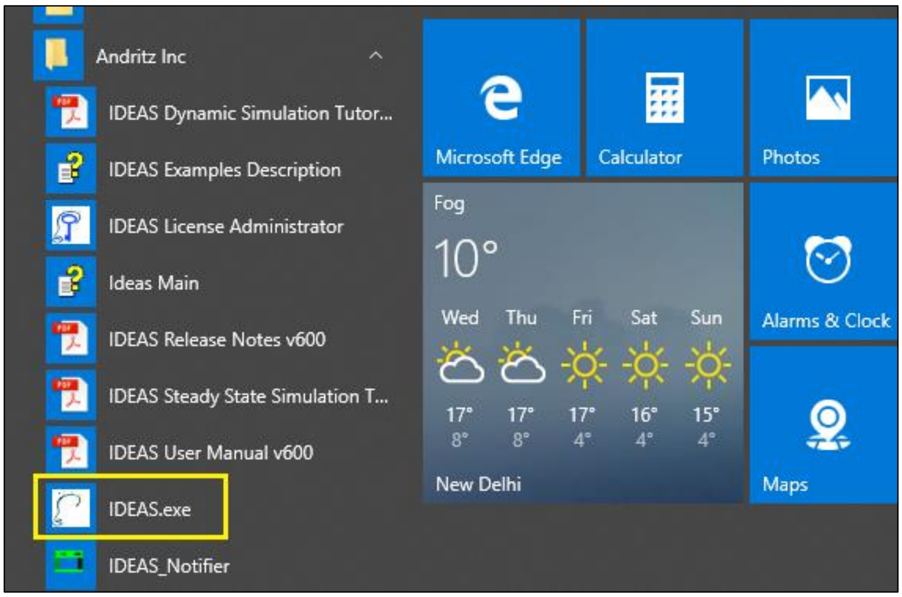


Figure 1: IDEAS XXX Location

2.2 OPENING A WORKSHEET

Once the IDEAS window opens, go to the **File** menu and select **New Model** (Ctrl + N). A blank worksheet named Model-1 appears. Maximize the worksheet to fill the entire screen (see Figure 2 and Figure 3) if desired.

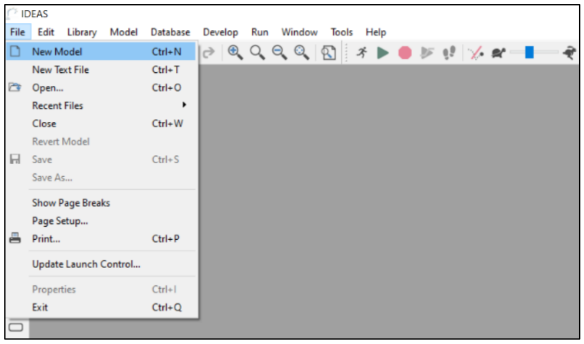


Figure 2: Opening Model-1 Worksheet

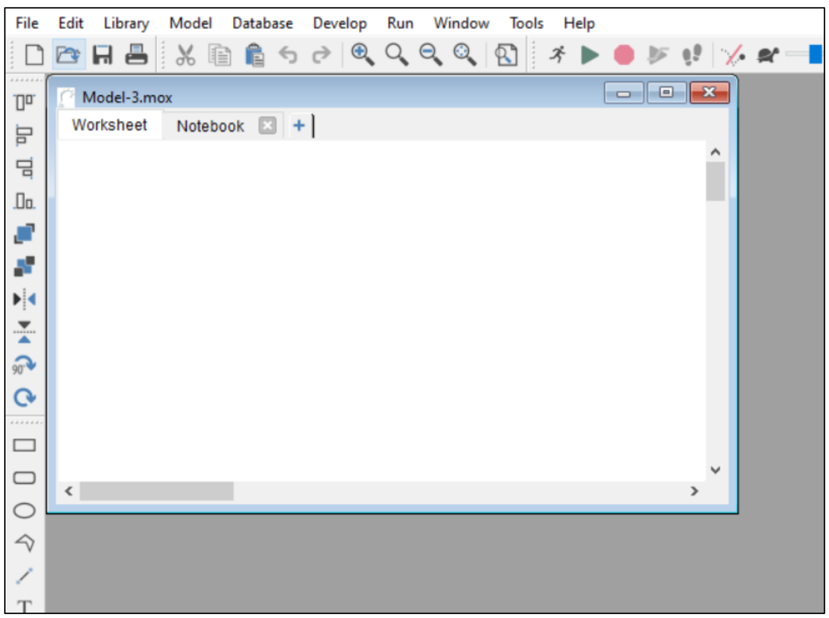


Figure 3: Model-1, Blank Work Sheet

2.3 SAVING THE MODEL

Go to the **File** menu again and Select '**Save Model as**...' When the **Save File As** dialog box appears, navigate to the folder that you had created at the beginning of this tutorial, viz. IDEAS Tutorial, click on the box labeled **Save In:** with the dropdown option "▼." Type *Standard Worksheet* in File name and click on the **Save** button. If a dialog box opens asking "Replace existing Standard Worksheet?" - click on the **Yes** button.

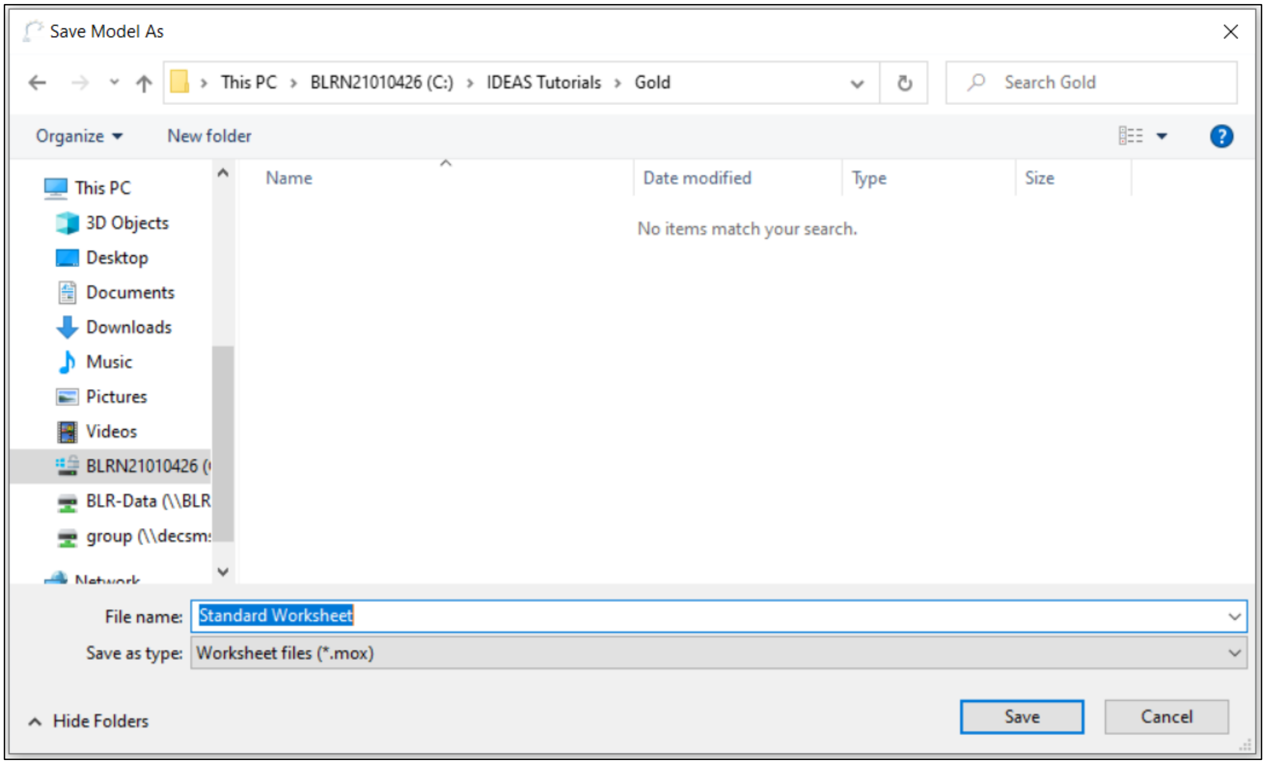


Figure 4: Save File As Dialog Box

Note: Make sure that you save the model at regular intervals throughout this tutorial. You may not be reminded to save your model until the end of the tutorial.

2.4 OPENING LIBRARIES

On the IDEAS menu bar, go to **Library** and **Open Library** (Ctrl + L) as shown below.

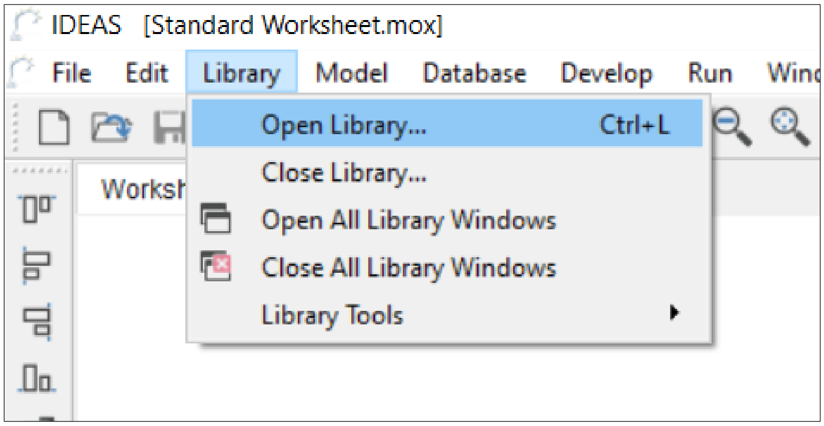


Figure 5: Opening Library Window

When the dialog box opens, navigate to the folder containing the libraries. This folder will display all the available libraries (as shown in Figure 6).

Depending on the IDEAS products you have purchased, your list of libraries may vary from the list shown in Figure 6.

To open a particular library, highlight the library name and click on the **Open** button OR double-click on the selected library name.

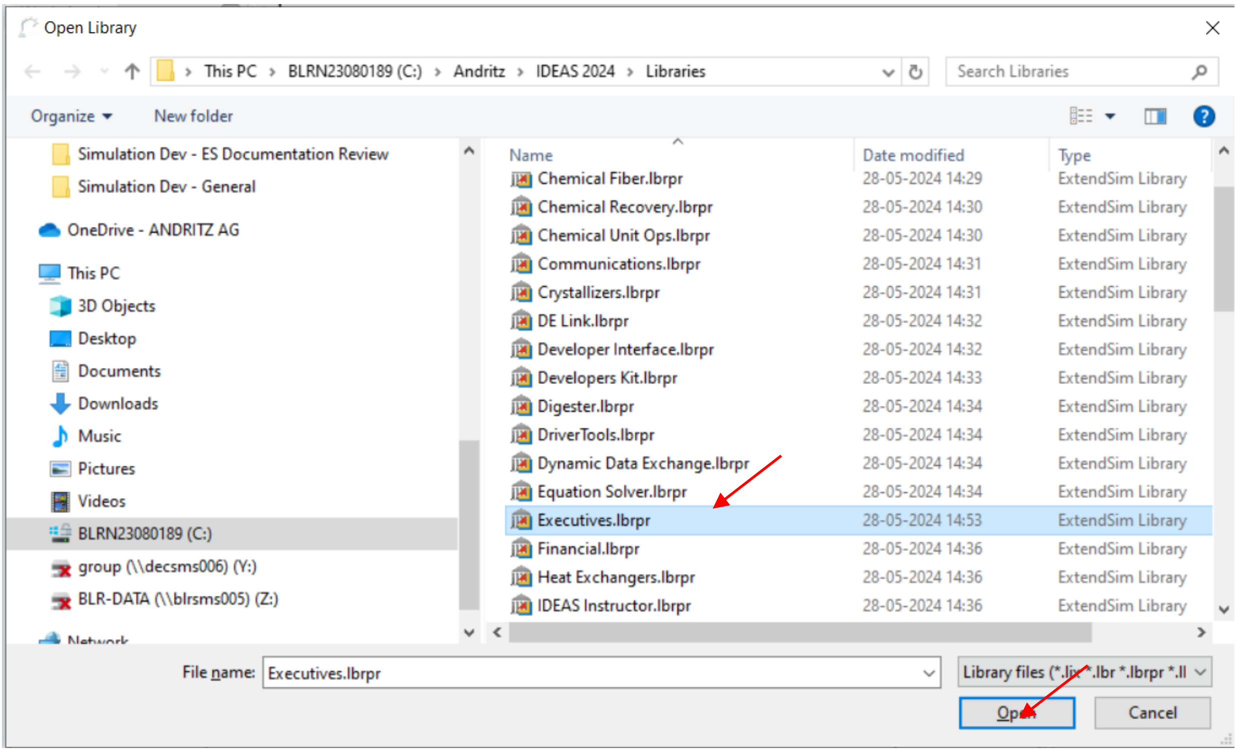


Figure 6: Location of Libraries

2.5 LISTING OF LIBRARIES

Open the following libraries using the steps mentioned above, which will be used in this Tutorial.

- EXECUTIVES
- MATERIAL PROPERTIES
- NODES
- PIPES&VALVES
- PLOTTERS
- PUMPS&COMPRESSORS
- TANKS DYNAMIC
- TOOLS
- TRANSMITTERS

2.6 ADDING ADMINISTRATIVE AND MATERIAL PROPERTIES OBJECTS

Section Concepts:

- Placing Administrative objects
- Setting Global units
- Placing a Material Properties object
- Selecting components
- Defining Default Liquid and Gas components to the model

2.7 PLACING IDEAS ADMINISTRATIVE OBJECTS ONTO THE WORKSHEET

Place the Discrete/Cont. Executive object from the EXECUTIVES library onto the worksheet by selecting the EXECUTIVES library under Library menu on the menu bar. This is done by moving the mouse to EXECUTIVES and clicking on Discrete/Cont. Executive when the submenu opens to the right of the main menu (see Figure 7 below).

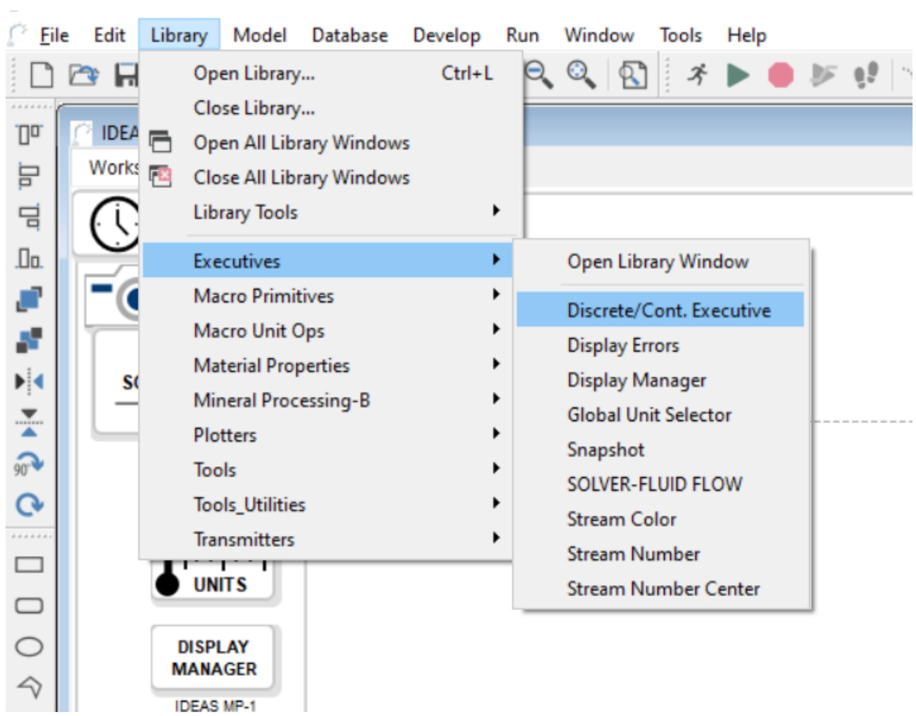


Figure 7: Placing Objects on Worksheets

Alternative method for object addition to model: Click on *Open Library Window* on the submenu (see Figure 7). A new window opens up on the worksheet containing all objects of that particular library (as in Figure 8). This library window shows each object's name, icon, and some additional details.

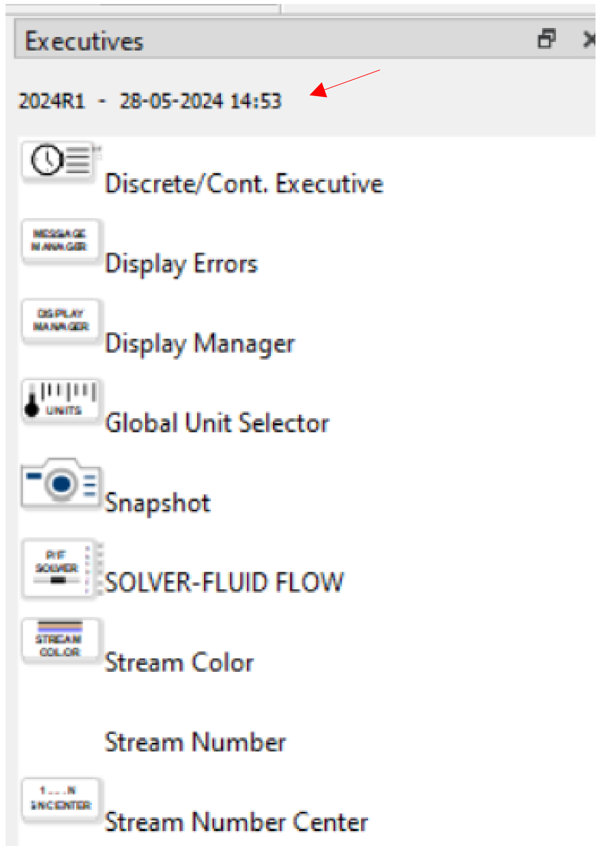


Figure 8: Alternative Method of Opening Libraries

To select an object from this window, click-and-place the object with the mouse to the desired location on the worksheet. The object will move to a default location – all administrative objects have a default location and should not be moved from this location.

To view all the open libraries from the Library Window, under the Library menu select Open All Libraries Windows. All the open libraries will show as tabs on the right side of library windows (see Figure 9). There are two buttons as a quick option to open all library windows for open libraries or close all library windows at the top-right of the application screen (see Figure 10).

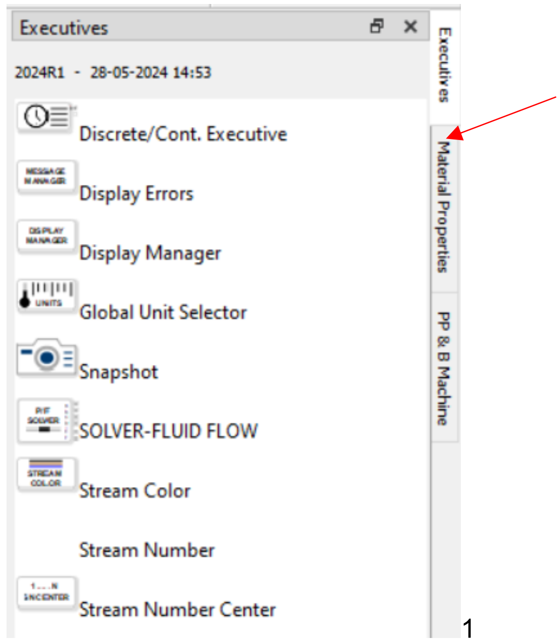


Figure 9: Tabs to View Opened Library Windows

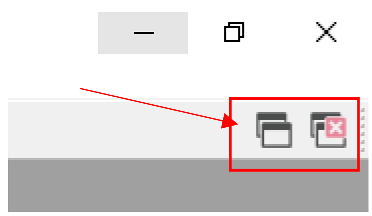


Figure 10: Buttons to Open (Left) or Close (Right) Library Windows for Open Libraries

All such menus as shown in Figure 10 with dotted lines are tear-away menus that can be relocated, so the top right is only the default location, and only if the menu is made visible from the Tools menu selection – under Tools this menu is “Library Window”. The Tools menu recommended selection is depicted in Figure 11.

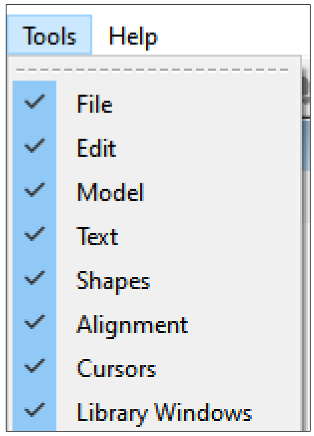


Figure 11: Tools Menu Recommended Selection for Visible Menus

Users can check and uncheck menus as needed; Icon menu is needed during Hierarchical Block creation (further discussed in the IDEAS 7.0.0. User Manual).

2.8 PLACING OTHER ADMINISTRATIVE OBJECTS ONTO THE WORKSHEET

Place the **Display Errors** and the **Solver-Fluid Flow** objects from the EXECUTIVES library onto the worksheet. The **Display Errors** object defines how errors and other messages are reported to the user. The **Solver-Fluid Flow** object performs flow calculations throughout the model for many of the objects with process stream flow. As noted earlier, allow these objects to move to their default location. Do not move them from those default locations.

2.9 PLACING A GLOBAL UNIT SELECTOR OBJECT ONTO THE WORKSHEET

Place the **Global Unit Selector** object from the EXECUTIVES library onto the worksheet. Double-click on the **Global Unit Selector** object to open the **Global Units Selector** dialog box. Under the **Primary Selection** tab, note that the **Metric** radio button is set. Click on **Accept Primary Units**. Go to the **Secondary Selection** tab. Click on the box to the right of the words **Flow Units** and change the setting from *Mass* to *Volume*. Proceed to the next two boxes and change them to liters (*L*) and minutes (*min*), respectively. Now, click the **Accept Secondary Flow Unit** button. Click **OK** to close the dialog box.

