

Break Modeling Exercise

OBJECTIVES

- Review of break modeling best practices
- Break model validation

OVERVIEW OF STEPS

1. Preliminary Setup (Open the Model)
2. Add 3 inch Cold Leg Break to the PWR Model
3. Configure Break Valve to open on SBLOCA Trip
4. Run SNAP 'Model Check' and Fix Any Errors
5. Run a Short SBLOCA Simulation to Test the Break
6. Check Results. Make Corrections if Necessary.
7. Rerun the short SBLOCA Simulation and Recheck the Results.

STEP 1. PRELIMINARY SETUP (OPEN THE MODEL)

Open the 'Day4/Morning/PWR4_Break_Model' folder and double click on 'PWR4-Break.med' to open the PWR model.


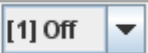


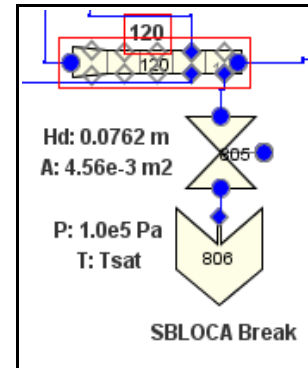
The “Exercise Key” included in the workbook may be useful to help locate the various parts of the SNAP Model Editor that are referred to in this exercise.

STEP 2. ADD 3 INCH COLD LEG BREAK TO THE PWR MODEL




Step Summary

1. Add a 3 inch (0.0762 m) break to the bottom of Loop 1 Cold Leg.

- Use a VALVE  with zero cells and zero length (an SJC valve) to model the break.
- Connect the valve to PIPE 120 cell 1.
- Set the component number to 805 and configure the valve parameters.
- Set Internal Loss Model  [1] Off and set the 'Form Loss Table' values for 0 (closed) and 1 (fully open) valve positions to 1E-6 (a negligible K loss factor).
- Orient the valve downward and enable the 'Offtake Model' on the valve to pipe connection.



2. Add break discharge boundary conditions (BCs).

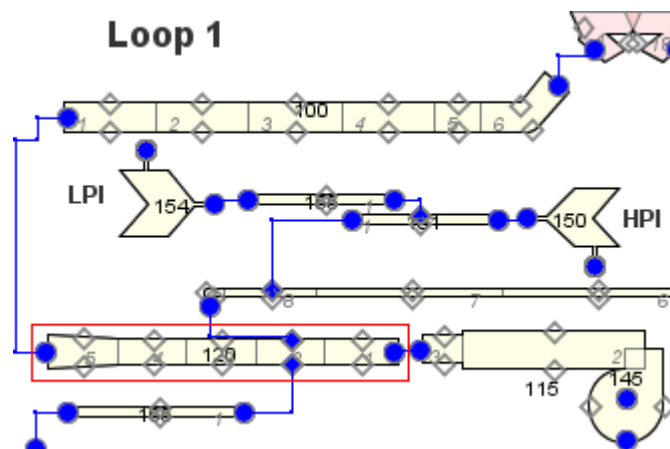
- Connect a BREAK  component to the valve outlet.
- Set the component number to 806.
- Set the BCs to 1.0E5 Pa (~atmospheric pressure) and 373 K (TSat).
- Set Temperature Table Option  [3] Set liquid and gas to Tsat.
- Set Initial Gas Volume Fraction  1.0.




Noncondensibles can significantly slow down a simulation. In some simulations backflow from the break will enter the system. Pure steam conditions can be set at the break to avoid backflow of noncondensibles.

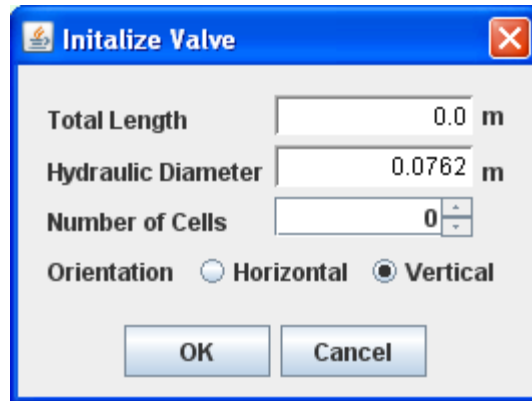
DETAILED STEPS

1. Add a 3 inch (0.0762 m) break to the bottom of Loop 1 Cold Leg.
 - a) If it is not currently visible, click on the **Hydro Comps** tab below the **View Window** to show the PWR model.
 - b) The cold leg break model will be connected to cold leg pipe 120. Find pipe 120 typing 'Ctrl-F' to bring up the 'Find Component' dialog. Click on the **Number** column header in the dialog to sort the list by component number and look for 120. Click on the 120 to select this component and click **Find**. Cold Leg pipe 120 should now be highlighted as shown below:



- c) Click on the VALVE component icon  from the **Toolbar** and then click below pipe 120 in the **View Window** to insert a valve.
 - d) An 'Initialize Valve' dialog should pop-up. Set the following values and click

OK. 'Number of Cell' and 'Total Length' of zero indicate that an SJC VALVE is added.



- e) In the **Properties Window** set **Component Name** , and hit enter.



Naming components is a good practice since it makes the components easier to identify in the TRACE input file, and in various SNAP dialogs as well.

- f) In the **Properties Window**, set **Component Number** .



A good component numbering scheme makes the model easier to interpret. In this case the loop one component numbers are in the 100-199 range, loop 2 components are in the range 200-299, and loop 3 in 300-399. Corresponding components use the same last two digits. For example, the pumps in the three loops are 115, 215, and 315. Although the break is located on Loop 1, it was elected to use a unique numbering scheme for the break components.

- g) In the **Properties Window** set the **Unknown** values for the valve as shown below:

Maximum Valve Rate	1.0E5 (1/s)	◀▶
Off Adjustment Rate	1.0E5 (1/s)	◀▶
Minimum Position	0.0 (-)	◀▶
Maximum Position	1.0 (-)	◀▶
Valve Flow Area	4.56E-3 (m ²)	◀▶
Valve Hydro Diameter	0.0762 (m)	◀▶
Initial Flow Area Fraction	0.0 (-)	◀▶
Valve Stem Position	0.0 (-)	◀▶





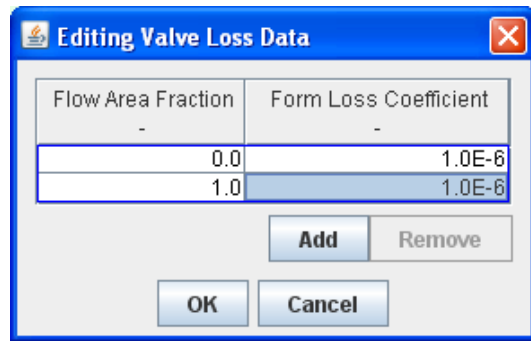
The **Maximum Valve Rate** needs to be large enough for the valve to be responsive. The valve behavior will be configured later. The **Minimum Position** and the **Maximum Position**, as well as the **Initial Flow Area Fraction** and **Valve Stem Position** values are all fractions (values between 0 and 1). The valve is closed at the start.

- h) In the **Properties Window**, set **Internal Loss Model** to **[1] Off**.



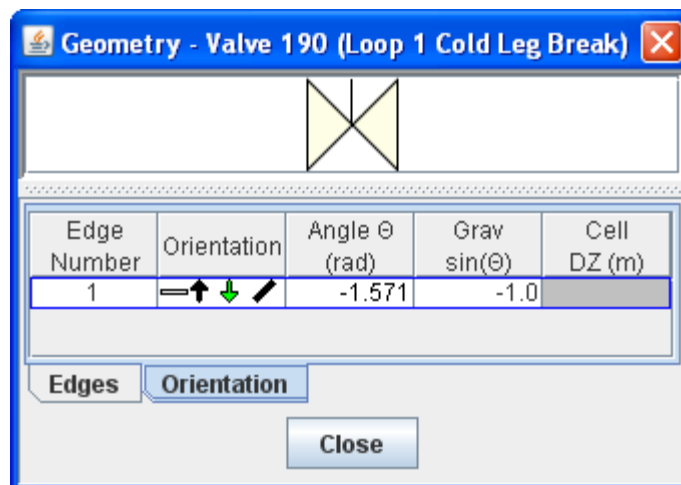
This is an easy step to forget, and one which can affect the break flow. TRACE VALVE components have a built-in loss factor. Since we are modeling a hole in the cold leg pipe and not a physical valve, this loss factor should be disabled. No SNAP or TRACE warning or error messages will indicate that this step has been neglected.