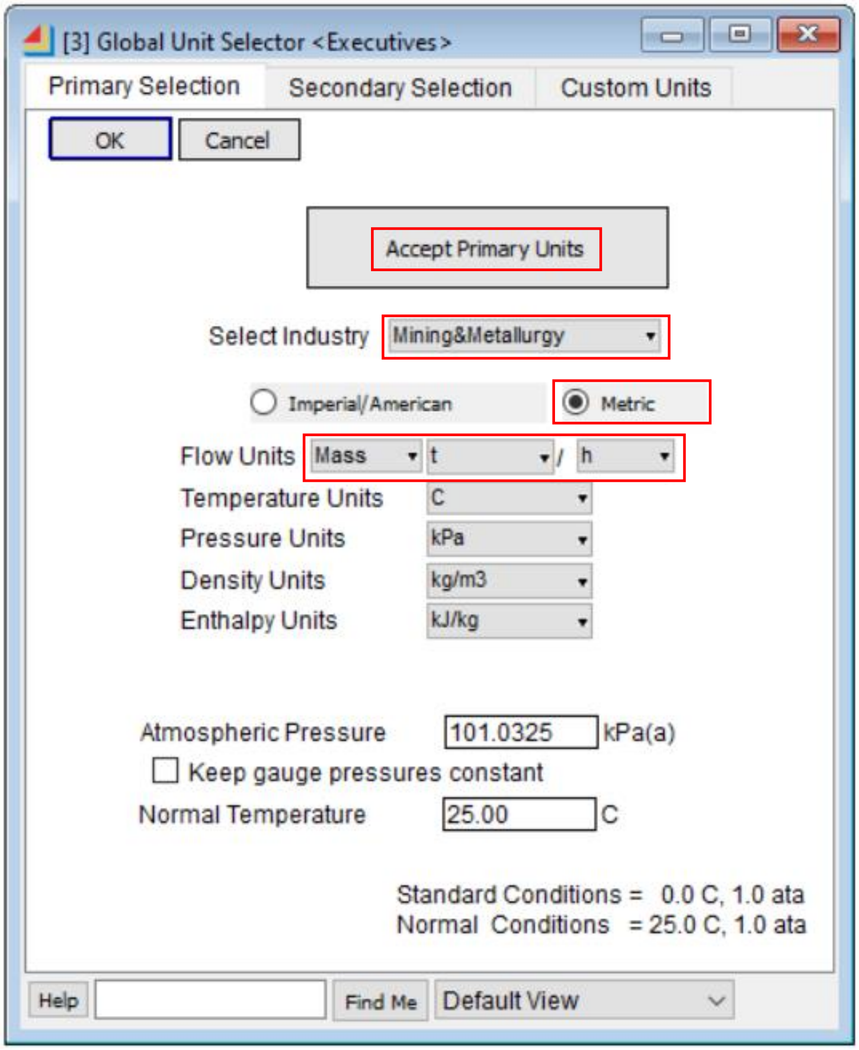


Figure 12: Global Unit Selector Dialog Box

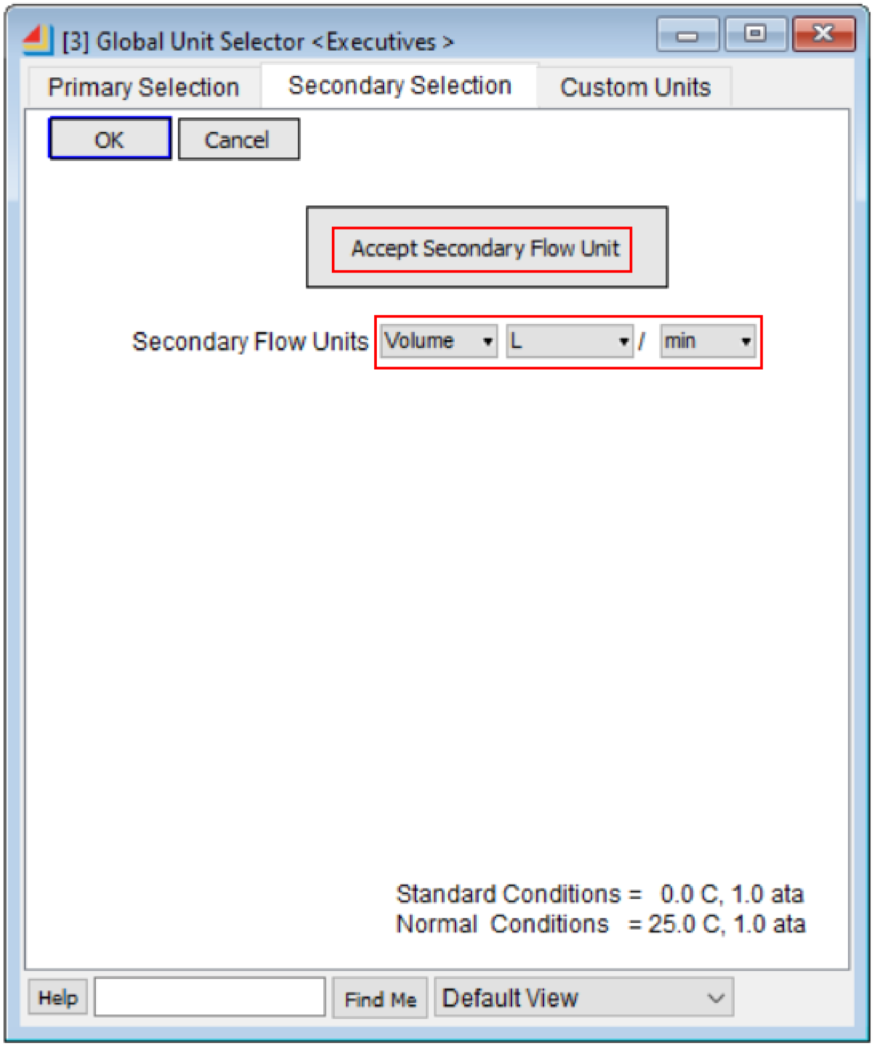


Figure 13: Secondary Flow Unit Selection

2.10 PLACING A MATERIAL PROPERTIES OBJECT ONTO THE WORKSHEET

Next, we will select the materials/chemicals/components for this model. Select the **Material Properties** object from the MATERIAL PROPERTIES library. When the **Material Properties** object is placed onto the worksheet, its dialog box automatically opens. In addition, the **Select IDEAS Components** window opens, prompting the user to select the components to be loaded. Loading a component loads the physical properties of that component into the **Material Properties** object and makes that component available for use in the model. Navigate through the IDEAS Components window where all the available components are displayed (see Figure 14).

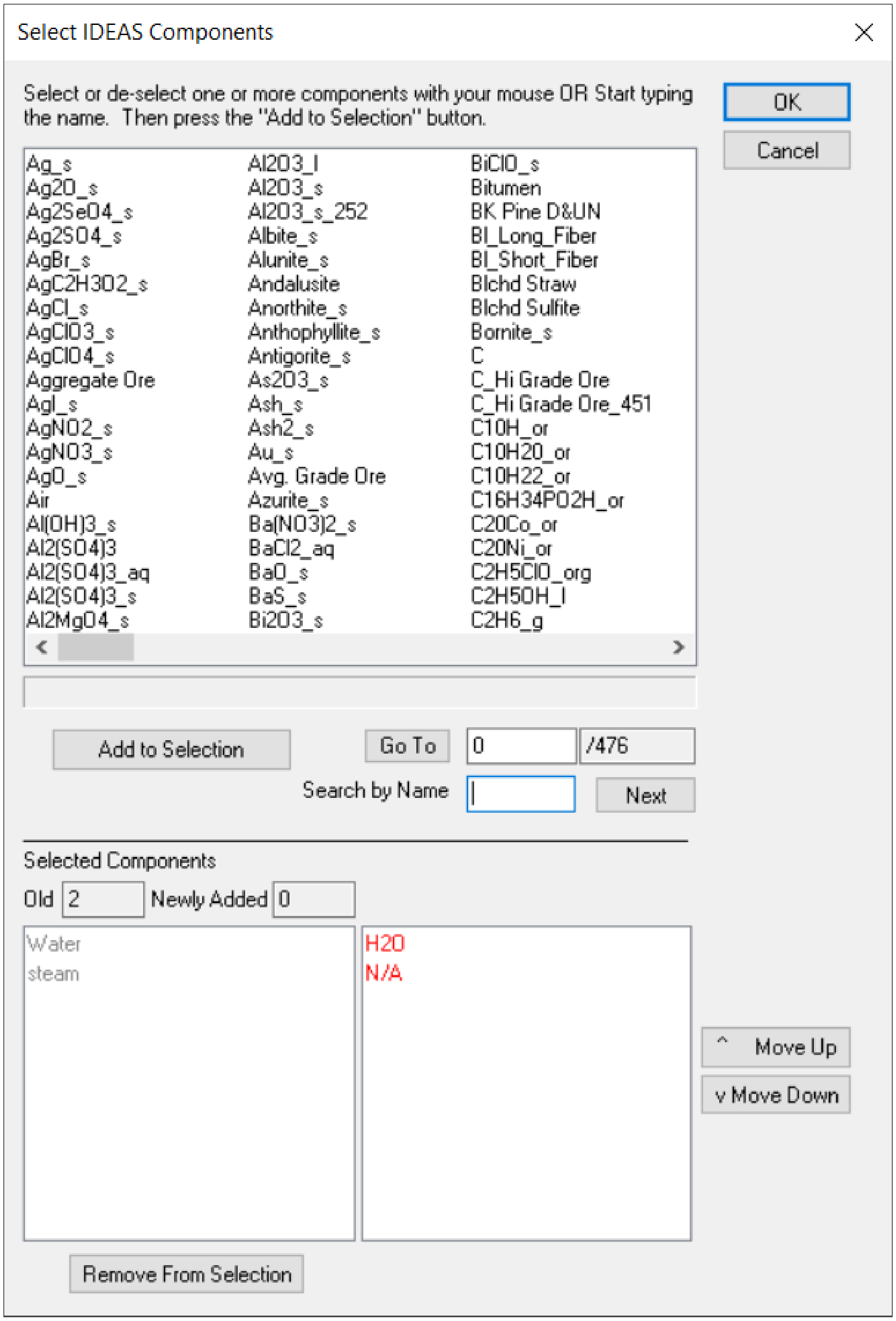


Figure 14: IDEAS Component Selection Window

2.11 OPENING A MATERIAL PROPERTIES COMPONENT

Water and Steam are selected by default. The user can select the component from the **Select IDEAS Components** window and make it available to the streams by double-clicking on the component or by pressing the **Add to Selection** button. The user can search for the components by entering full or part of a name in the **Search by Name** field. You will notice that *Water* is also now the default liquid component and *Steam* is now the default gas component because these are the first liquid and first gas components loaded into this **Material Properties** object. Press **OK** after selecting the components.

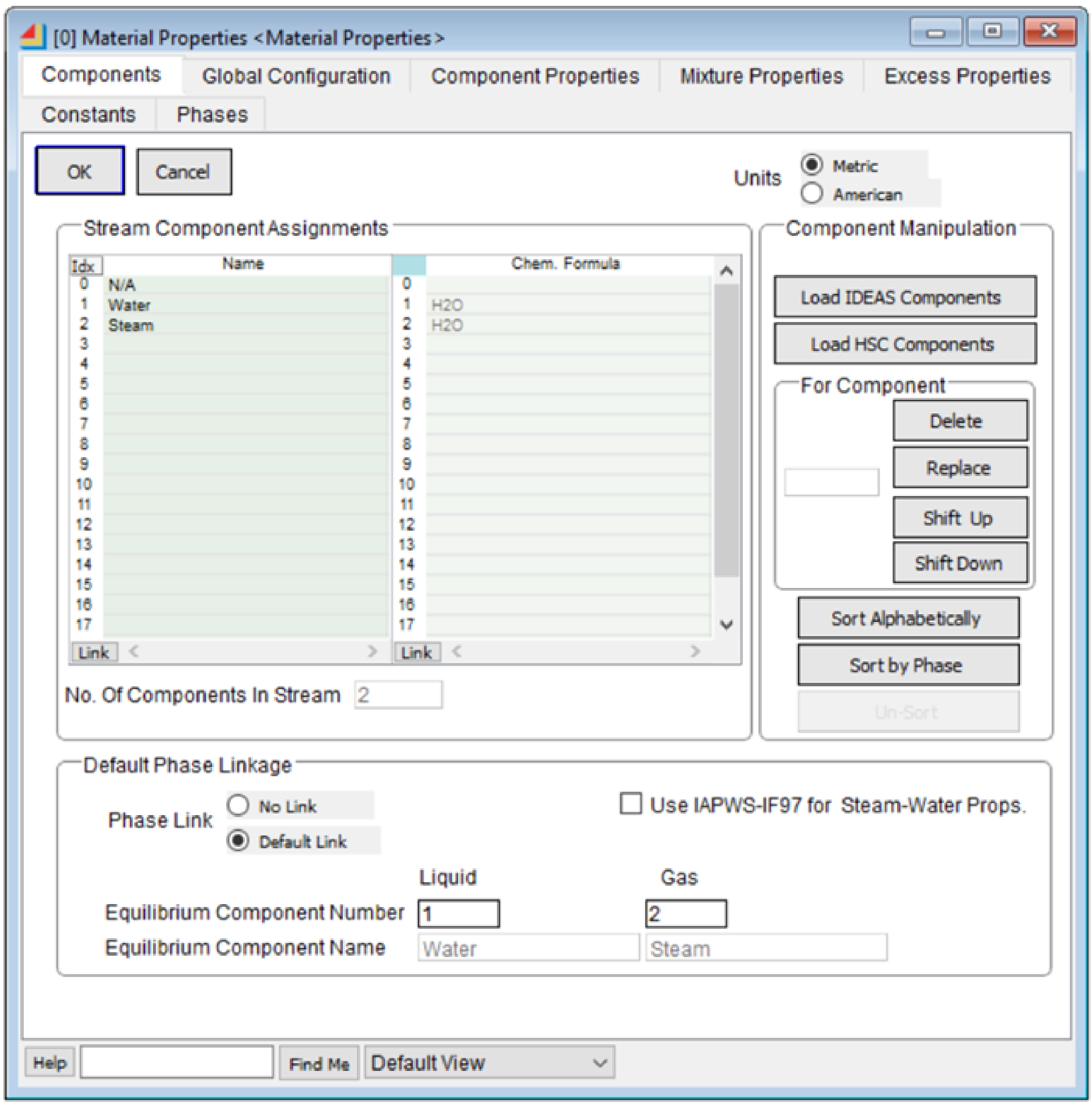


Figure 15: Stream Components Loaded

2.12 DEFINING THE EQUILIBRIUM COMPONENTS

Water and *Steam* are loaded as the first liquid and gas components, water, component number (1), was automatically entered into the **Equilibrium Component Number** field for liquids and steam, component number (2), was automatically entered into the **Equilibrium Component Number** for gas. In some cases, the user may want to change the default liquid to be some other component by changing the value in the **Equilibrium Component Number** field to match the desired default liquid component (see Figure 16). The component number is the number to the left of the component name in the **Stream Component Assignments** table. In Figure 15, steam is listed as the **Equilibrium Component Number** under the Gas subheading. The equilibrium component pair will be monitored for conditions leading to a phase change between the liquid and gas phases. Such phase change will automatically occur where appropriate.

Select the Default Link option next to Phase Link (see Figure 16) to enable Water/Steam Vapor-Liquid Equilibrium. If left as No Link, the flash calculations to convert water to steam or vice-versa will not occur.

The Default Component (see **Global Configuration** tab) is the fluid that by default will be present in IDEAS objects, such as pipes and tanks, when the model starts, unless specified otherwise in the object itself, and is also by default the equilibrium component. It can be changed to be a different component on the **Global Configuration** tab.

In this tutorial, we will only consider water present. So, remove the Steam component from the list by selecting it and then pressing the **Delete** button in the **Components** tab. A confirmation dialog box appears asking for confirmation to delete the component. Press **OK** to delete. Notice in Figure 16 that there is no gas equilibrium component now.

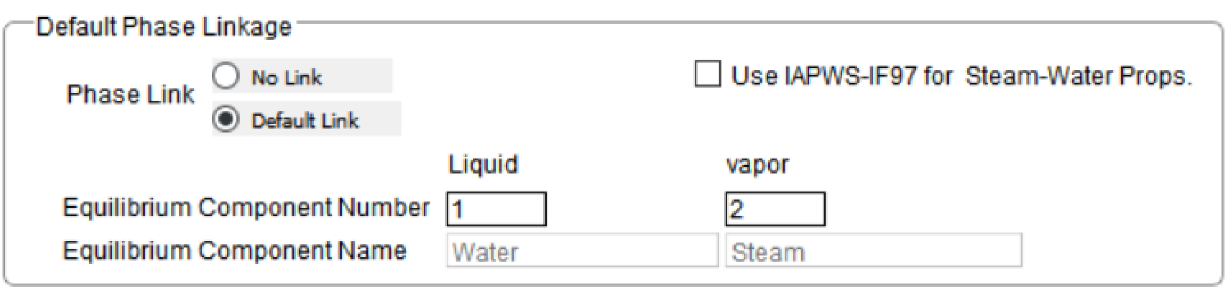


Figure 16: Setting the Default Liquid Component

Click on the **OK** button to close the dialog box.

For information about additional features of the **Material Properties** object, see the object's help text, and Appendix A: Introduction to Excess Properties.

Note: Always be sure to click on the OK button to close a dialog box if you wish to accept the changes that you made within a dialog box. If you wish to discard your changes, press the Cancel button.

2.13 ANIMATION OPTIONS

Select *Show 2D Animation* from **Run** on the menu bar. When selected, this option has a write symbol ( ) next to it. (Animation takes more CPU time, so if model speed is critical, deselect *Show 2D Animation*. In this case, we will leave the animation option enabled). When animation is disabled, the symbol turns red ( )

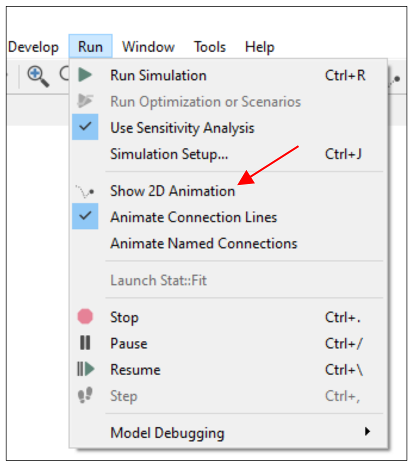


Figure 17: Selecting 2D Animation

SECTION 3. BUILDING A MODEL

Section Concepts:

- Build a Model of a Tank Filling and Emptying
- Use a Control Valve to Adjust the Flow
- Establish Settings for a Plotter to Record Data During Simulation

3.1 PLACING THE STREAM SOURCE AND SINK OBJECTS ONTO THE WORKSHEET

Place a **Stream Source** and a **Sink** object from the MATERIAL PROPERTIES library onto the worksheet. Move the objects to the left side of the worksheet. Please note that ALL objects placed on the worksheet from this point on must be placed to the right of the **Discrete/Cont. Executive** and **Solver-Fluid Flow** objects. Reserve the left edge of the worksheet for administrative objects only, and DO NOT move the administrative objects from their default locations. Additional administrative objects also should be placed to the left, but not further left than the **Material Properties** object.

3.2 SETTING UP THE STREAM SOURCE OBJECT

Double-click on the **Stream Source** object to open its dialog box. Select the **Composition** tab. You will notice that the component list is identical to the component list in the **Material Properties** object. (The **Material Properties** object is the storage center of information about each component used within the model.) In the Stream Source object, the **% Mass Composition** radio button should be selected. In the **% Mass** table, notice that the composition is 100% water at this point. This is true because Water is our default liquid (the only component so far).

3.3 SETTING CONTINUOUS UPDATE OPTIONS

Next, select the **State Variables** tab on the **Stream Source** object dialog box and click on the checkbox next to **Continuous Update** (Continuous Update takes more CPU processing power. If model speed is critical, do not select the **Continuous Update** checkbox for every object; instead, update the scan manually with the **Single Update** button during a simulation run). Notice how the dialog box is configured to display kg/s, which is our primary unit. Click on the **Flow** radio button under the **Source Type** heading. Change *kg/s* to *L/min* by using the two dropdown menus under the **Selectable Units** heading. Enter a value of *3000* in the **Flow** field. This **Stream Source** will now provide a constant 3000 L/min of 100% water when the simulation is running (see Figure 18).

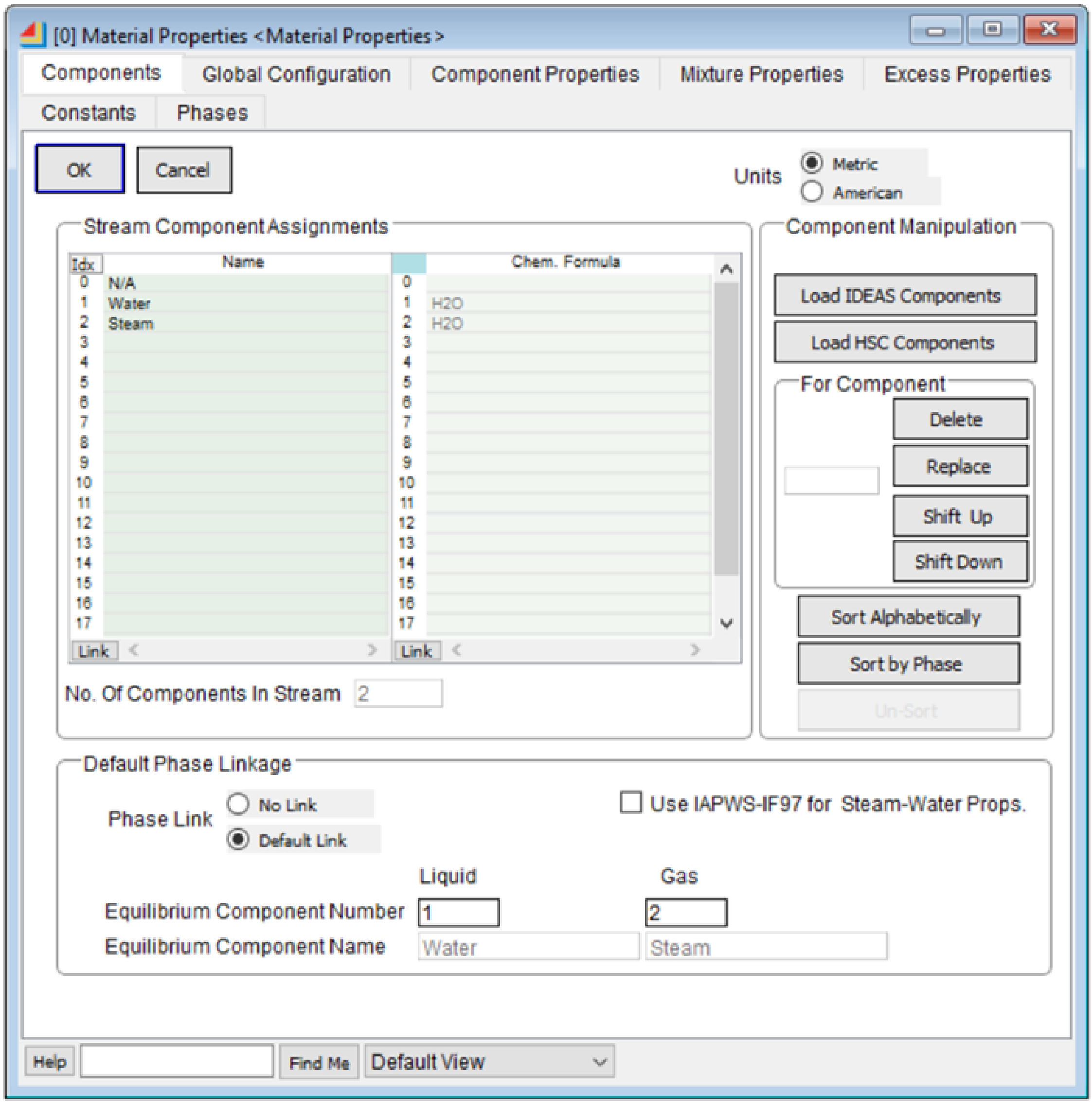


Figure 18: Stream Source Dialog Box Settings

Click on **OK** to close the dialog box. Notice that the color of the **Stream Source's** outlet connector has changed from green to red, which means that **Stream Source** is setting the flow rate (see Table 1).

Table 1: Red and Green Connector Descriptions

	Red	Green
Inlet	Sets Pressure	Sets Flow Rate

Outlet	Sets Flow Rate	Sets Pressure

3.4 PLACING A TANK-INCOMPRESSIBLE OBJECT ONTO THE WORKSHEET

Place a **Tank-Incompressible** object from the TANKS DYNAMIC library onto the worksheet and to the right of the **Stream Source** object.

3.5 CONNECTION LINE OPTIONS

Select Connection Lines under **Model** on the menu bar. When you move the mouse over *Connection Lines*, a submenu opens to the right of the main menu. Select and de-select options until your selections are identical to the ones shown in Figure 19. (Your default selections probably match those shown already.)

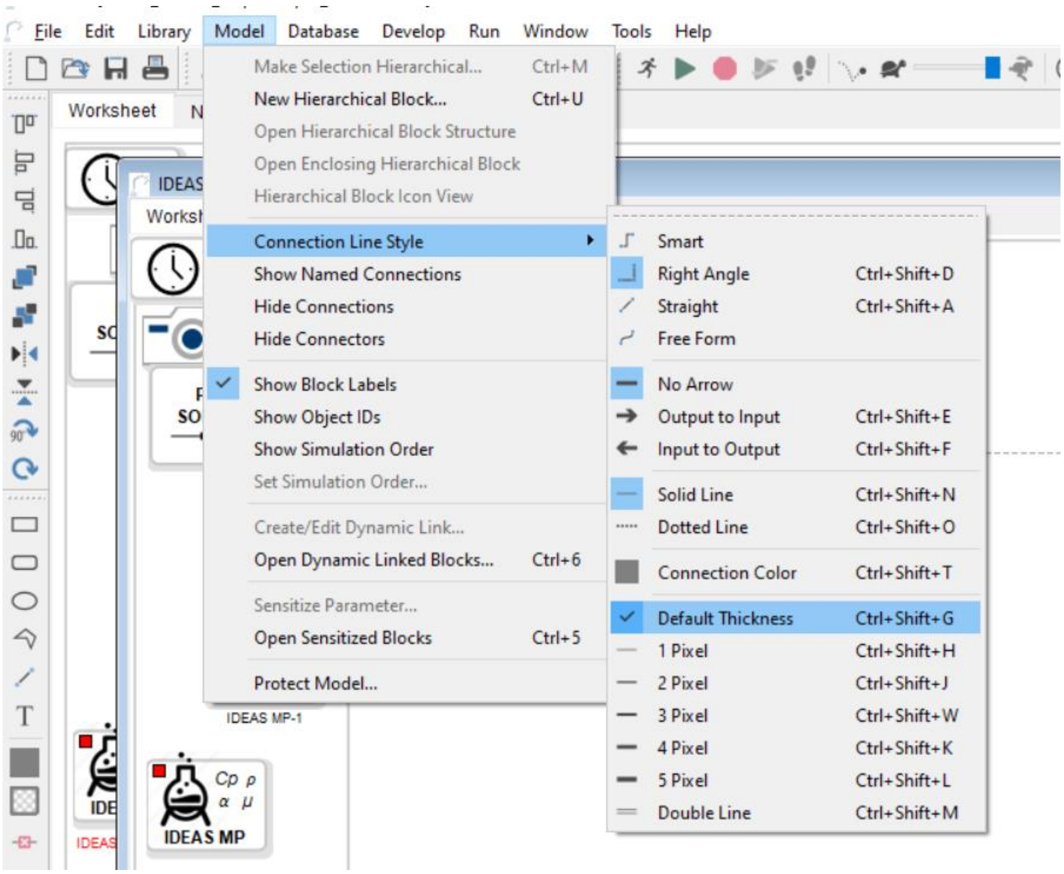
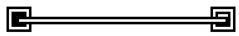


Figure 19: Choice of Connection Line Styles

Notice that we have selected the connection lines to be perpendicular, solid, and grey, and have the default thickness. Whenever we connect objects on the worksheet, the connection lines will obey these selections.

3.6 CONNECTING OBJECTS

To make connections between two objects, place the cursor over a connector. The cursor changes from an arrow to a new double-headed or head and tail arrow . Click on the first connector to start the line. Move the mouse to the destination connector and make sure the arrow changes again before clicking to complete the line. If you miss the connector, the mouse will still be in the connection line mode. Double-click to release the connection line without connecting to a final connector - if the connection is successful, the line is solid (inlet connector  outlet connector), if not, the line is dashed (inlet connector  outlet connector).

3.7 CONNECT STREAM SOURCE AND TANK OBJECTS

Connect the outlet of the **Stream Source** to the first inlet of the **Tank**. See Figure 20 for all connections and objects described so far. Notice that the tank also has scalar inlets and an outlet. We will use the LT scalar outlet later to help control the level of liquid in the tank.

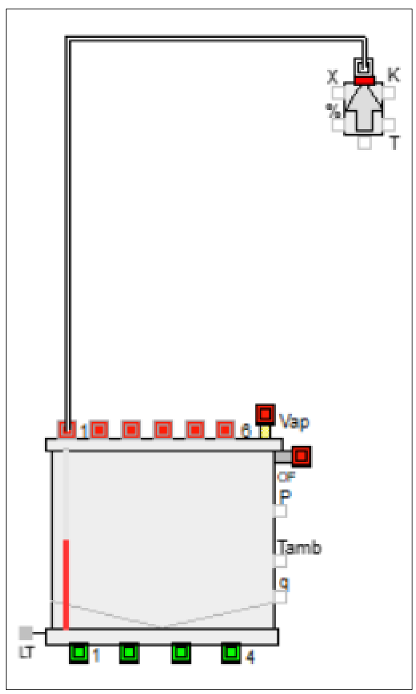


Figure 20: Tank-Incompressible Placement and Connection

3.8 OPEN THE TANK-INCOMPRESSIBLE DIALOG BOX

Open the Tank-Incompressible dialog box and click on the **Continuous Update** checkbox under the Display Parameters tab. Select the **Inputs** tab. Click on the **Level Animation On** checkbox. This displays an animated sight glass at the left side of the Tank-Incompressible icon (see Figure 20). Make sure the **Cylindrical Tank** radio button is selected and enter the following dimensions for the Tank-Incompressible: *3* for **D Inner**, *3.2* for **D Outer**, and *3* for the **Tank Height** (default unit = meter). Enter *3* in the **Overflow** field under the **Elevations from Bottom** subheading, which sets the overflow outlet at the top of the tank. Leave everything else at its default value. Note that the **Initial Liquid Level** default value is 50%.

3.9 CHECK TANK-INCOMPRESSIBLE PARAMETER SETTINGS

Examine Figure 21 and ensure that your dialog box settings match those shown. Choose the **Composition** tab and note that the default initial composition of the tank is 100% Water (because Water is the default liquid). Click **OK** to close the dialog box.

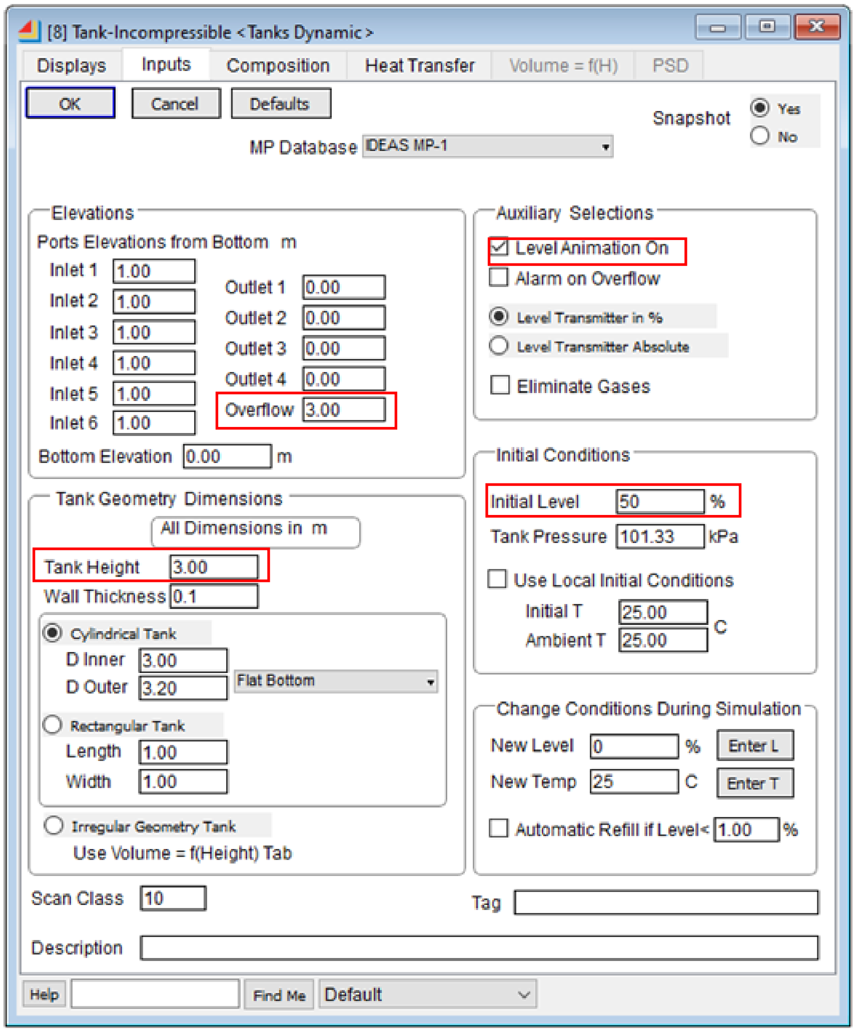


Figure 21: Tank-Incompressible Parameter Settings

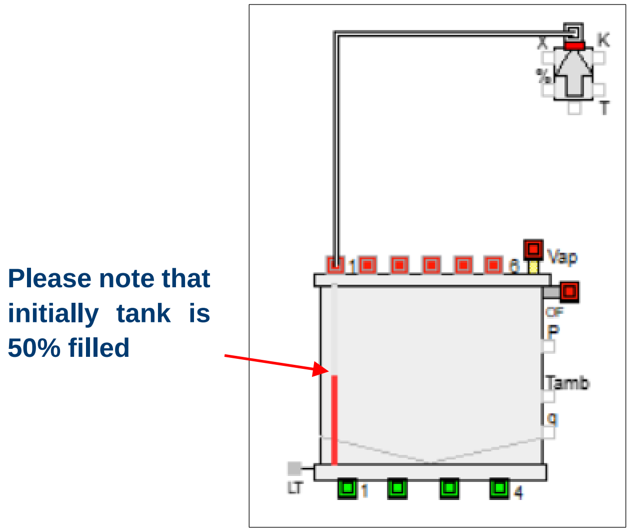


Figure 22: Tank-Incompressible, 50% Level Animation View

3.10 ADDITION OF A PUMP CENTRIFUGAL W/MOTOR OBJECT TO THE WORKSHEET

Place a **Pump Centrifugal w/Motor** object from the Pumps & Compressors library onto the worksheet. Move it below the Tank-Incompressible and connect the first outlet of the **Tank-Incompressible** object to the inlet of the **Pump Centrifugal w/Motor** object.

3.11 CONNECTING A SWITCH TO THE PUMP CENTRIFUGAL W/MOTOR

Place a Switch object from the TOOLS library onto the worksheet above the Pump Centrifugal w/Motor. Connect the outlet of the Switch to the motor ON connector of the Pump Centrifugal w/Motor (see Figure 23). The default setting for the Pump Centrifugal w/Motor is ON. With the addition of the switch, we can turn the pump ON or OFF.

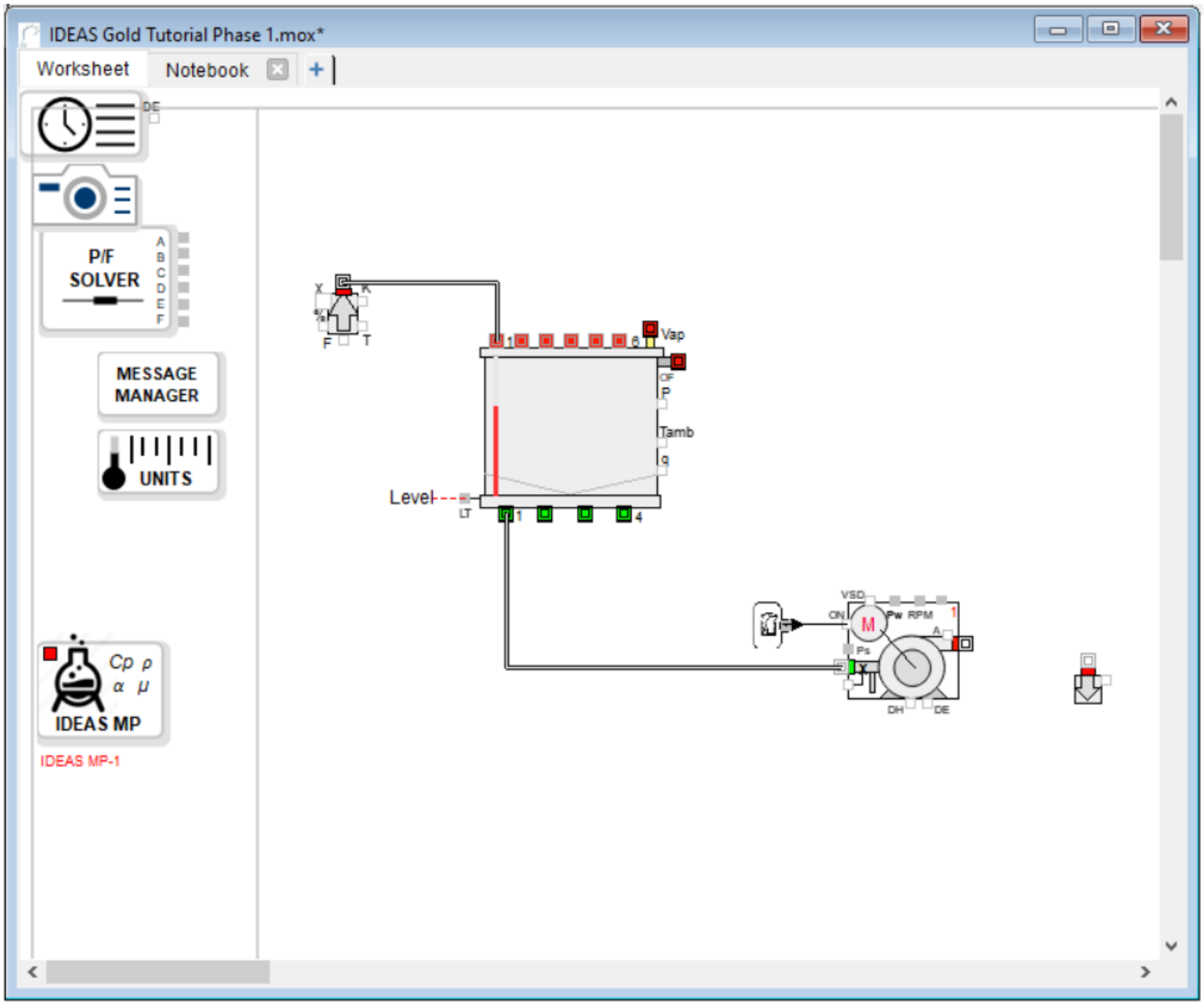


Figure 23: Pump-Selectable Impeller and Switch Placement and Connection

3.12 SETTING THE SWITCH

Open the **Switch** dialog box and click on the **Switch** icon so that it changes from 0 to 1. When the switch is set to 0, the pump will be OFF. When the switch is set to 1, the pump will be ON. Click OK to close the dialog box and notice later when running the simulation that the output of the switch is now green.

Note: For the purpose of this tutorial, it is necessary to place a Valve-Control at the outlet of the Pump Centrifugal w/Motor. However, the Pump Centrifugal w/Motor is a “flow” object (i.e., its connectors are “flow” connectors – green inlet/red outlet) and a Valve-Control is also a “flow” object. Since they cannot be connected directly, they must have a “pressure” object between them. This is accomplished by using a node that is a “pressure” object.

3.13 PLACING NODES

Nodes are fundamental building blocks in flow networks and serve two functions. First, nodes are used to connect various flow objects such as pipes, pumps, and valves. Second, network pressures are calculated at the nodes, and thus, the nodes can be seen as specifying boundary conditions for the flow objects. Node pressures are adjusted by the **Solver-Fluid Flow** object in order to solve network flows. A node is a pressure-flow object, which has its pressure calculated by the **Solver-Fluid Flow**.

Place a **Node-1 in-1 out** object from the Nodes library onto the worksheet to the right of the **Pump Centrifugal w/Motor**. Connect the outlet of the **Pump Centrifugal w/Motor** to the inlet of the **Node-1 in-1 out**.

3.14 PLACING A VALVE-CONTROL OBJECT ONTO THE WORKSHEET

Place a **Valve-Control** object from the PIPES & VALVES library to the right of the **Node-1 in-1 out**. Connect the outlet of the **Node-1 in-1 out** to the inlet of the **Valve-Control**. Connect the outlet of the **Valve-Control** to the inlet of the **Sink**. The **Sink**, in its default setting, is a pressure object, and since the **Valve-Control** is a flow object with flow connections, this combination works without any modifications. Notice that for all stream connections to be appropriate, the connections will be either red to red or green to green. Now, the model appears as in Figure 24.

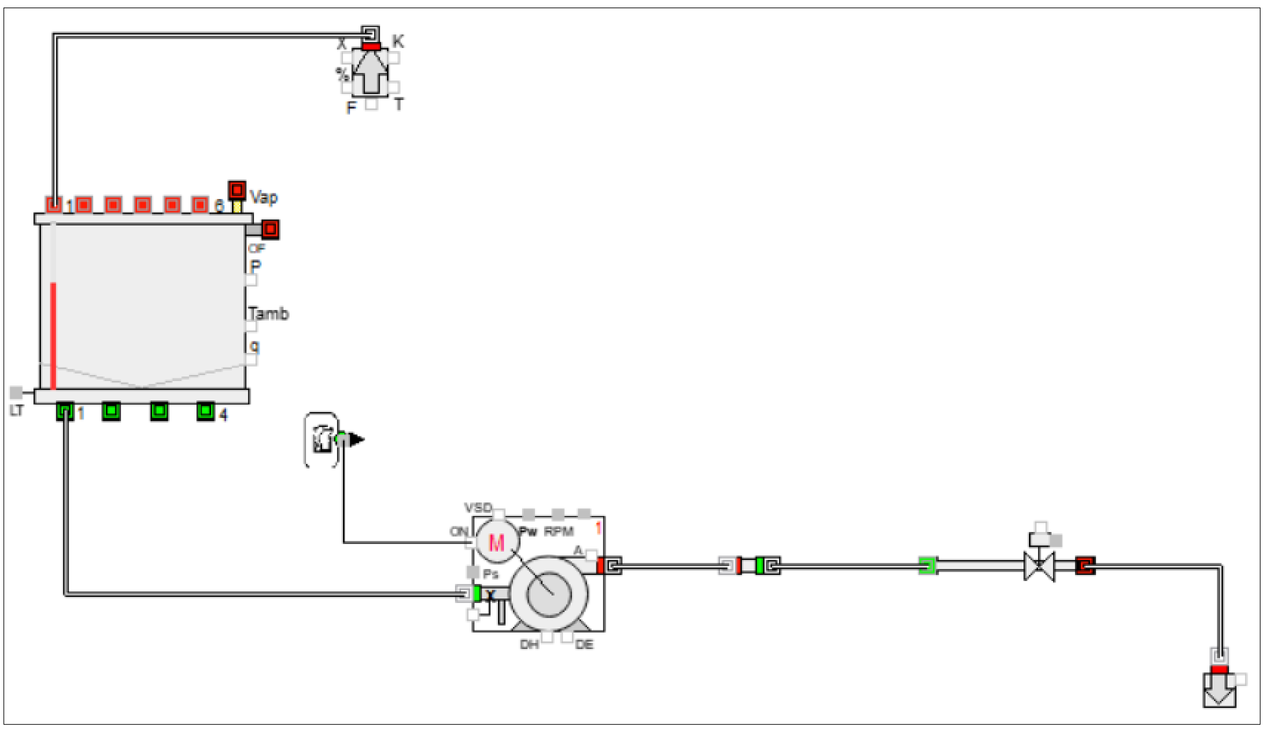


Figure 24: Valve-Control Object Placement and Connections

3.15 PLACING A SLIDER OBJECT ONTO THE WORKSHEET

Place a **Slider** object from the TOOLS library onto the worksheet above the **Valve-Control**. Connect the center outlet of the **Slider** to the scalar inlet on top of the **Valve-Control**. The default input range for the **Slider** is 0 to 100, and the default output is 50. Therefore, the **Valve-Control** stem position will now be 50%.

Note: To change the slider output, the user can drag the slider bar, or click anywhere in the gray box for the slider and use the keyboard. The up/down keys move the slider output by 1% of the range, and the Page Up/Page Down keys move the output by 10% of the range. The range limits at the top and bottom of the slider are dialog entries if the user clicks on them.

3.16 PLACING A TRANSMITTER-FLOW OBJECT ONTO THE WORKSHEET

Place a **Transmitter-Flow** object from the TRANSMITTERS library onto the worksheet. Connect the outlet of **the Valve-Control** to the inlet of the **Transmitter-Flow**.

3.17 PLACING A TRANSMITTER-PRESSURE OBJECT

Place a **Transmitter-Pressure** object from the TRANSMITTERS library onto the worksheet. Connect the outlet of the **Pump Centrifugal w/Motor** to the inlet of the **Transmitter-Pressure**.

Your worksheet should now resemble Figure 25.

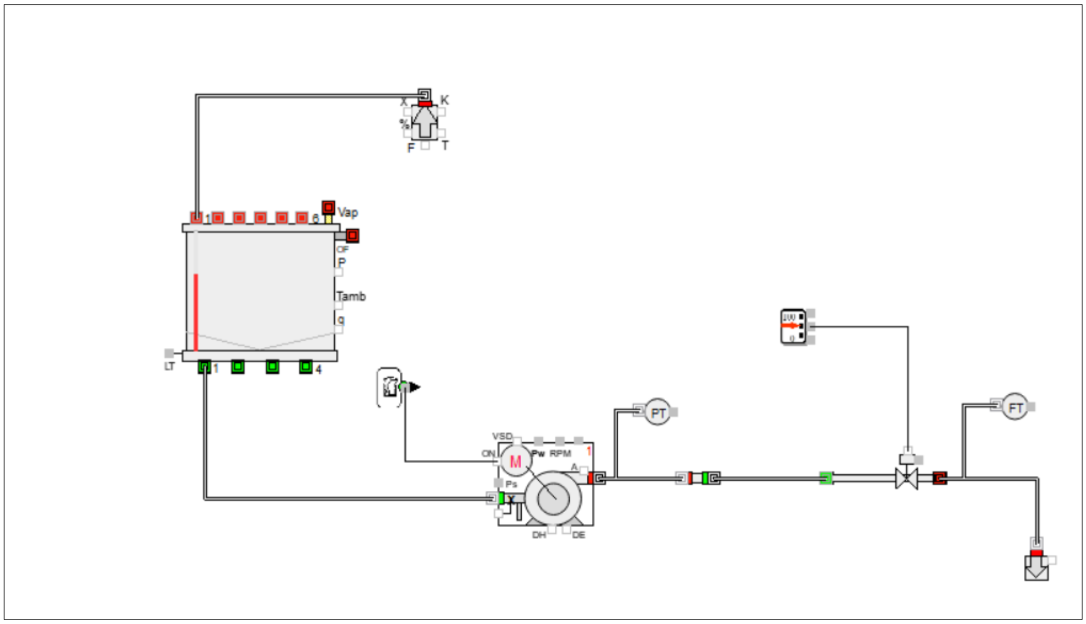


Figure 25: Transmitter Objects Placements and Connections

3.18 MONITORING THE FLOW

Open the dialog box of the **Valve-Control** object. Click on the **Continuous Update** checkbox so that the flow can be monitored. Next, choose the **Valve** Inputs tab. Change the *Cv @ 100% open* value from *1000* to *300*, thereby increasing the resistance through the valve. Choose the **Displays** tab and close the dialog box.

3.19 VIEWING THE FLOW INDICATOR BAR

Open the **Slider** dialog box and note that, as previously stated, the middle indicator bar is set at *50*. This indicator bar opens the **Valve-Control** to the corresponding percentage when the simulation runs. For example, 0 means the valve is closed, 50 means it is halfway open (50%), and 100 means it is fully open (100%). Close the dialog box.

3.20 SETTING OPTIONS FOR TRANSMITTER FLOW

Open the Transmitter-Flow dialog box and select the **Continuous Update** checkbox. Click on the Inputs tab and change the units from *kg/s* to *L/min*. Close the dialog box. This transmitter will monitor the flow rate of water exiting the control valve. There are many other options within this and other transmitters that we are not using in this tutorial exercise at this time. For more information on these additional features, see the TRANSMITTERS help text.

3.21 SETTING OPTIONS FOR TRANSMITTER PRESSURE

Open the **Transmitter-Pressure** dialog box and select the **Continuous Update** checkbox. Click **OK**. This transmitter will monitor the pressure at the pump's outlet.