- i) In the **Properties Window**, expand the **Form Loss Table** **Undefined Values**  dialog by clicking on , and set the following values in the dialog:

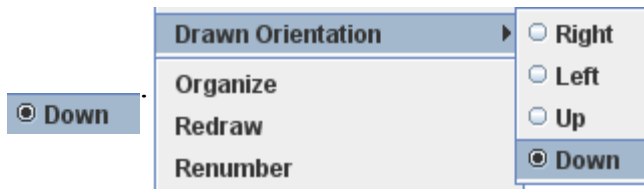


Losses of zero can be used. However, TRACE will print out warning messages. To avoid this a very small value was used for the loss coefficients.

- j) In the **Properties Window**, expand the **Component Geometry** Cells: 0 properties dialog by clicking on . Click on the **Orientation** tab on the dialog and set the orientation to down (Grav of -1) by double clicking the down arrow, then click on **Close**.

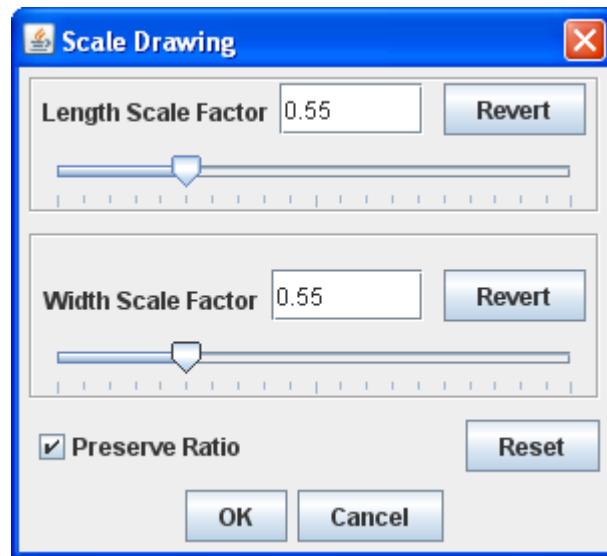


- k) Right click on the valve . From the pop-up menu, set **Drawn Orientation** to

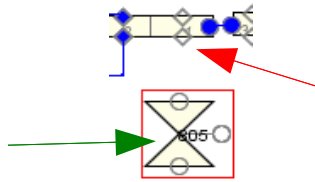


Setting the 'Drawn Orientation' is cosmetic. It doesn't actually modify the valve component orientation as in the step above. However keeping the visual orientation consistent with the component orientation makes the model easier to interpret.


- l) Right click on the valve component and select **Scale Drawing...**. On the 'Scale Drawing' dialog select ☒ **Preserve Ratio** and set the **Length Scale Factor** to about 0.55, and click **OK**. (Again this is cosmetic).

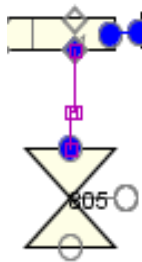



- m) Drag the valve such that it is located below PIPE 120 cell 1.



The previous image includes arrows which point to the connection points for the SJC VALVE (green arrow) and PIPE 120 cell 3 (red arrow) which will be used in the next step.

- n) Click on the 'connection tool' icon  from the **Toolbar**, then click on the connection point for the SJC VALVE (see green arrow above) followed by the connection point for PIPE 120 cell 1 (see red arrow above). A line should appear connecting the valve to the pipe.



- o) The **Properties Window** should show the properties for the valve to pipe connection. If it doesn't click on the 'selection tool' icon  in the **Toolbar** and select the line connecting the valve to the pipe. In the **Properties Window**, change **Angle:** (deg) to **Angle:** (deg) and hit enter.



Setting the angle is an easy step to forget. If **Check Model** is selected from the **Tools** menu (Step 4 in this tutorial), SNAP issues the following error to help you find the problem:

Hydraulic [478] from Valve 1 to Pipe 120 [1] Error
 Connection Angle has not been set.

p) Below **Angle:** (deg), set **Offtake Model:** .




The vertical orientation of a pipe break is since it affects whether steam or liquid goes through the break when there is stratified flow. The offtake model is a flag that indicates whether offtake orientation should be tracked. The position of the break depends on the orientation of the valve that models the break. If it is oriented down, the break is at the bottom of the pipe, and liquid is expelled before steam in stratified flow.



Turning on the offtake model is an easy step to forget and can affect the vessel depressurization timing since it influences when steam is able to reach the break. For a LOCA, a break that is oriented downward is typically more conservative. No TRACE or SNAP warning or errors messages will be shown indicating that you have not turned on the offtake model.