3.22 PLACING A PLOTTER ONTO THE WORKSHEET

Place a **Plotter, I/O-scan** object from the Plotters library onto the worksheet.

3.23 CONNECTING OBJECTS VIA NAMED/TEXT BLOCK CONNECTIONS

Connect the **Transmitter-Pressure** outlet to the first (top left) inlet of the **Plotter, I/O-scan** by using a text connector in place of a line connection. Create a text block by double-clicking the mouse on the worksheet near the **Transmitter-Pressure** outlet; a text box will appear. Type in the label *Pressure*. Click somewhere else on the worksheet to close the text block. Copy this text block by selecting it and then choosing **Duplicate** from the **Edit** menu or CTRL+D. Move the second label near the first inlet of the **Plotter, I/O-scan**. Click-and-connect from each of the labels to their corresponding connectors. Note that the connector arrow only appears on the very edge of the text block, make sure the arrow changes before clicking to make a connection. The text block connectors makes a connection that is equivalent to a direct connection.

3.24 CONNECTING THE TRANSMITTER FLOW OUTLET

Connect the **Transmitter-Flow** outlet to the second inlet of the **Plotter, I/O-scan** using the same method, with a label (named) connector, *Flow*. Label connectors are also called named connectors.

3.25 CONNECTING THE LEVEL TRANSMITTER

Connect the **Level Transmitter** (LT) outlet on the **Tank-Incompressible** to the **Plotter, I/O-Scan** by using a named connector in place of a line connection (see the labels highlighted in yellow in Figure 26). Name this label, *Level*. The scalar values from each of the transmitters will now be sent to the plotter. You can also change the style of the labels by choosing the label, selecting the text itself, and then selecting the Text menu item for font and size changes and the color swatches tool to change the color of the text.

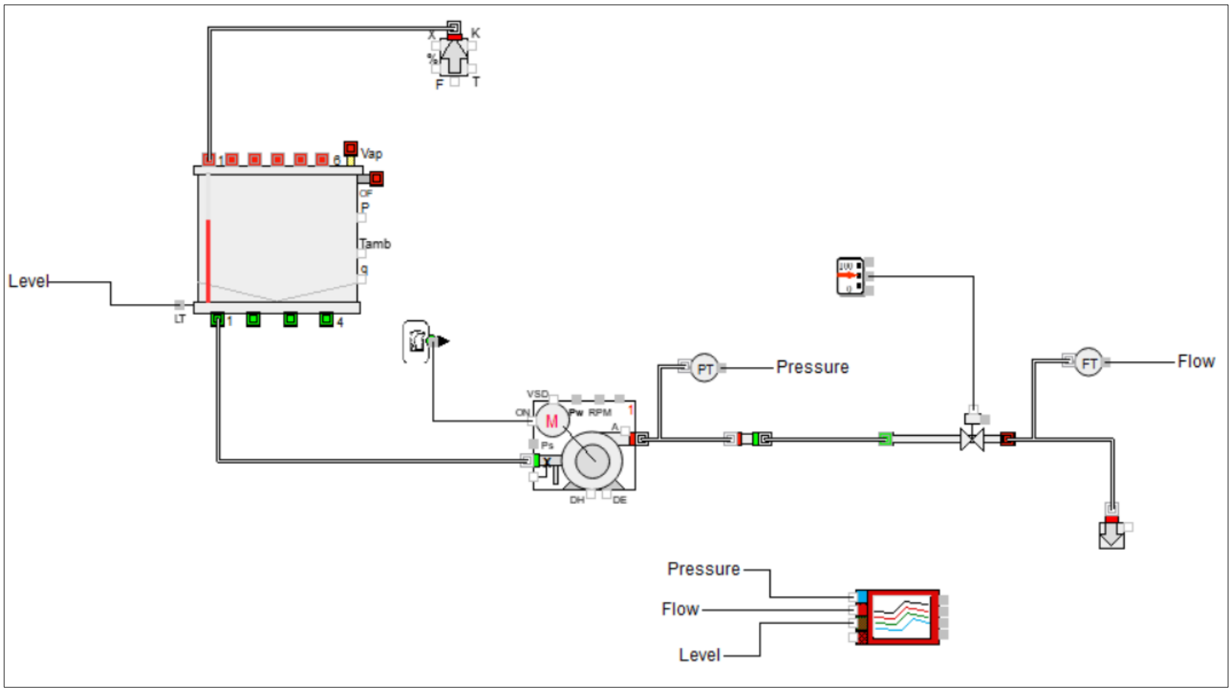


Figure 26: Plotter, I/O-Scan Label Connection

3.26 SETTING THE PLOTTER, I/O-SCAN DIALOG BOX

Open the **Plotter, I/O-scan** window. Right-click anywhere in the window and select the first option, Trace Editor. A second sub-window opens. In the first field (to the left of the blue square), type *Pressure*. In the second field (to the left of the red square), type *Flow*. In the third field (to the left of the green square), type *Level* (see Figure 27). These labels now coincide with the three labels connected to the **Plotter, I/O-scan**.

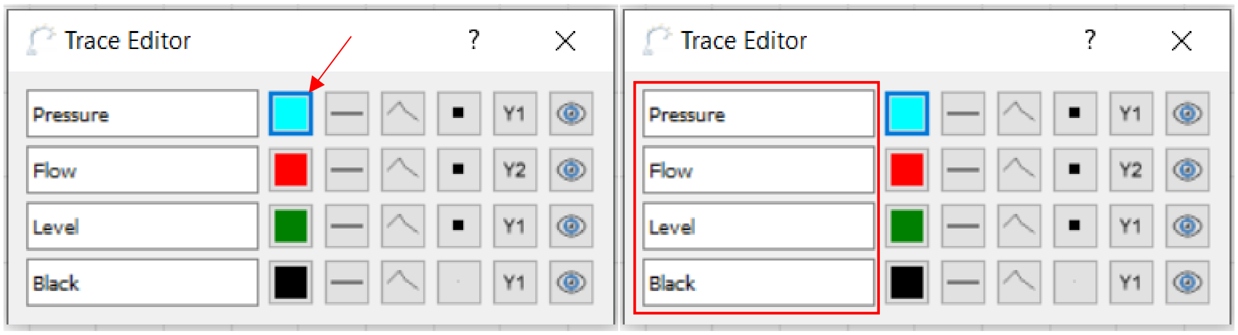


Figure 27: Plotter, I/O-Scan Sub-Dialog Box with Inlet Names

3.27 CHANGING THE PLOTTER, I/O-SCAN SCALE

Since the Flow values are an order of magnitude higher than the Pressure and Level values set for this model (i.e., Flow of about 3000 L/min, Pressure of about 445 kPa, and Level in the range of 0 to 100%), two different vertical scales will be used. Within the Trace Editor sub-dialog box, find the column that is second from the where it says Y1. This column represents the y-axis on the left-hand plotter scale. Click on the Y1 button associated with Flow (second row) and note that it flips Y2, which indicates that it is now using the right-hand scale (see Figure 28).

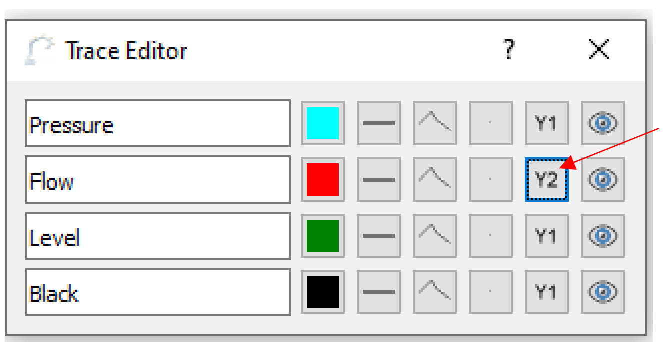


Figure 28: Modified Plotter Settings for Multiple Axes

3.28 SETTING PLOT LINES APPEARANCE

Next, by default a curved continuous line is selected (fourth column from left). The thickness of the lines can be adjusted by clicking in the third column for each row (just to the left of the curvy line) and selecting the desired line thickness. Close this sub-dialog box by clicking on the **X** at the top right-hand corner. The main **Plotter, I/O-scan** dialog box should still be open (see Figure 29).

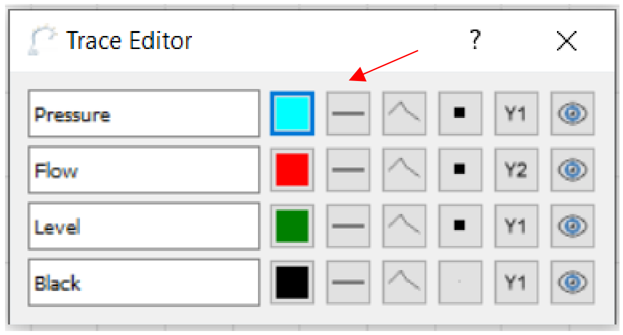


Figure 29: Modified Plotter Settings for Line Thickness

3.29 ADJUSTING PRESSURE, LEVEL AND FLOW VALUES

The left and right-hand vertical (y) scales for the expected Pressure, Level, and Flow values need adjustment. Click on the upper left-hand number; it should highlight. Type in a value *500* and press the **Enter** key. The value is set to accommodate both the Pressure and Level variables (i.e., the Pressure is about 445 kPa and the Level is between 0 and 100%). Change the upper right-hand value to *4000* and press the Enter key. The value is set to accommodate the Flow out of the control valve (i.e., the Flow is about 3000 L/min). Similarly, reset the low values for both scales from *-1* to *0* (see Figure 30). Click on the **X** in the upper right-hand corner to close the dialog box.

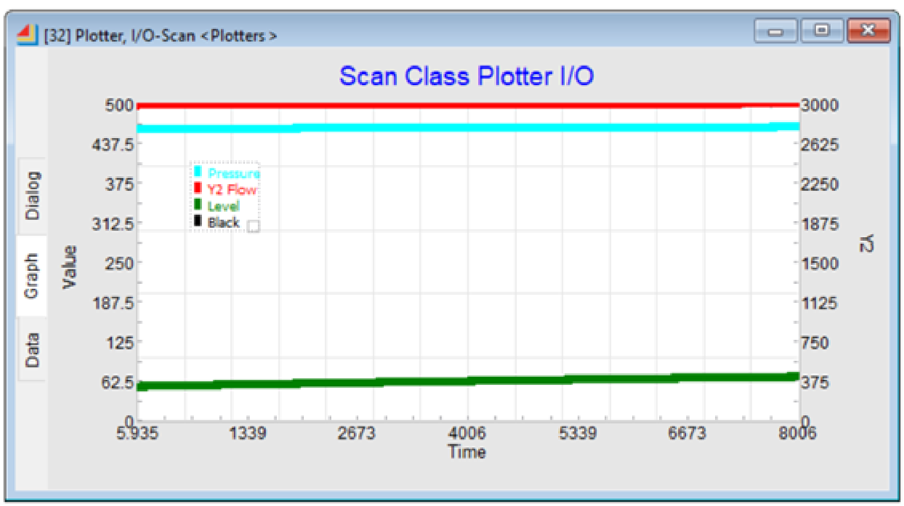


Figure 30: Plotter, I/O-Scan Dialog Box Adjustments

3.30 MODIFYING THE SIMULATION RUN TIME

Select **Simulation Setup** from **Run** on the menu bar. Choose the **Continuous** tab. Type *8000* in the End time field (see Figure 31). This ensures that the simulation runs for a longer period before stopping on its own, thus providing a long enough span of time during which the model may be paused for review. The **Time per step (dt)** radio button is selected and the number *1* in the **Time per step (dt)** field. The Global time units are set as **Generic**. These settings make each step of the simulation represent one (1) second of real time.

The time units have to be either Generic or Seconds to viscosity calculations correct. The time per step can be altered but 1 second works for the vast majority of models. Make sure that the *Left to right* simulation order is selected. Close the dialog box.

Note: This concludes the basic construction of the model. The model is now ready to be run. Since parameters for valve and pipe diameters, valve size, pump curves, elevations, etc., were not specified, default values are being used for these variables.

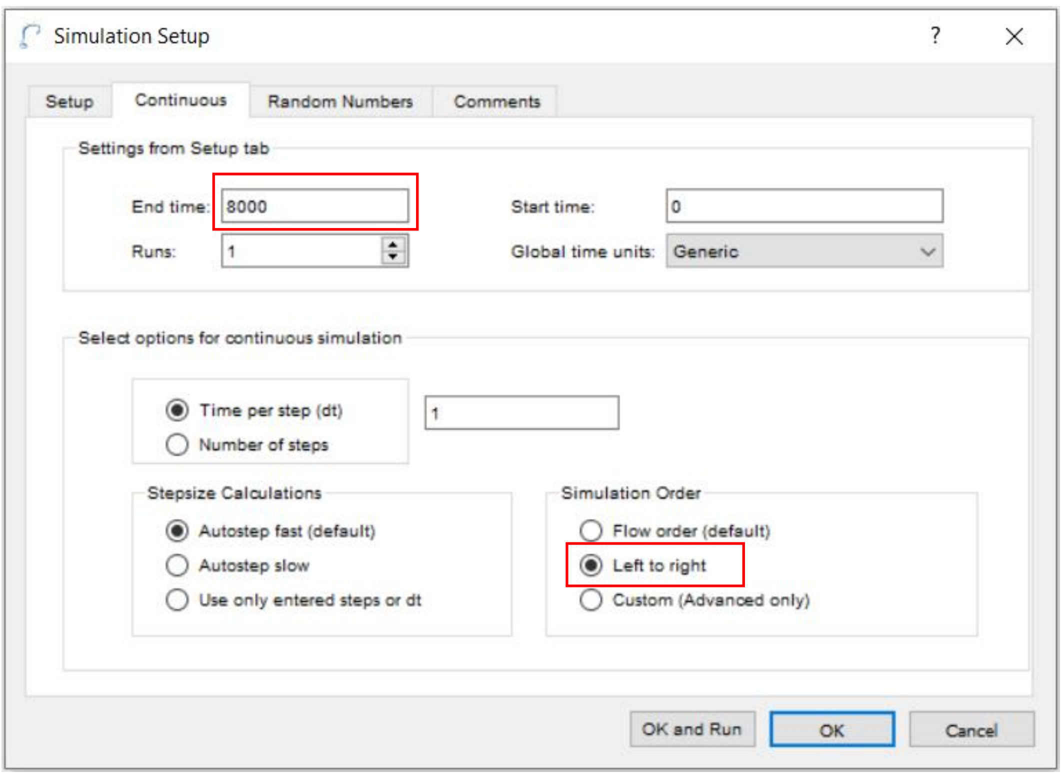


Figure 31: Simulation Setup Settings

SECTION 4. RUNNING THE MODEL

Section Concepts:

- Run the Model
- Adjust the Input Parameters for Objects
- Observe Changes in Pressure, Flow, and Level Using the Plotter
- Open Dialog Boxes to Observe Specific Changes in Objects
- Adjust Variable Plots

As you read the remainder of the tutorial, it may be necessary to pause the simulation in order to read ahead and become familiar with the further steps. Please do this as often as needed. The **Start** button is located in the tool bar (see Figure 32), and becomes the **Pause/Resume** button one simulation has started. Press the **Pause/Resume** button to pause and to resume the simulation. You may have to restart the simulation if you run out of time. You can increase the simulation time by selecting **Simulation Setup** from **Run** on the menu bar. Choose the **Continuous** tab and type a larger number in the **End time** field. This ensures that the simulation runs for a longer period of time before stopping on its own, thus providing a long enough span of time during which pauses can be made for model review.

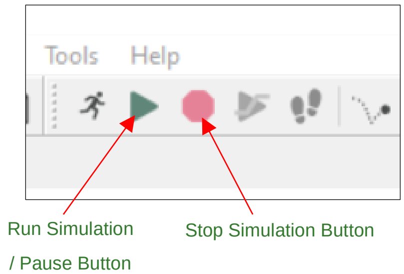


Figure 32: Run Simulation, Stop Simulation and Pause/Resume Buttons

4.1 EXAMINATION OF MODEL CONDITIONS

Open the **Tank-Incompressible** dialog box and select the **Displays** tab. With the dialog box still open, select Run Simulation from **Run** on the menu bar. After a short initialization period, the simulation starts. Click on the **Inlet1** dialog box field to alter to secondary Units chosen in the **Global Unit Selector** object.

What you see: The flow into the **Tank-Incompressible** is 3000 L/min from the **Stream Source** object and the flow out of the **Tank-Incompressible** should reach 2951 L/min as well as rising to a level of 90 to 95% by the time the simulation finishes—this depends on some of the defaults of the equipment and the version of software you have.

Click on the worksheet to bring it to the front of the screen. This leaves the **Tank-Incompressible** dialog box open but in a window behind the worksheet.

4.2 CHECKING VALVE-CONTROL FLOW CONDITIONS

Open the **Valve-Control** dialog box and select the **Displays** tab. Click on the **Pause/Resume** button in the toolbar. The **Continuous Update** checkbox is selected, and the **Choked Flow Conditions** field shows Normal Flow (versus Critical Valve Flow), which means that the flow is not in the choked flow region (see Figure 33).

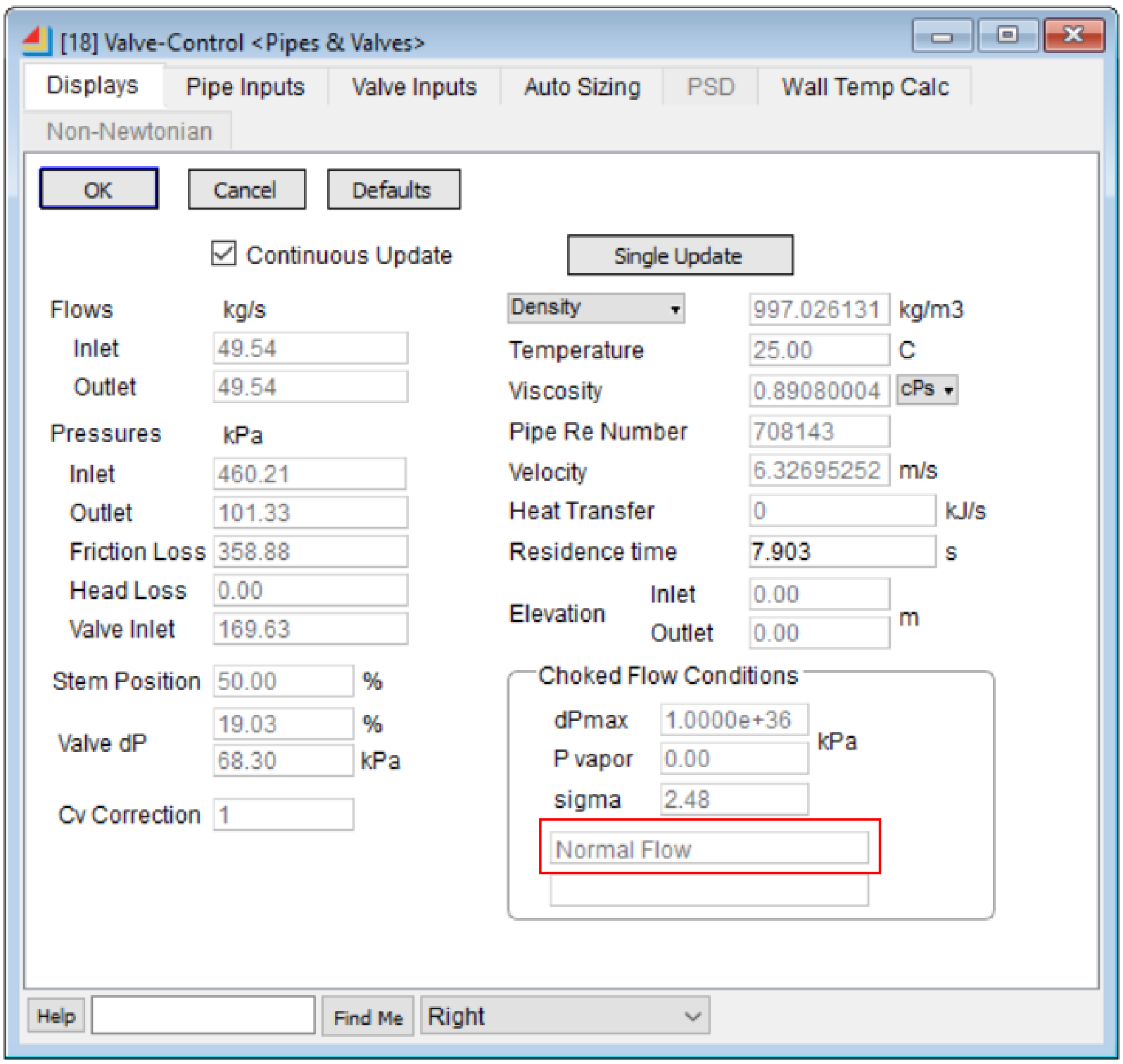


Figure 33: Valve-Control Display Parameters

4.3 CHANGING THE SINK DIALOG BOX SETTINGS

Open the **Sink** dialog box. Select *kPa* as the pressure unit from the dropdown list. Type *445* in the Pressure value field (see Figure 34). Close the dialog box by clicking OK. In previous IDEAS versions, this would pause the model, typing in an entry during model running would pause the model, but it no longer pauses. Users have to click away from the entry before the altered value is accepted or click the OK button.

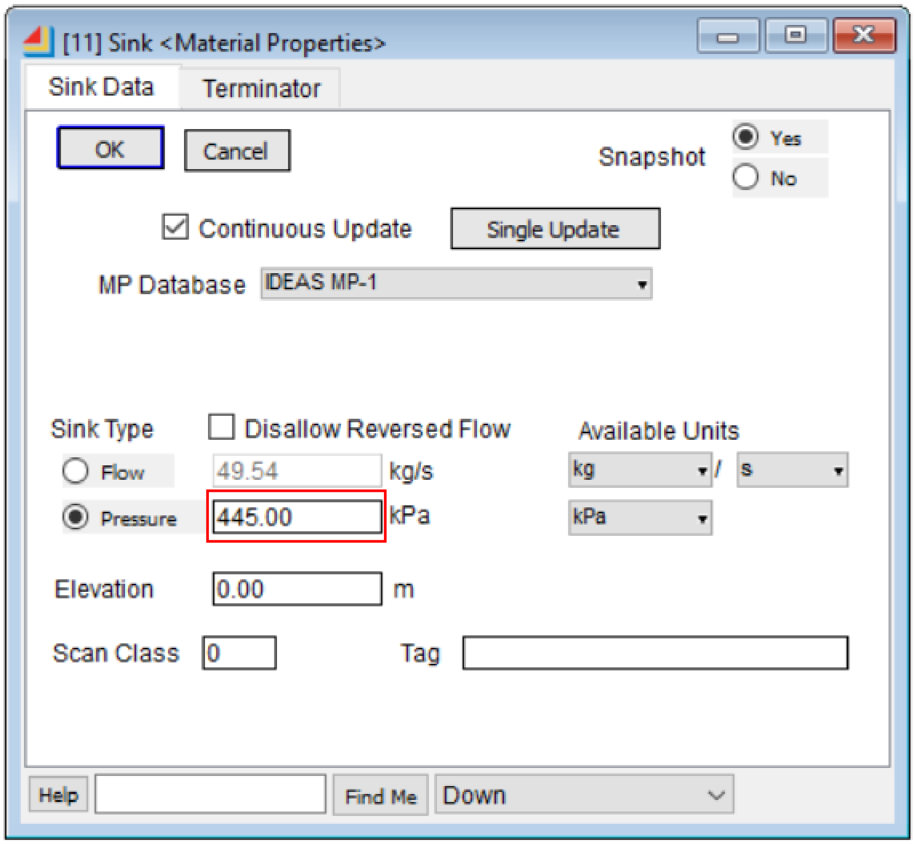


Figure 34: Sink Dialog Box during Simulation Run

4.4 VIEWING CHANGES IN THE TANK-INCOMPRESSIBLE DIALOG BOX

Open the **Tank-Incompressible** dialog box and click on the **Displays** tab. Notice that because the flow through the **Valve-Control** is less than 3000 L/min entering the **Tank-Incompressible** (caused by the recently changed pressure at the **Sink**), the **Tank-Incompressible** level is increasing. After the tank fills, some of the water will exit through the overflow outlet at the top of the tank (see Figure 35). Close the dialog box.

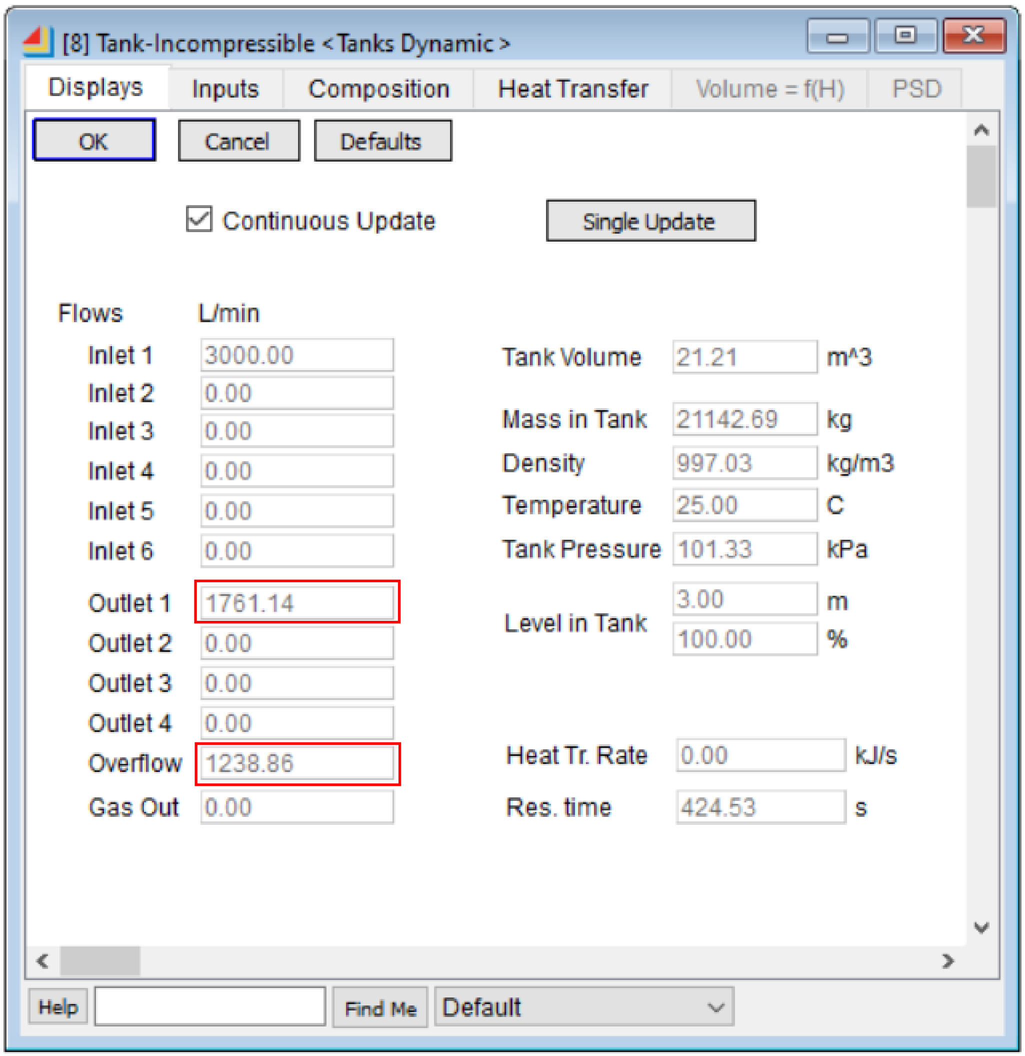


Figure 35: Tank-Incompressible Liquid Level

4.5 ADJUSTING THE PLOTTER I/O-SCAN PLOTS

Open the **Plotter, I/O-scan** dialog box. Notice the step change of the Flow, Level, and Pressure caused by changing the pressure in the **Sink**. The new, higher downstream pressure provides more resistance to flow.

4.6 MANUALLY ADJUSTING THE DISPLAY RANGE

The **Plotter, I/O-scan** automatically scales the time scale (x-axis) to the total run time of the simulation. The plot can be re-scaled by changing the run time to a smaller value (and running the model again), or by manually entering new minimum and maximum values for the x and y axes, or by selecting the **Autoscale XY** option from the right-click menu. Double-click on the maximum or minimum value on the X or Y or Y2 scale, type a new value and press **Enter** to manually change the display range.

4.7 MODIFYING THE PLOTTER, I/O-SCAN WHILE THE SIMULATION RUNS

Changes can also be made to the **Plotter, I/O-Scan** while the simulation is running. Using the same techniques as above, try rescaling the **Plotter, I/O-Scan** as the simulation runs.

Click on the **Stop** button in the toolbar and close the Plotter window.

This is the end of the initial model building. **Save** the model and proceed to the next section.

SECTION 5. SIMULATION OPTIONS

<u>Section Concepts:</u>

- Clone Dialog Box Items and Place them On the Worksheet
- Use the Snapshot Function as a New Starting Condition for The Simulation

5.1 SAVING THE MODEL

Select **Save Model as** under **File** on the menu bar. When the **Save File As** dialog box appears, navigate to the directory created at the beginning. Next, type *Standard Worksheet II* in the **File Name** field and click on the **OK** button. If a dialog box opens that indicates "This file already exists. Replace existing file?" - click on the **Yes** button.

Open the **Sink** dialog box and change the **Pressure** field back to atmospheric pressure (101.325 kPa) if it is not already. Close the dialog box. See Figure 36.

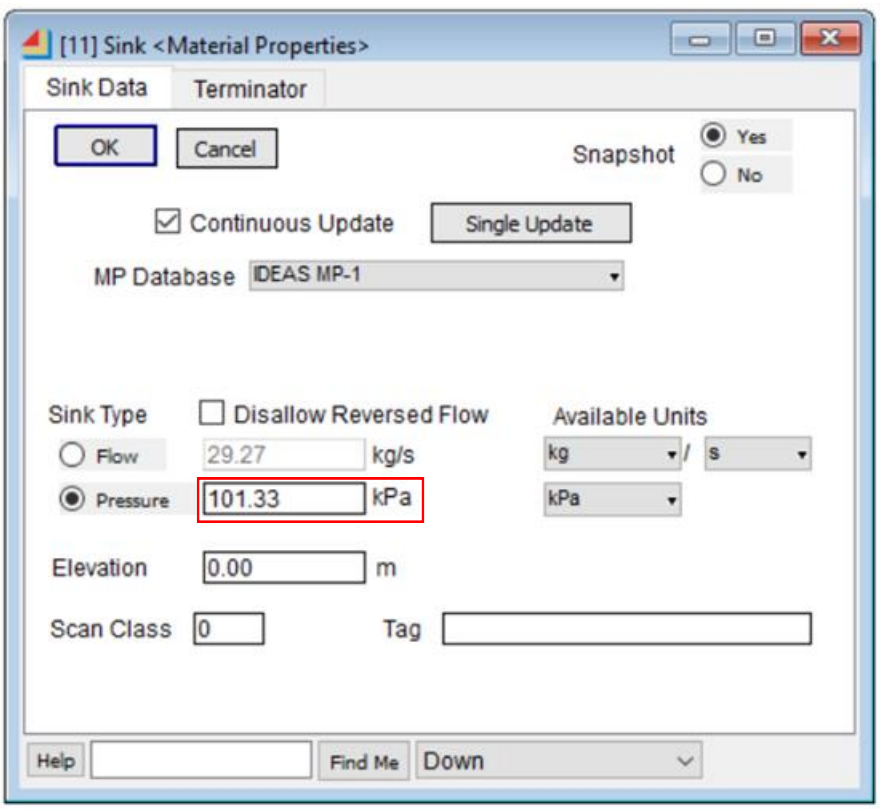


Figure 36: Sink Pressure Setup

5.2 CLONING A DIALOG BOX ITEM

- Cloning of ANY dialog box item can be accomplished by following these steps:
- Select the clone layer tool ( ) from the toolbar. See Figure 37 for further example.
- Open the dialog box and select the dialog box item to be cloned. You can choose multiple items at once by clicking on an empty space in the dialog box, hold down the mouse button and drag to highlight all the selected entries.
- Stop holding down the mouse button and the cloned entries will follow your mouse. Click on worksheet window to drop the clones to that location.
- Once you are done, click on the Main Cursor tool (arrow) in the toolbar.
- Besides residing the toolbar, the cursor options are often available in the right-click menu.

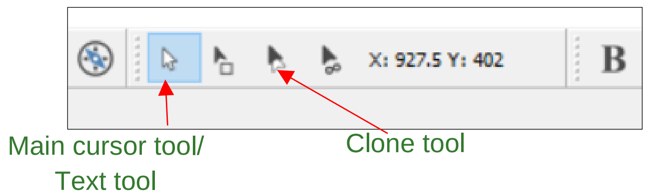


Figure 37: Clone Tool and Main Cursor Tools Location in IDEAS Toolbar

5.3 CLONING THE PRESSURE TRANSMITTER

Open the **Transmitter-Pressure**'s dialog box. Select the cloning tool ( ) from the toolbar at the top of the screen. Using this tool, click-and-drag to create a selection box that highlights both the Pressure value and units field. The two boxes highlight once the mouse button is released (see Figure 38).

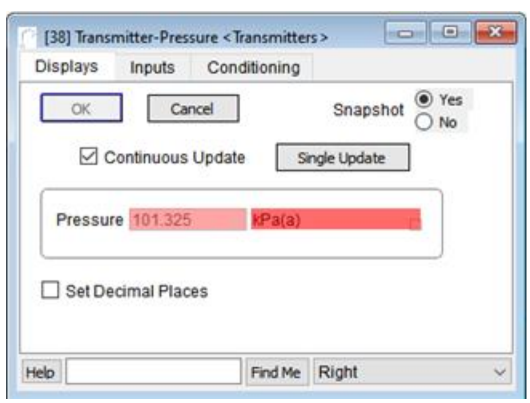


Figure 38: Transmitter-Pressure Boxes Clone Selected

Once the mouse is released, the cloned displays will follow you mouse until your next worksheet or notebook click. Place the cloned boxes at a location near the **Transmitter-Pressure** icon by clicking near the transmitter.

5.4 CLONE THE VARIOUS DIALOG BOX FIELDS

In a similar manner, clone the Flow value and units items from the **Transmitter-Flow** dialog box, the **Switch** icon from the **Switch** dialog box, and the **Slider** icon from the **Slider** dialog box.

5.5 PLACE THE BOXES AND ICONS ONTO THE WORKSHEET

Place the boxes and icons onto the worksheet at a location near their corresponding objects (see Figure 39).

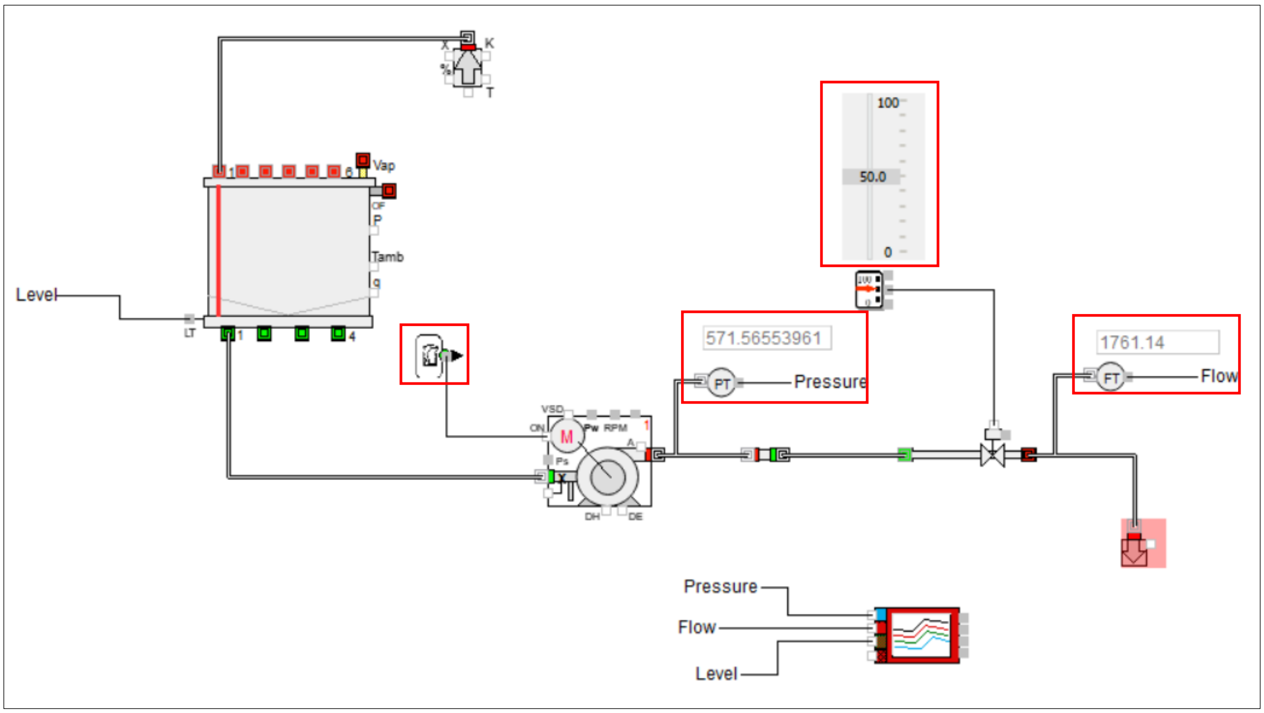


Figure 39: Transmitter-Flow/Switch/Slider Cloned Items Placement

5.6 CLOSE THE DIALOG BOXES

Select the pointer tool ( ) from the toolbar at the top of the screen, and close each of the dialog boxes.

There are a few important qualities one should know about the clone tool and cloned displays:

- Cloned objects on the worksheet can only be selected when the Clone Layer Tool or the All Layers Tool (the button to the right of the clone tool) is selected from the toolbar.
- A cloned item acts exactly as the item does in the dialog box. If the cloned item is of an input field, changing its value in the clone will also change its value in the dialog box. Be careful when altering information in a cloned item.
- In order for the display in the clone to be Continuously Updated, the Continuous Update checkbox inside the dialog box must be checked.

5.7 RUNNING THE UPDATED SIMULATION

Select **Run** Simulation from under **Run** on the menu bar or by pressing the **Run simulation** button (the first green arrow). Note that the clone values of the Transmitter-Pressure and Transmitter-Flow update automatically. Manipulate the cloned **Switch** and **Slider** and observe the changes in the values of the **Transmitter-Pressure** and **Transmitter-Flow**.

Click on the **Pause/Resume** button in the toolbar.

5.8 SETTING FLOW OPTIONS

Open the **Stream Source** dialog box, go to **State Variables** tab, and type “0” in the **Flow** field (see Figure 40). Close the dialog box.

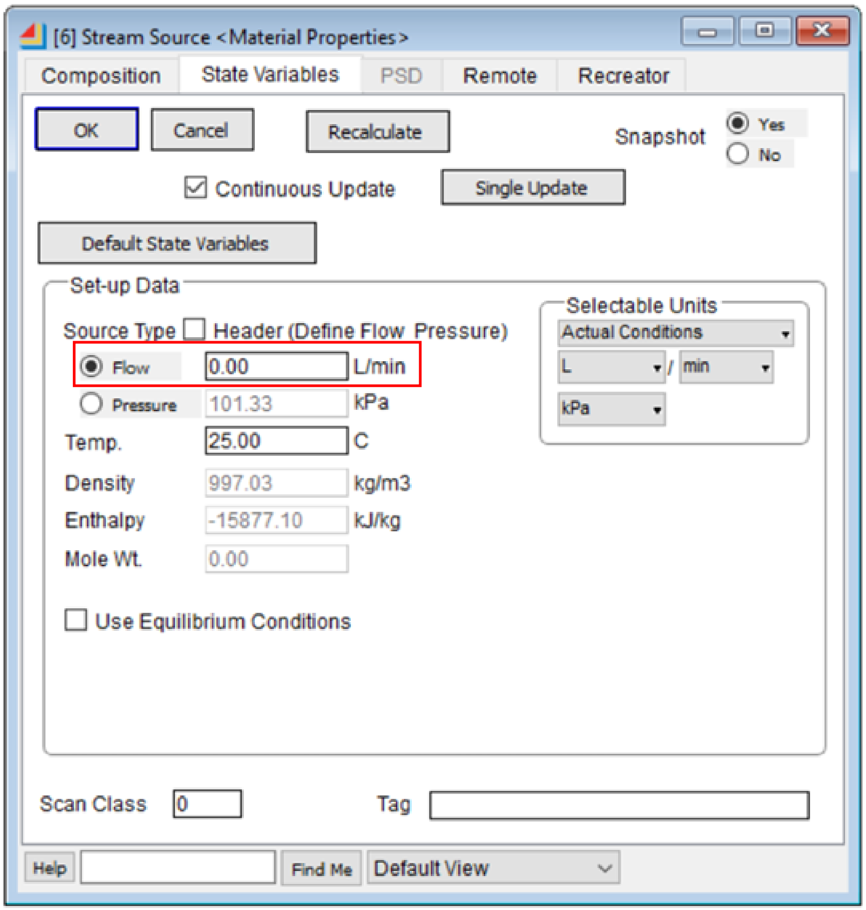


Figure 40: Stream Source Dialog Box Setting

Notice that the **Tank-Incompressible** level drops until it is empty. Let the simulation complete or stop it manually.

After completing this simulation, return the flow to *3000* L/min.

5.9 USING THE SNAPSHOT FUNCTION

Snapshot is a useful tool for saving various conditions in the model. Place the **Snapshot** object from the EXECUTIVES library onto the worksheet.

5.10 START THE SIMULATION

Set the simulation **End time** to *200000* (keeps the simulator running for an extended period). Run the simulation and set the **Slider** to *100* and let the **Tank-Incompressible** empty. Toggle the Switch to 0, and set the Slider to 0 (see Figure 41).

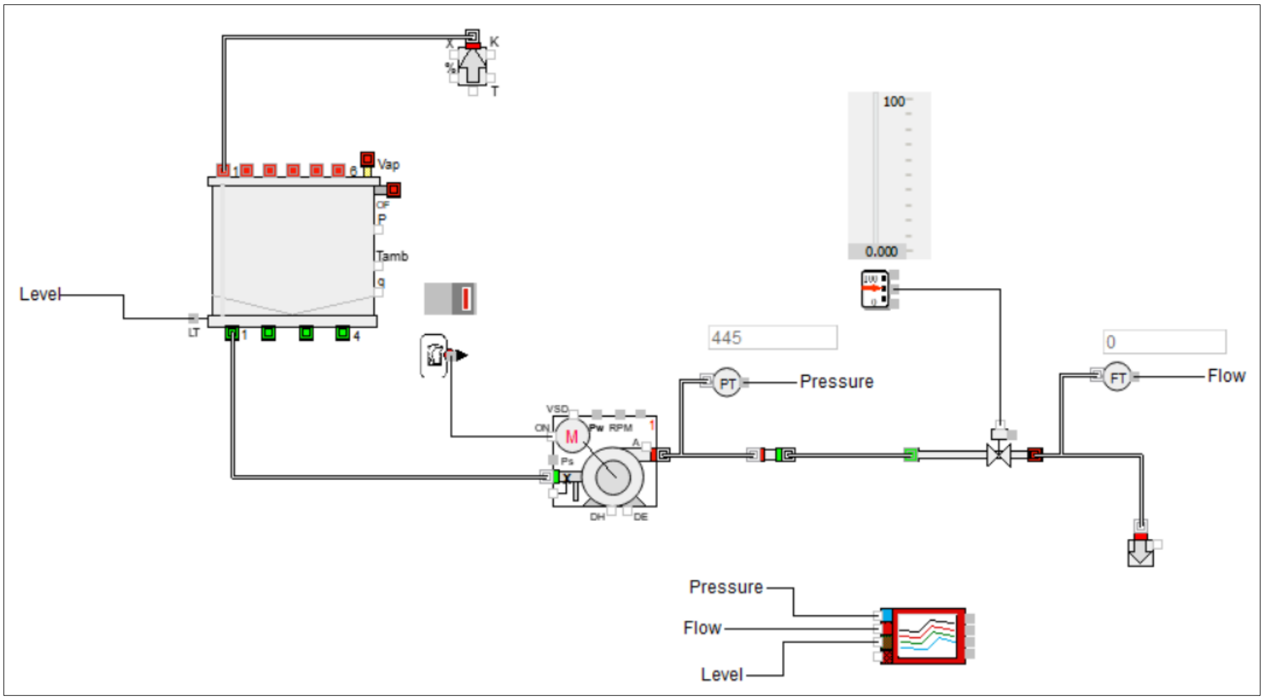


Figure 41: Switch/Slider Setting

5.11 SETTING SNAPSHOT SETTINGS

Open the **Snapshot** dialog box and click on the **Turn All 'Yes'** button. This ensures that the snapshot option for all objects on the worksheet is enabled. Go to **Storage Options** tab and Click on the **Store snapshot** button (make sure that your simulation is running). This will store your snapshot in the same mox file. (For a complete explanation of the varying formats of the **Snapshot** object, refer to the Snapshot description in the EXECUTIVES Library help text.) See Figure 42 for the **Snapshot** dialog box settings. Close the dialog box.

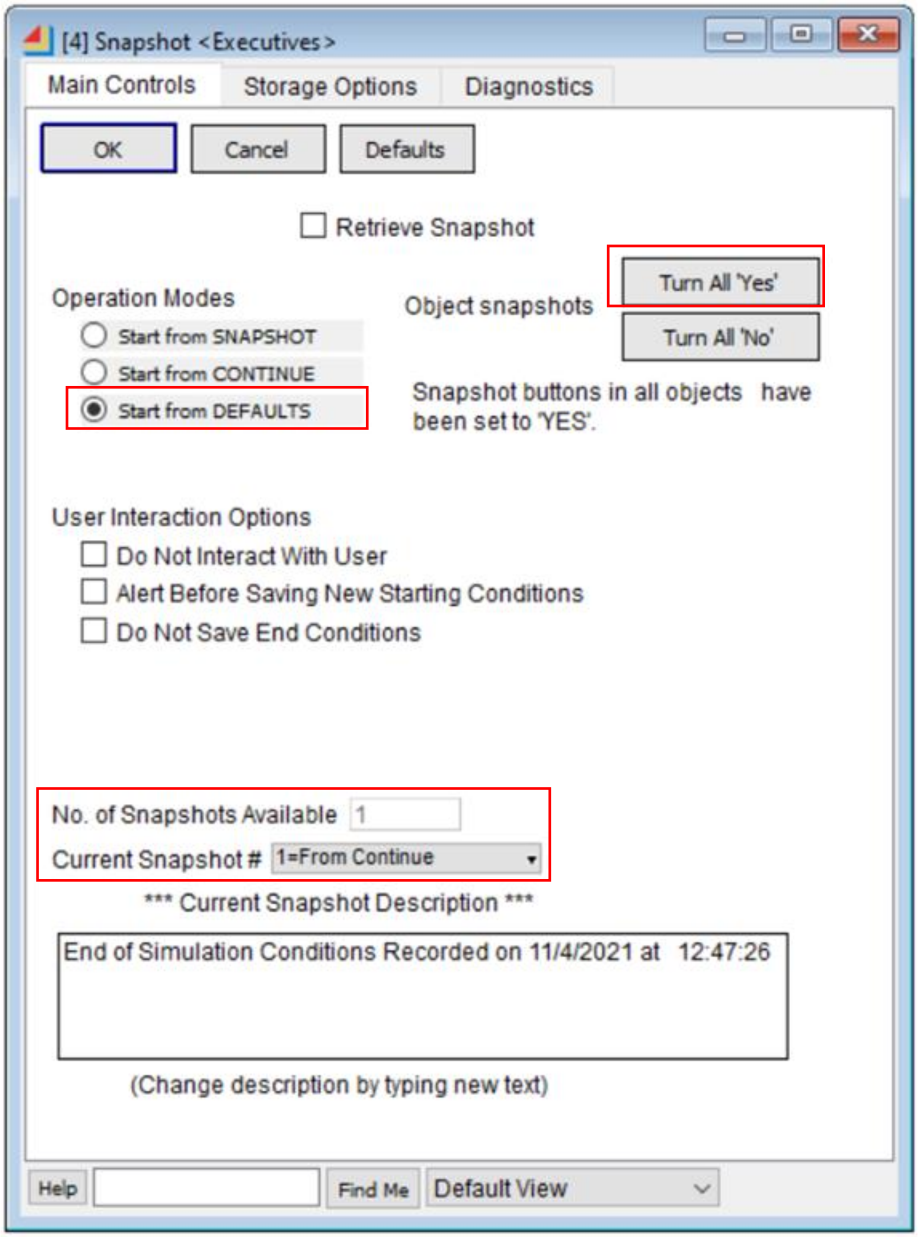


Figure 42: Snapshot Dialog Box Settings

5.12 RUNNING THE SIMULATION WITH A DIFFERENT SETTING

Click on the **Stop** button in the tool bar. Toggle the Switch to 1 and change the Slider to *100*.

Open the Snapshot dialog box and click on the **Start from SNAPSHOT** radio button. Run the simulation.

After snapshot is retrieved, the simulation begins. The **Switch** and **Slider** have returned to 0 as stored in the snapshot. As you now see, Snapshot can be used to initialize the model's conditions. If needed, different snapshots of the same model can be stored.

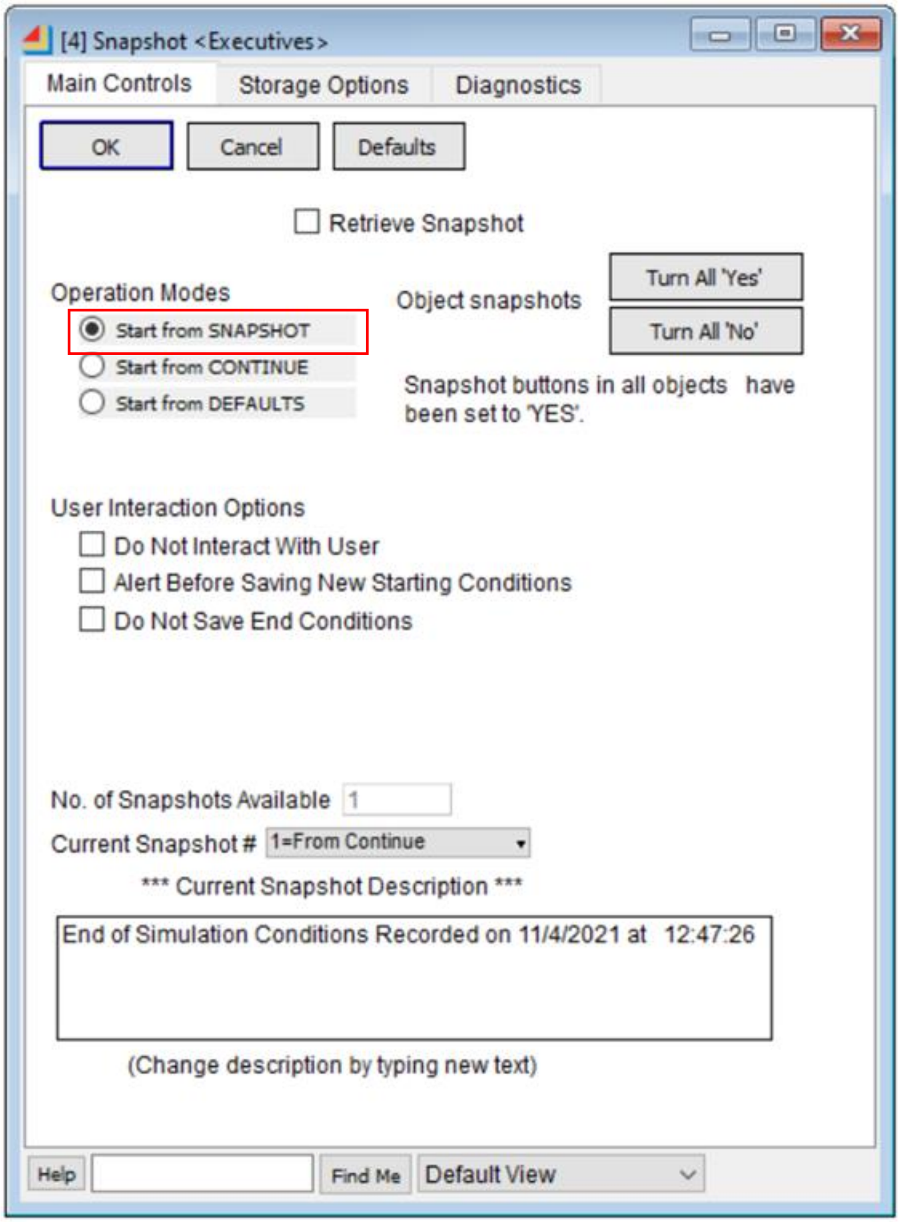


Figure 43: Transmitter-Pressure Boxes Clone Selected

5.13 RESETTING THE SNAPSHOT SETTINGS

Click on the **Stop** button in the toolbar. Open the **Snapshot** dialog box and click on the **Start from DEFAULTS** button. This provides the proper setting for the remaining sections of the tutorial. Close the dialog box.

This is the end of **Simulation Options** section. Select **Save Model** from under **File** on the menu bar and proceed.

SECTION 6. PID CONTROL

<u>Section Concepts:</u>

- Add a Control Scheme to the Model
- Tune the Controller to Optimize Performance

6.1 SAVING THE MODEL

Save the model with the new name *Standard Worksheet III*. If a dialog box opens that indicates “This file already exists. Replace existing file?” - click on the **Yes** button.

6.2 ADDITION OF CONTROL SCHEME TO THE MODEL

Open the ANALOG CONTROLS library. Place a **Controller PID** object on the worksheet from the Analog Controls library. Move it to the bottom right of the worksheet. The **Controller PID** object is used to model the function of a velocity-based PID controller. This **Controller PID** will control the level in the **Tank-Incompressible**.

6.3 DELETING THE SLIDER OBJECT

Delete the **Slider** object that is connected to the **Valve-Control** by selecting the **Slider** and pressing the delete key. Both the **Slider** object and the clone from the slider will be deleted. Create a label entitled *Valve* next to the scalar inlet of the **Valve-Control** (refer to Build a Model section for information on how to create a label.) Connect the label to the scalar inlet of the **Valve-Control**. Duplicate this label and connect it to the *Out* connector of the **Controller PID**. With this connection, the **Controller PID** can control the stem position in the **Valve-Control**.

6.4 CREATING AN AUTOMATIC LEVEL CONNECTION

Create a label entitled *Level* and connect it to the PV inlet on the **Controller PID**. This communicates with the *Level* label connected to the Level Transmitter on the **Tank-Incompressible.** The **Controller PID** can now adjust the level in the **Tank-Incompressible** automatically (see Figure 44).

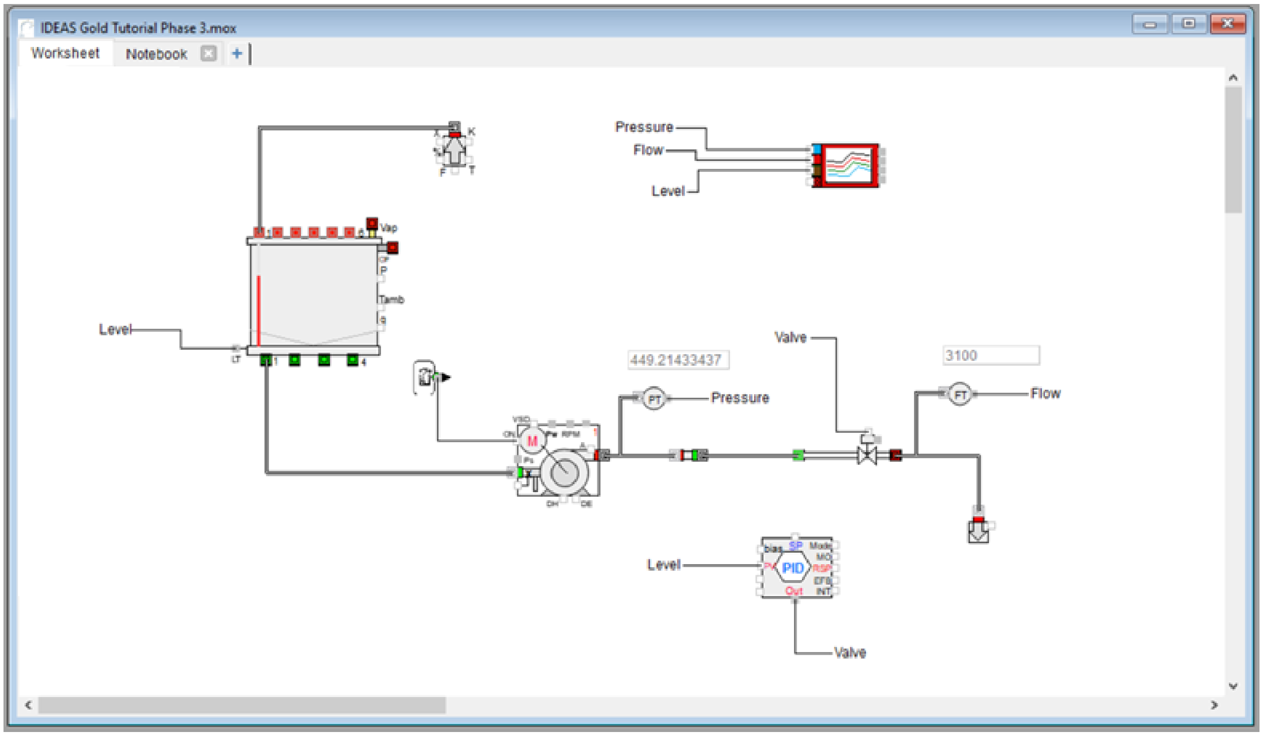


Figure 44: Controller PID Placement

6.5 SETTING THE FLOW INTO THE TANK

Open the **Stream Source** dialog box and type *3100* in the **Flow** field. Close the dialog box.

6.6 SETTING CONTINUOUS UPDATE OPTIONS

Open the **Controller PID** dialog box and check **Continuous Update**. With the **Controller PID** dialog box open, select **Run Simulation** from **Run** on the menu bar. The level being monitored by the **Controller PID** is going up as indicated by the PV display field at the top of the dialog box. The level of the tank increased because the **Controller PID** is in manual mode and the output to the Valve-Control is "0" as shown in the **Output** field. An output of "0" means the Valve-Control is shut and there is no flow out of the **Tank-Incompressible** (see Figure 45). Pause the simulation.

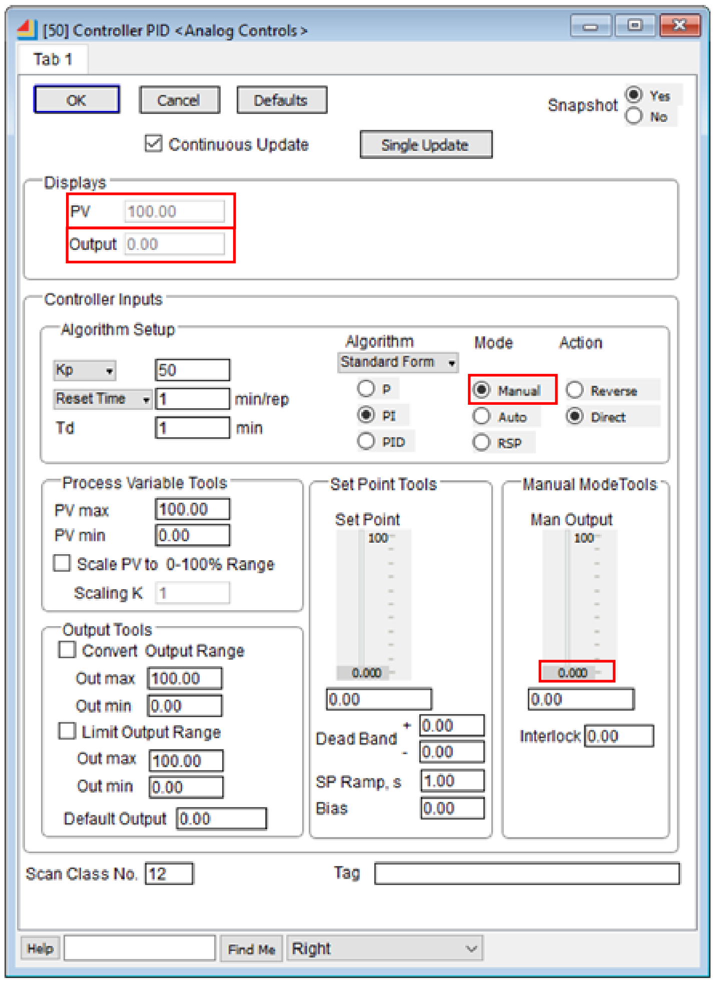


Figure 45: PV Level Changes

6.7 SETTING THE CONTROLLER PID ACTION POLARITY

Before putting the **Controller PID** in automatic mode, let us first set the **Controller PID** action to the proper polarity. Click on the **Direct** radio button under the **Action** label. Since the **Controller PID** must respond to an "increasing" level with an "increasing" output (i.e., increasing valve stem position and the resulting flow out of the tank), the action is direct. If the flow into the **Tank-Incompressible** were being controlled, the Controller PID action would be reversed to respond to an increasing level by decreasing its output (i.e., inlet valve stem position). Adjust the Set Point of the **Controller PID** to *50* using either the dialog box entry field or the slider. Put the **Controller PID** into **Auto** (automatic) mode by clicking on the **Auto** radio button under the **Mode** label (see Figure 46). Click on **OK** to close the dialog box.

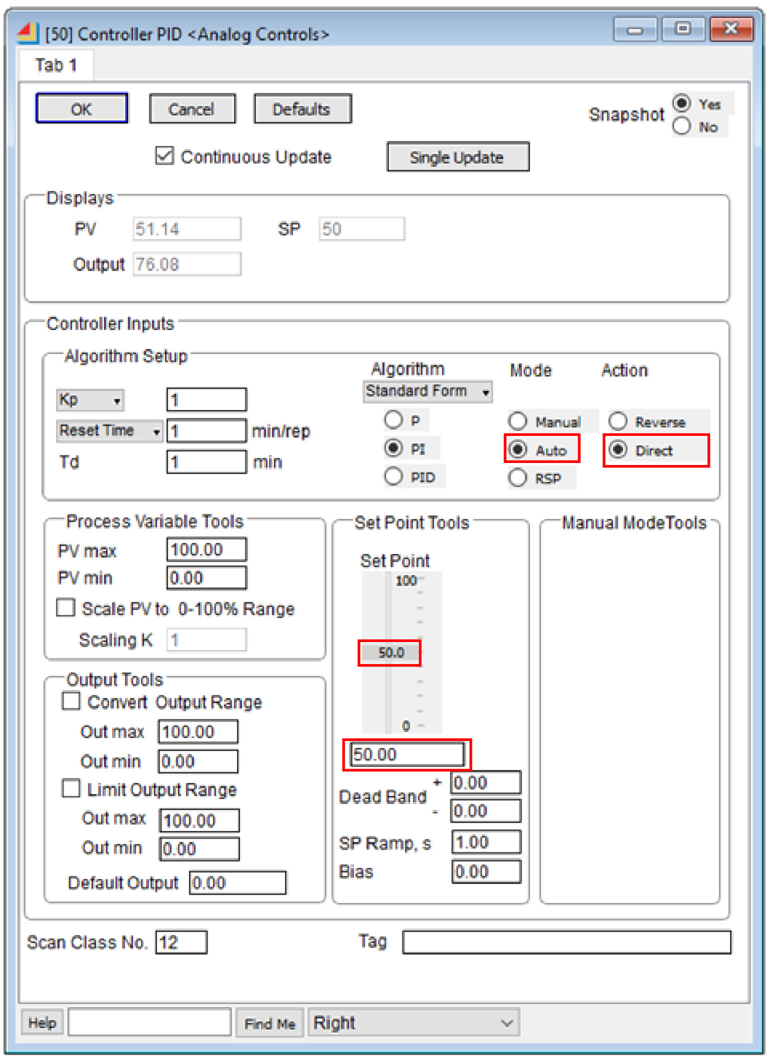


Figure 46: Controller PID Settings

6.8 STARTING THE PUMP-CENTRIFUGAL

Click on the right side of the cloned **Switch** button to turn ON the **Pump-Centrifugal**. Open the **Controller PID** dialog box and notice that the **Tank-Incompressible** level (PV) is beginning to drop because the flow out of the **Tank-Incompressible** is above the 3100 L/min going into it. As the level approaches the 50% set point, the Valve-Control output begins to decrease. The **Controller PID** does not respond fast enough to the changing conditions, and the level drops below fifty percent (50%). This is because the **Controller PID** tuning is not yet optimized (see Figure 47).

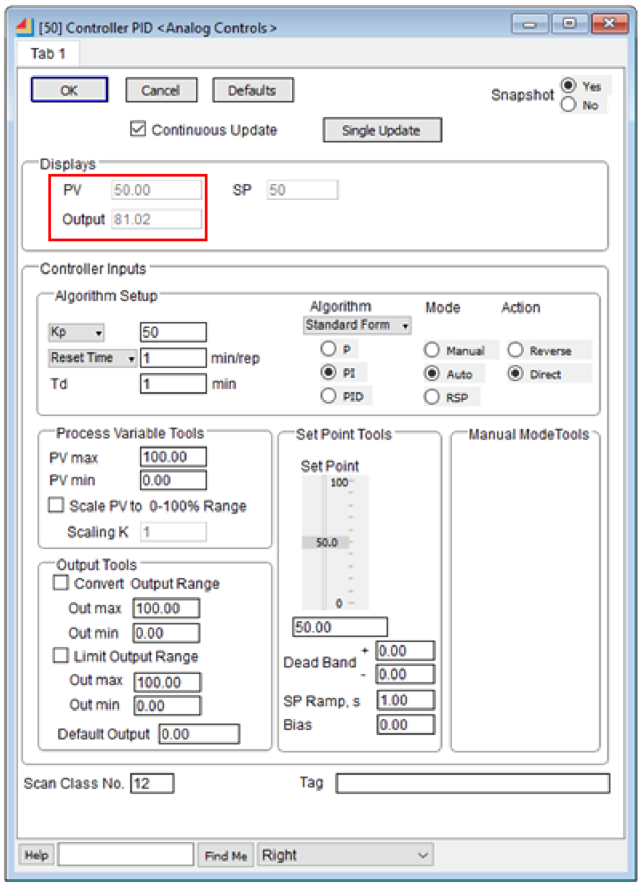


Figure 47: Controller PID Changes

6.9 TUNING THE CONTROLLER PID FOR BETTER PERFORMANCE

Level control loops, depending on tank size and residence time, may require a higher gain with little or no integral (reset) action. Change the Gain (Kp) in the **Controller PID** dialog box from 1 to 50. Restart the simulation and observe the improvement in the level control response. The **Tank-Incompressible**'s level drops below 50% for only a few time steps as opposed to the oscillation of the level when the gain was 1.

Vary the set point with the slider (suppose 70) in the Controller PID dialog box and note the level control response. The system responds relatively smoothly to changes in the set point (see Figure 48).

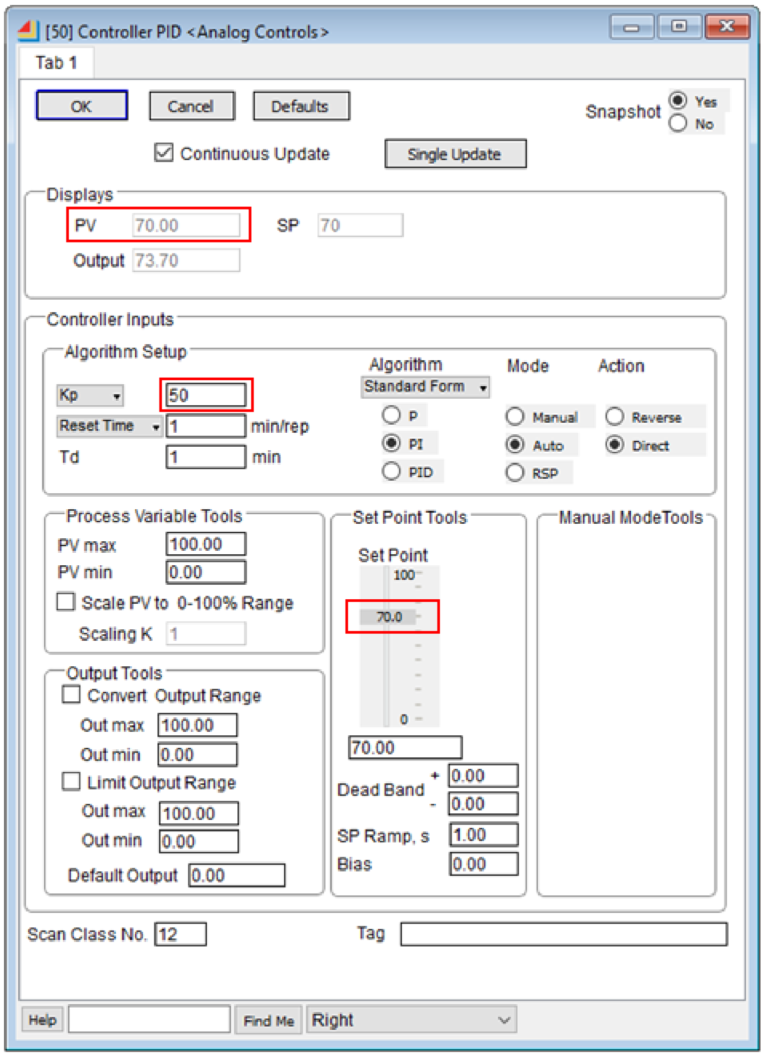


Figure 48: Controller PID Set Point Changed to 70

To further examine the control response, open the **Plotter, I/O-scan** window and view the plots of pressure, flow, and level.

6.10 ENDING THE SIMULATION

Click on the **Stop** button in the toolbar.

This is the end of IDEAS Gold Tutorial. Select **Save** Model from under **File** on the menu bar.

We hope you have found this material helpful in learning how to use the IDEAS simulation software. Another tutorial, the IDEAS Bronze tutorial, is also available to assist you in your learning.

This tutorial is meant to provide a basic introduction to the IDEAS Simulation Software.

ANDRITZ Inc. also offers standard and customized software training classes.
Please contact us to inquire about attending a classroom session.

For more information, contact ANDRITZ Inc.:

Automation-Sales
automation-sales@andritz.com

Software Training and Support:
Kelly Rogers
m: +1 (470) 223 9349
f: +1 (770) 640 2557
simulation.support@andritz.com
andritz.com

User Group
IDEAS LinkedIn User group

Appendix A Introduction to Excess Properties

<u>Section Concepts:</u>

- Modify the System to Carry Components other than Water
- Examine Excess Properties

A.1 Saving the Model

Save the model as *Standard Worksheet IV*. If a dialog box opens that indicates “This file already exists. Replace existing file?” - click **Yes**.

A.2 Adding a Component to the System

Open the **Material Properties** dialog box. In the Components tab, click on the **Load IDEAS Components** button. It will open IDEAS components window. Navigate through IDEAS Components window and highlight the *Sulfuric Acid (H2SO4)* component and click on the **Add to Selection** button or double-click on the component name to load *Sulfuric Acid* into the **Material Properties** object. Once the component has been loaded, the Stream Component table should have three components listed with their corresponding names in the **Stream Component Assignments** table. Click on **OK** button to accept the selection. Sulfuric Acid (H_2SO_4) is now the second component in the stream array under **Components** tab, which will be available throughout the model (see Figure 49).

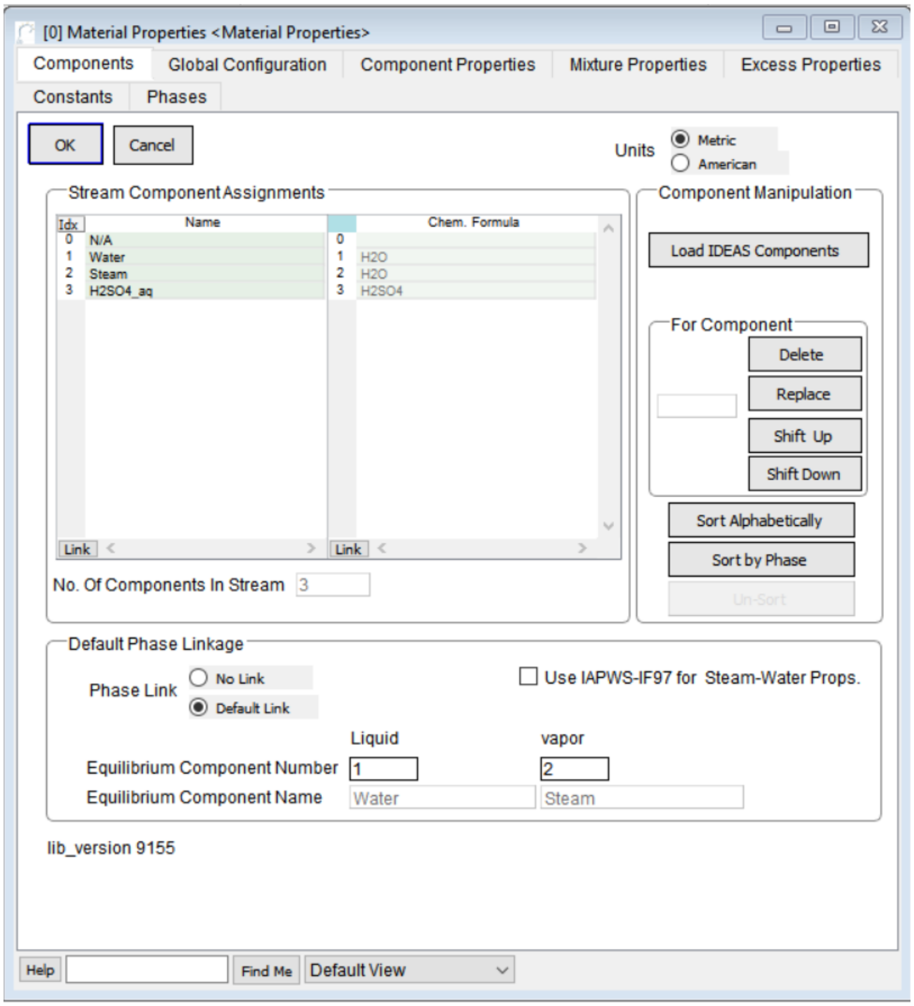


Figure 49: Material Properties Dialog Box Entry

A.3 Examining Excess Properties

The addition of H_2SO_4 to the system presents an opportunity to examine the phenomena of volume of mixing and heat of mixing. Click on the **Component Properties** tab of the **Material Properties** dialog box. Type *1* in the Component Index field and *1* in the Property Number field. Type *25* in the box under the X1 heading (specifies the temperature as 25°C). By looking under the Available Materials and Properties heading at the bottom of the box, you can see that Component 1 (Water) and Property 1 (Density as function of Temperature) have now been specified.

A.4 Calculating Material Properties

Click the **Calculate** button. The density of component 1 (water) as a function of temperature at 25°C is now shown in the box under the heading Y. This density is 997.0261306 kg/m³ (see Figure 50).

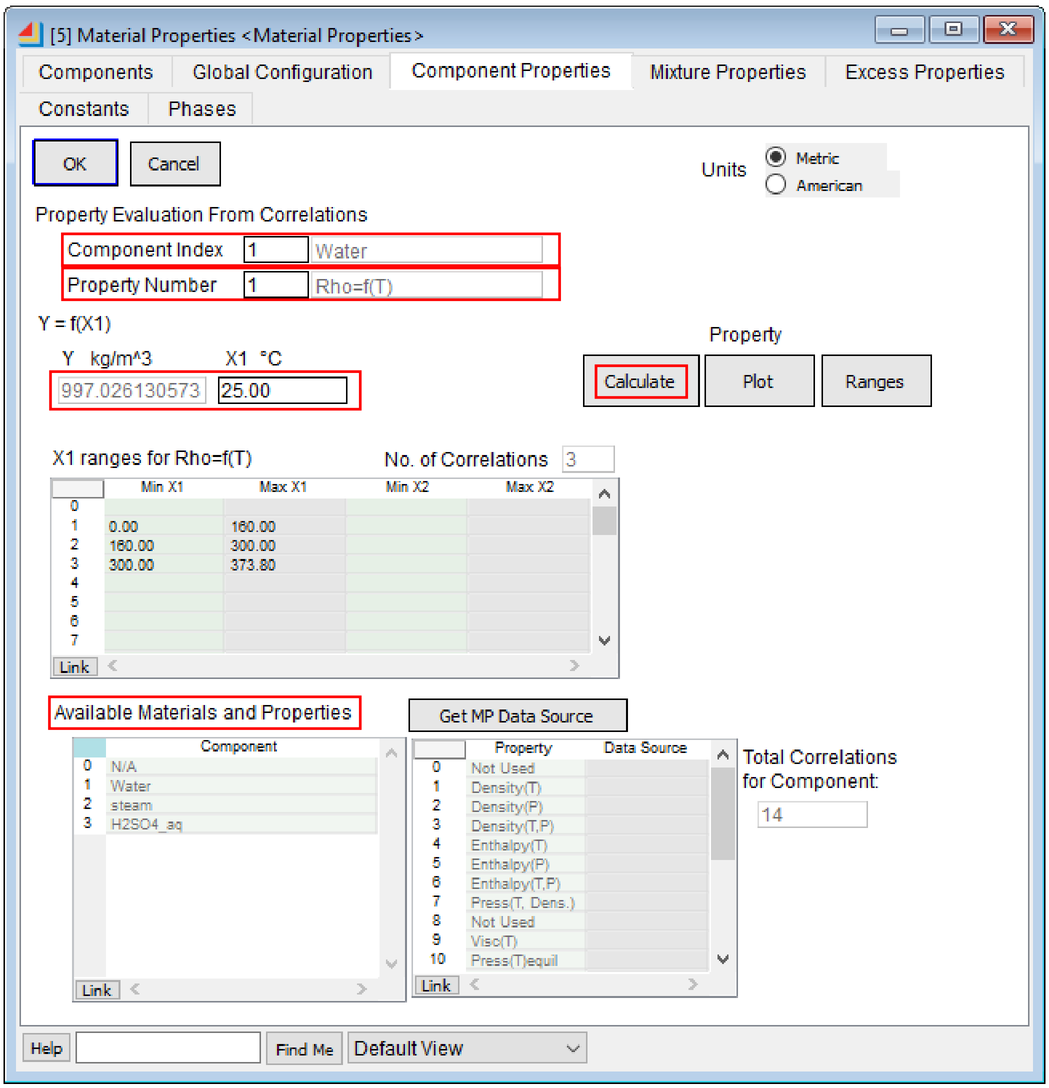


Figure 50: Calculation of Density (Water) as a Function of Temperature at 25°C

Repeat this process to find the density of Sulfuric Acid at 25°C. Type *2* into the Component Number field and press the **Calculate** button. The density of component 2 (Sulfuric Acid) as a function of temperature at 25°C is now shown in the box under the heading Y. This density is 1862.49 kg/m³ (see Figure 51).

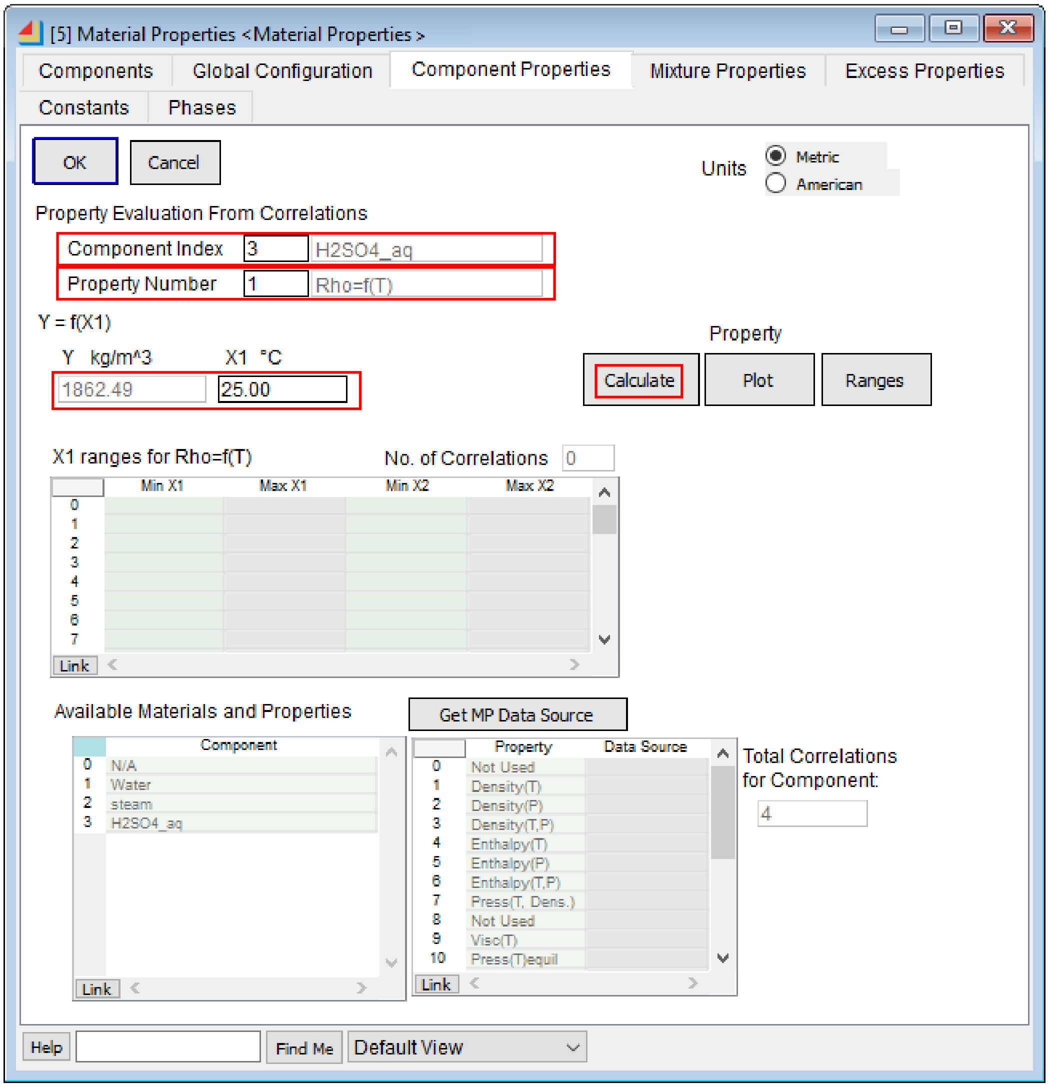


Figure 51: Calculation of Density of (H_2SO_4) as a Function of Temperature at 25°C

What would happen if two streams (at the same temperature) of equal mass of water and sulfuric acid were mixed? Would the density of the combined stream be equal to the average of the two densities? Would the volume of the combined stream be equal to the additive volume of the two streams? Would the temperature of the combined stream be the same as the temperature of the two pure streams? IDEAS can find the answers to these questions.

The **Material Properties** object can quickly and easily calculate the properties of a mixture. Go to the **Mixture Properties** tab. Make sure the **% Mass Composition** radio button is set. Type *50* into the % table for both Water and H_2SO_4. Enter *25°C* as the temperature. Under the **Calculate** button, select

Density (T) from the dropdown menu. Press the **Calculate** button. The density of a 50 mass % water/50 mass % H_2SO_4 stream at 25°C is calculated. The density of the mixture is 1298.79 kg/m³ (see Figure 52).

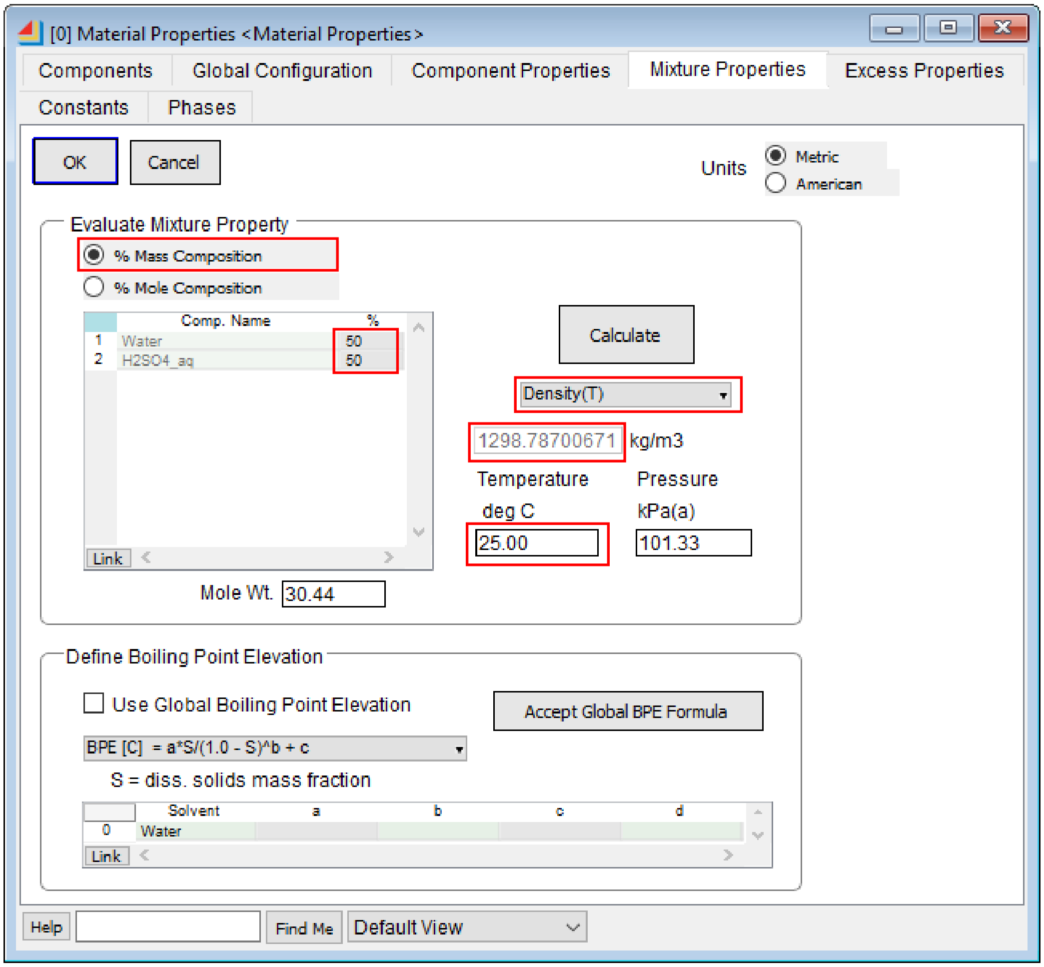


Figure 52: Density Calculation of 50% Water / 50% H_2SO_4 Mixture, without Volume of Mixing

Thus far, we had not considered volume of mixing in the density calculation. Once we do so, we will eventually obtain the density as shown in Figure 53:

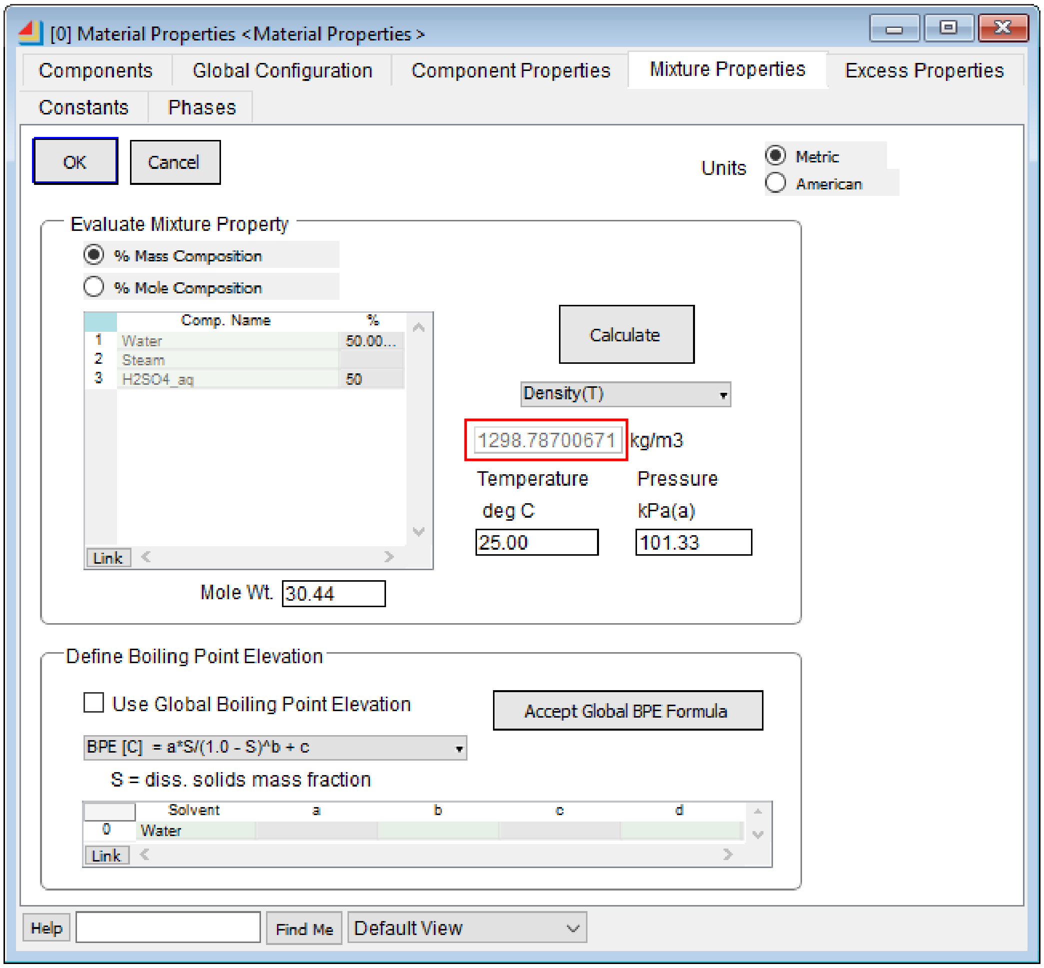


Figure 53: Density Calculation of 50% Water / 50% H_2SO_4 Mixture, with Volume of Mixing

The density of pure water at 25°C is 997.02 kg/m^3, and the density of pure sulfuric acid at 25°C is 1862.49 kg/m^3. 100 kg of water yields 0.10030 m^3 of volume. 100 kg of sulfuric acid yields 0.05369 m^3 of volume. Assuming additive volumes, i.e., no volume of mixing, the total volume is 0.15399 m^3. Dividing 200 kg by 0.15369 m^3 gives a mixture density of 1298.79 kg/m^3. From this calculation, you can see that IDEAS, thus far, is performing ideal density calculations for a mixture of water and sulfuric acid. However, IDEAS is also capable of calculating density while considering Volume of Mixing (excess specific gravity).

A.5 Setting Excess Properties Options

Click on the **Excess Properties** tab. Enter *1* (Water) into the **Solvent Index** box. Check the box in the table next to H_2SO_4, which makes H_2SO_4 the solute (see Figure 54). Click **Accept Stream Definition** and click **Set Local Excess Properties**. IDEAS will now use the excess properties for the mixture of water and sulfuric acid when calculating the density and enthalpy.

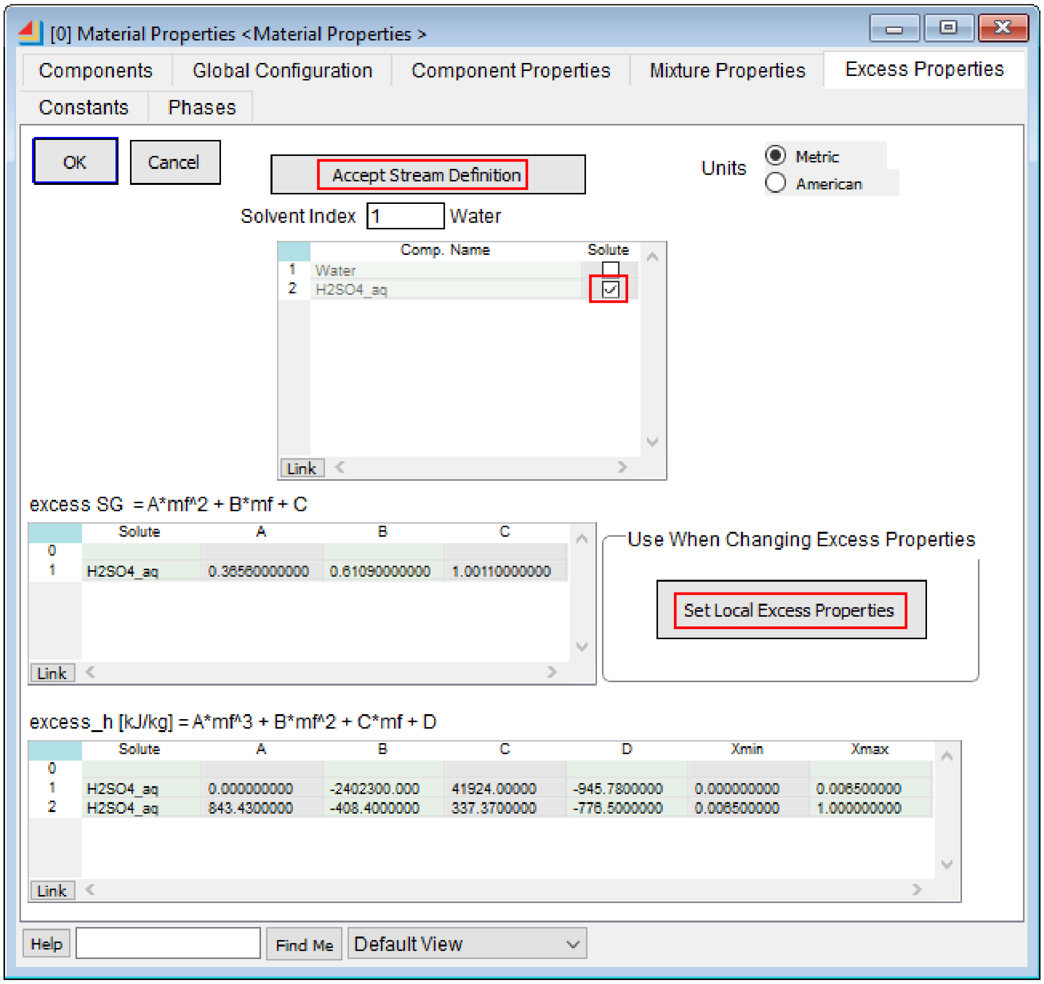


Figure 54: Excess Properties Tab

A.6 Re-Checking the Mixture Density

Now go back to the **Mixture Properties** tab, press **Calculate** button and recheck the density of the mixture. The density is 1393.793 kg/m^3. IDEAS is now computing density with consideration for the excess volume calculation being performed as is appropriate for this mixture.

IDEAS can calculate mixture properties while considering both heat of mixing and volume of mixing. Many mixtures act as ideal or near-ideal mixtures with respect to heat of mixing and volume of mixing. However, some mixtures have significant effects of these phenomena and should be considered for greater accuracy. Some components within IDEAS, including sulfuric acid, have coefficient values for these effects already present. Next, we will explore how these effects may appear in a model.

Note: This section will include the use of IDEAS macro (or Bronze) objects. The Gold version of IDEAS includes the functionality of both IDEAS Bronze and IDEAS Silver. In fact, most models will contain a mixture of both dynamic and steady state (macro) objects.

A.7 Starting the Model

Place two **Stream Source** objects from the MATERIAL PROPERTIES library onto the worksheet to the right of the rest of the objects. Open one of the new Stream Source's dialog boxes. Make sure the **% Mass Composition** radio button is clicked. Enter *100* into the field in the **% Mass** table to the right of number 1 (see Figure 55).

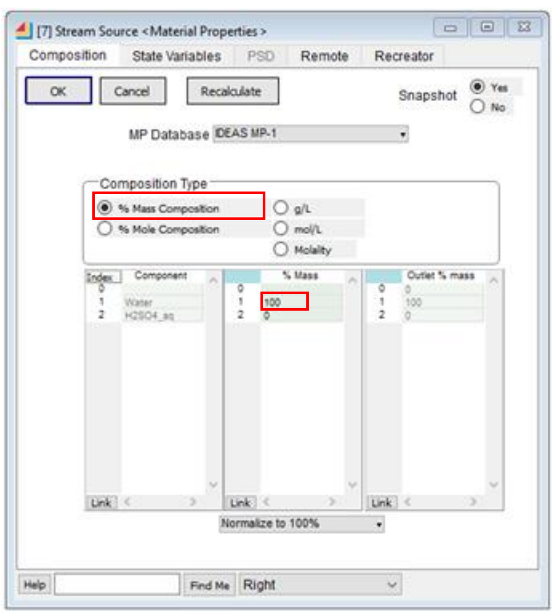


Figure 55: Stream Source 1 Dialog Box Entry, Composition Tab

A.8 Setting State Variables Options for Stream 1

Click on the State Variables tab. Turn on **Continuous Update**. Click on the **Flow** radio button and enter a flow rate of 100 kg/s (change the units if necessary by using the dropdown menus). Enter 25 into the Temp. field. This stream is now specified as pure water at 25°C with a flow rate of 100 kg/s (see Figure 56).

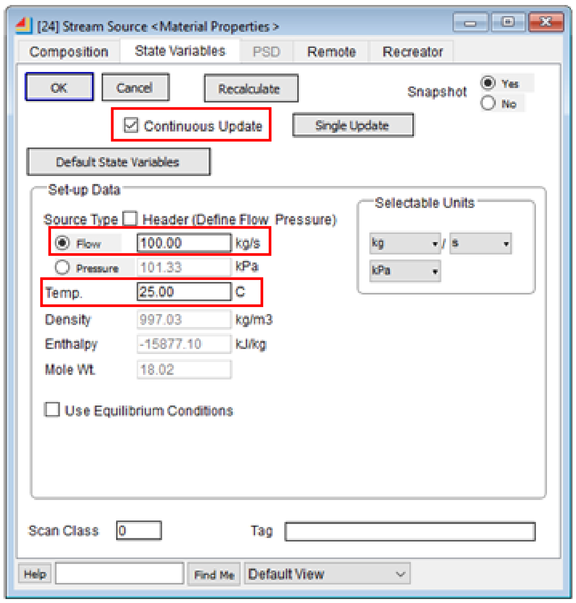


Figure 56: Stream Source 1 Dialog Box Entry, State Variables Tab

A.9 Setting State Variables Options for Stream 2

Repeat this process for the second Stream Source, but enter *100* into the field in the **% Mass** table to the right of number 2 (composition tab) and specify 0% in row 1 as needed to specify the stream as 100% sulfuric acid (see Figure 57).

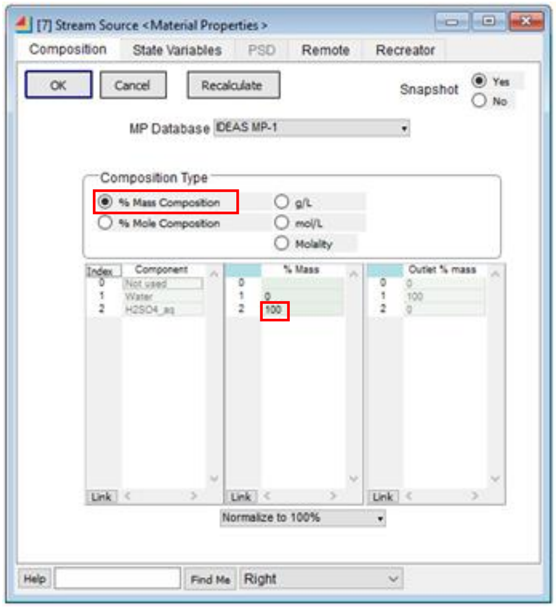


Figure 57: Stream Source 2 Dialog Box Entry, Composition Tab

A.10 Setting Continuous Update

Click on the **State Variables** tab. Turn on **Continuous Update**. Click on the **Flow** radio button and enter a flow rate of *100 kg/s* (change the units if necessary by using the dropdown menus). Enter *25* into the temperature field. This stream is now specified as pure sulfuric acid at 25°C with a flow rate of 100 kg/s (see Figure 58).

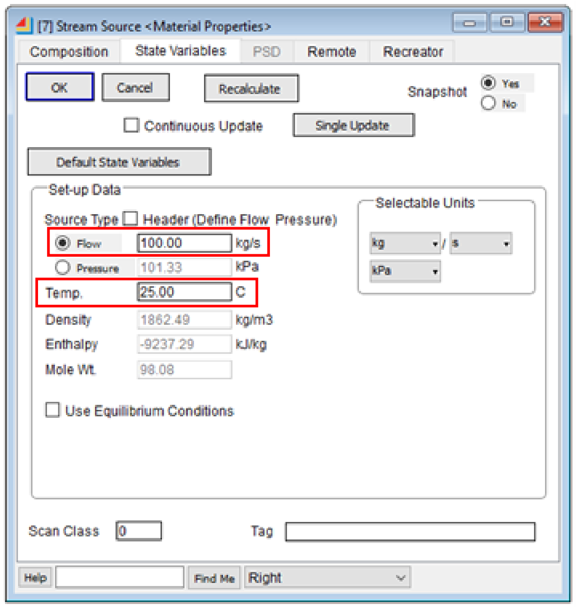


Figure 58: Stream Source 2 Dialog Box Entry, State Variables Tab

A.11 Connecting the New Stream Source Objects

Open the MACRO PRIMITIVES library. Place a **Mixer** object onto the worksheet to the right of the new **Stream Source** objects. Connect the outlet of the first Stream Source to inlet 1 of the Mixer. Connect the outlet of the second Stream Source to inlet 2 of the **Mixer** (see Figure 59). Open the **Mixer** dialog box and turn on **Continuous Update**. Close the dialog box.

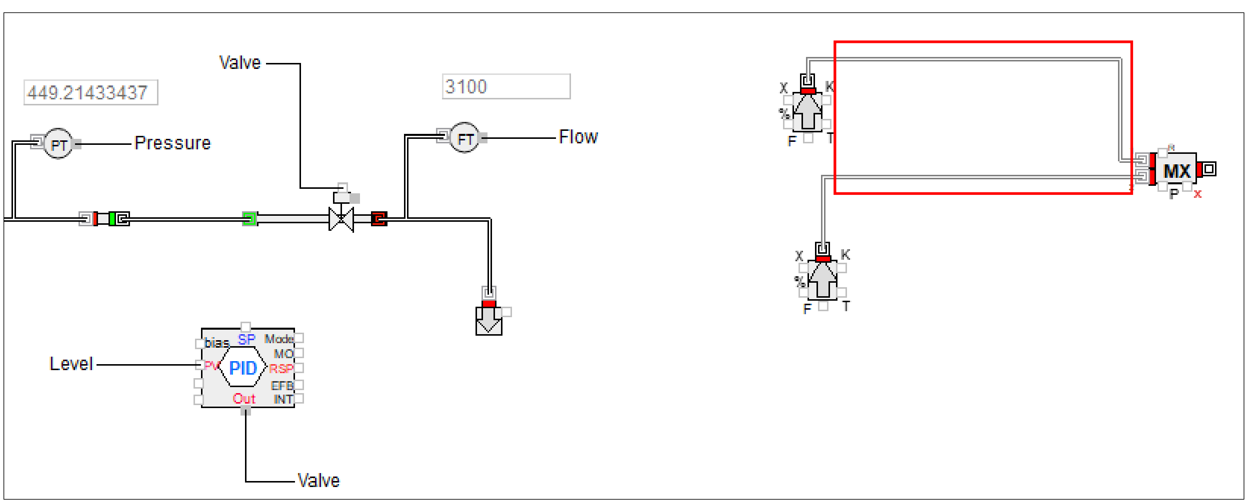


Figure 59: Connection of New Stream Source Objects to Mixer

A.12 Connecting Transmitter-Flow Objects

Place two **Transmitter-Flow** objects onto the worksheet and connect one of the transmitters to inlet 1 of the **Mixer**. Connect the other transmitter to inlet 2 of the **Mixer** (see Figure 60). Check **Continuous Update** for each **Transmitter-Flow**. In their respective Inputs tab, change the units to Liters per minute. Close each of the dialog boxes.

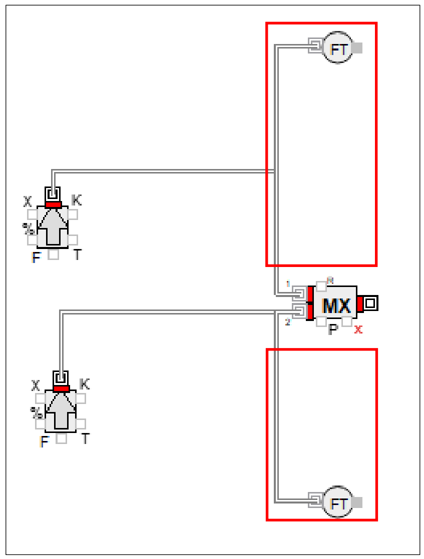


Figure 60: Connection of Transmitter-Flow Objects to the Mixer

A.13 Connecting Transmitter-Flow Outlets to Add_c Inlets

Place an **Add_c** object from the TOOLS library onto the worksheet. This object adds the values of the two inputs, multiplies their sum by the value in the dialog entry labeled ***Multiply output by***, and passes this result to the output connector (see Figure 61).

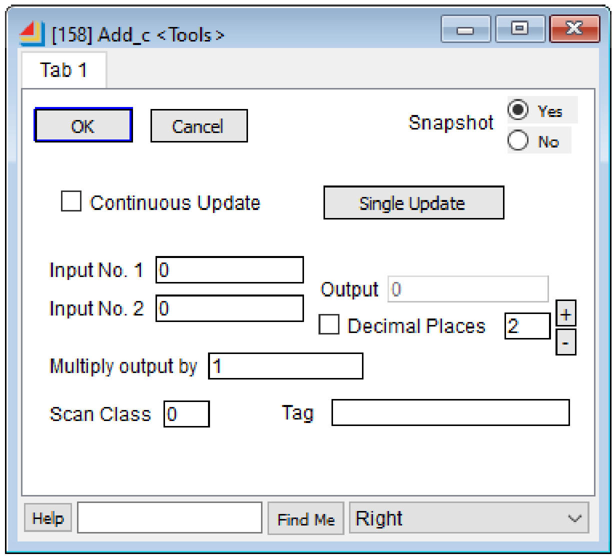


Figure 61: Add_c Dialog Box Parameters

Now, connect the outlet from each **Transmitter-Flow** object to one inlet of the **Add_c** object (see Figure 62). Open the **Add_c** dialog box and turn on **Continuous Update**.

Close the dialog box.

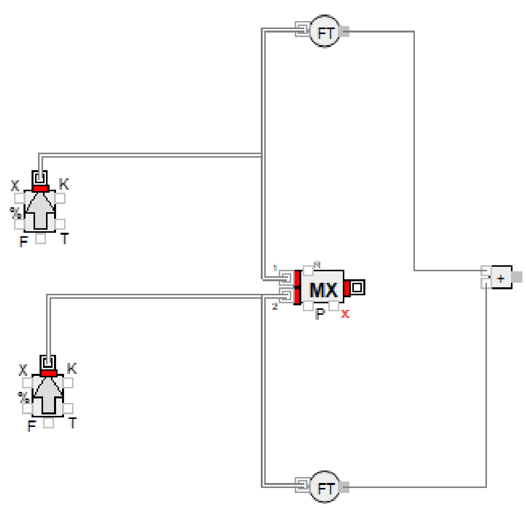


Figure 62: Connection of Transmitter-Flow outlets to Add_c inlet

A.14 Connecting the Mixer Outlet to the Sink Inlet

Place a **Sink** from the MATERIAL PROPERTIES library onto the worksheet and connect the outlet from the **Mixer** to the inlet of the **Sink** (see Figure 63). For the **Sink**, turn on **Continuous Update**. Close the dialog box.

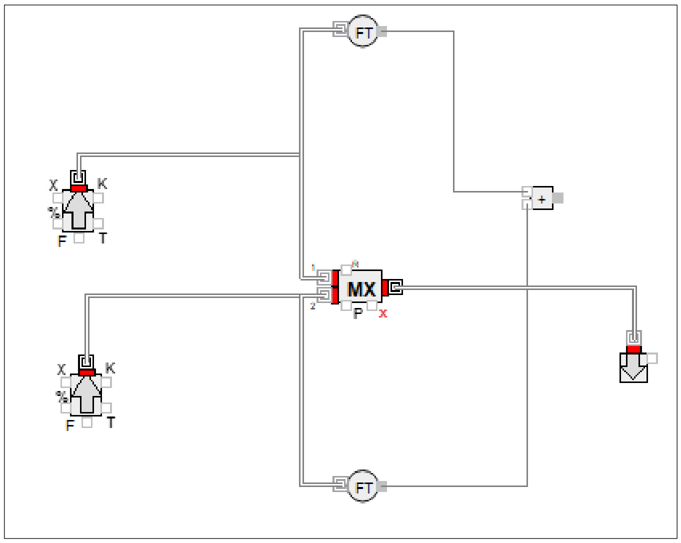


Figure 63: Connection of Mixer Outlet to Sink Inlet

A.15 Using the S-Click Feature

One way to monitor objects is by using the **S-Click** feature. Run the simulation. As the simulation is running, hold down the **S** key on the keyboard and click on an object with the mouse. (The **S-click** feature works for stream objects while the simulation is running. It does not function for H-blocks.) A table will appear that shows the state variables flow, temperature, and pressure into and out of an object and the flow and energy balance for the object (see Figure 64). You can also right click in the S-Click table to change the display from **Flow Parameters** to **% Mass/Mole Composition**.

A.16 Performing an S-Click on the Sink

As the simulation is running, perform an S-click on the **Sink**. Notice the temperature of the stream is approximately 129.9°C (If only liquid water is chosen without a link to steam. If the phase link remains and the outlet pressure is set high enough, there will be no steam and you will see roughly the same temperature. Otherwise, you will see an outlet temperature of 100°C with steam as well.) Obviously, there is some Heat of Mixing. To further examine the excess properties, we will cool the outlet stream from the **Mixer**.

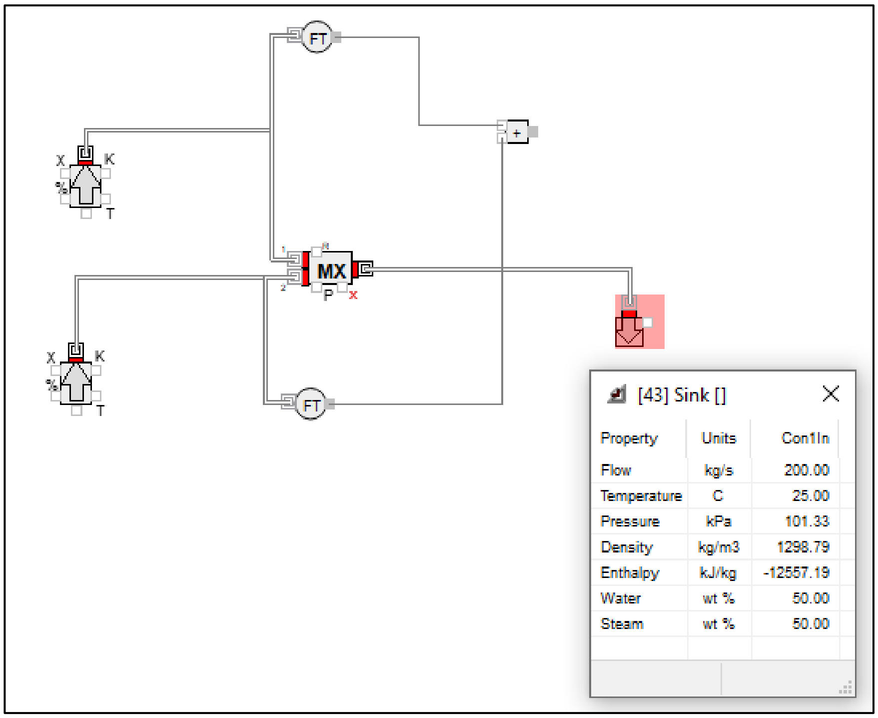


Figure 64: S-Click on Sink

A.17 Connecting the Heater to the Transmitter Flow

Stop the simulation. Open the MACRO UNIT OPS library. Place a **Heater** object from the MACRO UNIT OPS library onto the worksheet. Delete the connection between the **Mixer** and the **Sink**. Connect the outlet of the **Mixer** to the inlet of the **Heater** and the outlet of the **Heater** to the inlet of the **Sink**. Also, place a **Transmitter-Flow** object from the TRANSMITTERS library onto the worksheet and connect the outlet of the **Heater** to the inlet of the **Transmitter-Flow** (see Figure 65).

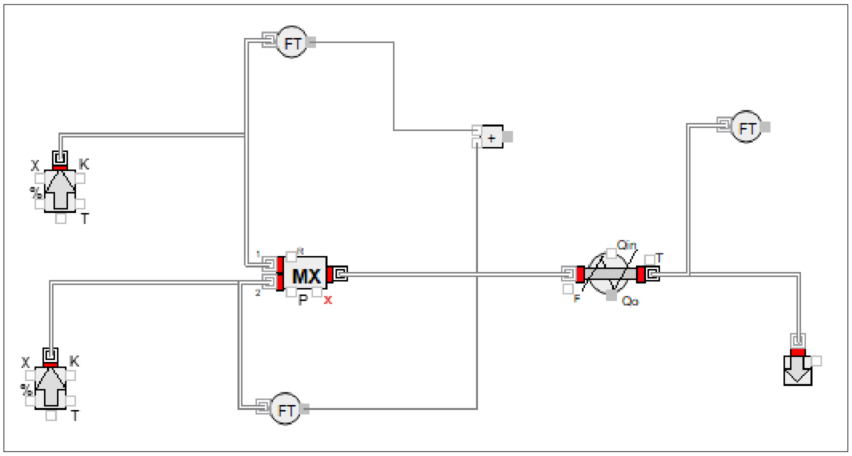


Figure 65: Connection of Heater and Transmitter-Flow

A.18 Setting Options for the Heater

Open the **Heater** dialog box and turn on **Continuous Update**. Click on the Inputs tab and select the **Outlet Temperature** radio button. Type *25* in the *Outlet Temperature* field. Close the dialog box.

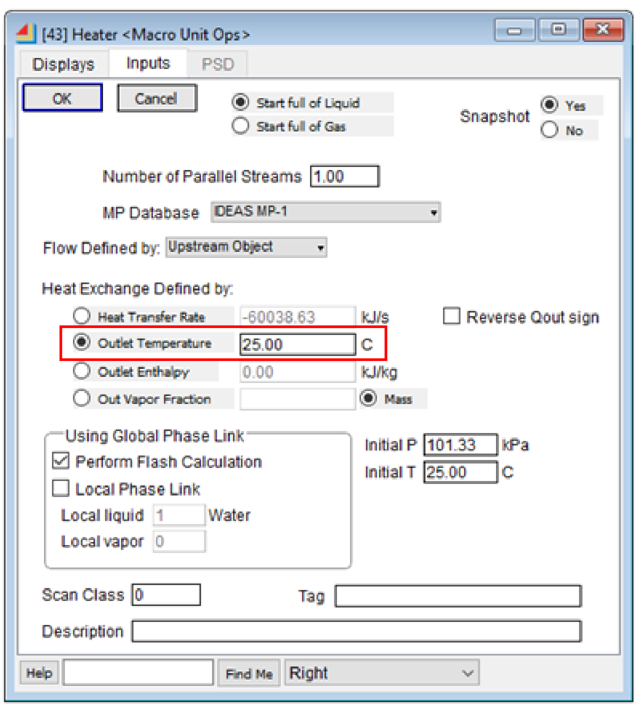


Figure 66: Heater Dialog Box Input Parameters

Open the **Transmitter-Flow** dialog box and turn on **Continuous Update**. Click the **Inputs** tab and change the units to *L/min*. Close the dialog box.

A.19 Observing Flow Rates

Run the simulation. Open the **Add_c** dialog box and the last **Transmitter-Flow** dialog box. Find the flow rates for each object. Notice the difference in the flow. 9060 L/min is entering the mixer (two pure streams) and only 8609 L/min is exiting the heater. We are comparing the mixer inlets with the heater outlet because each stream is at 25ºC, and therefore, there is no volume difference caused by a temperature difference. The volumetric flow rate difference (9060 vs. 8609) is caused by the Volume of Mixing phenomenon. The actual density of this mixture is greater than its ideal mixture density, which means the volume of the exiting mixture stream is smaller than the combined volume of the pure streams.

A.20 Perform an S-click on the Sink

Perform an S-click on the Sink. Notice that the flow rate is 200 kg/s, which is the same amount that is entering the mixer.

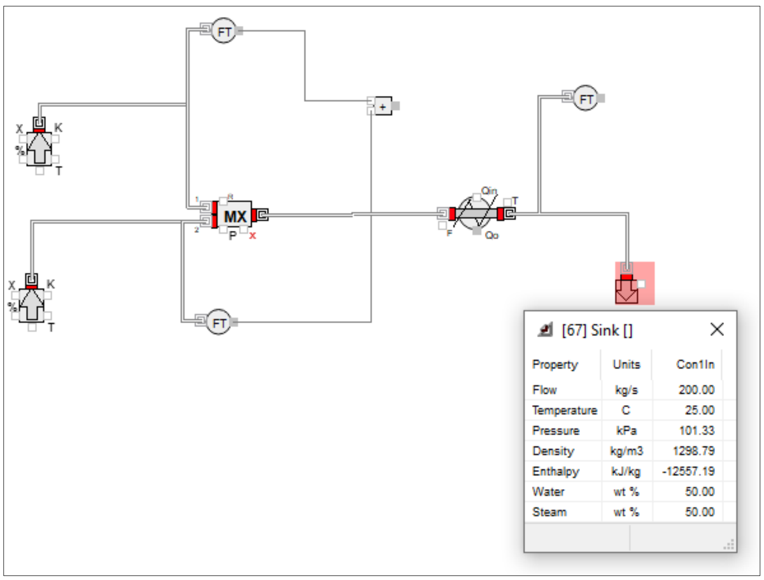


Figure 67: S-Click on Sink

Open the **Heater** dialog box and find the density in the **Displays** tab. The density of the combined stream at 25°C is 1393.79 kg/m^3, which agrees with the earlier calculations performed in the **Material Properties** object. IDEAS is performing volume of mixing calculations.

Save the model.