2. Add the break discharge boundary conditions (BCs).

- a) On the **Toolbar** click on the BREAK component icon , then click in the **View Window** below the break valve.
- b) In the **Properties Window**, set **Component Name** and

Component Number , and set the **Unknown** break properties as shown below:

Length	<input type="text" value="0.6985"/> (m)
Volume	<input type="text" value="1.0E5"/> (m ³)
Initial Gas Volume Fraction	<input type="text" value="1.0"/> (-)
Initial Mixture Temperature	<input type="text" value="380.0"/> (K)
Initial Pressure	<input type="text" value="1.0E5"/> (Pa)




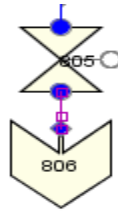
The length and volume values for the BREAK component are somewhat arbitrary, but the change in flow area between the VALVE and BREAK components affects choking. The BREAK component volume-to-length ratio (i.e. the BREAK component flow area) should be large to ensure that choking at the VALVE component is enabled appropriately. Since the frictional pressure drop calculation is based on the lengths of the upstream and downstream cells, the BREAK component length should be on the same order of magnitude as the length of the cell upstream of the choling location (which for this exercise is the cold leg cell).

c) Set Temperature Table Option **[3] Set liquid and gas to Tsat** .







As noted in the summary, the break conditions are set to saturated steam to avoid having noncondensibles enter the system due to backflow, since noncondensibles can significantly impact the simulation run-time.

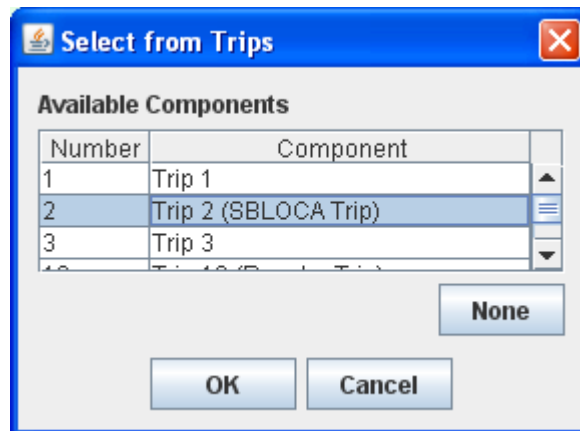
- d) Right click on the break component and set **Drawn Orientation** to **Down** .
- e) Right click on the break component and select **Scale Drawing...** . In the 'Scale Drawing' dialog enable ☒ **Preserve Ratio** and set **Length Scale Factor** .
- f) Drag break 806 below valve 805. Click on the connection tool  from the **Toolbar**, then click on the bottom connection point of VALVE 805 followed by the BREAK 806 connect point. A line should appear connecting the two components.





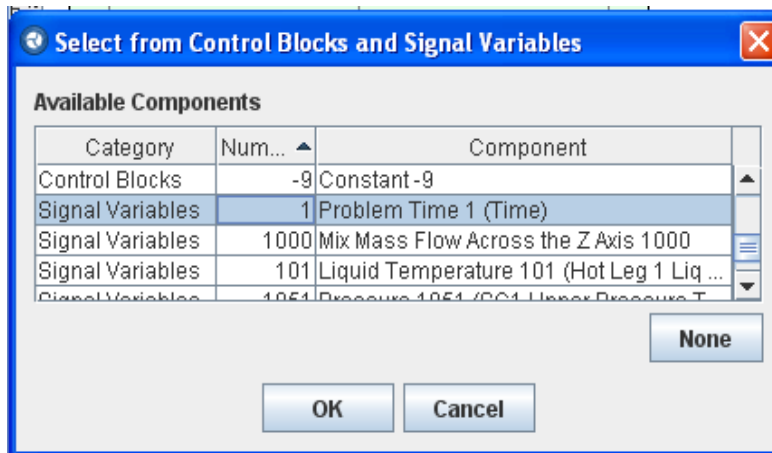
STEP 3. CONFIGURE BREAK VALVE TO OPEN ON SBLOCA TRIP

Configure valve 805 to open over $1/10^{\text{th}}$ of a second when trip 2 is activated.

1. Click on valve 805 (the cold leg break valve) and in the [Properties Window](#), click  on 'Valve Type' and select Valve Type [3] Flow Area Table After Trip .
2. Click  on Valve Trip <none> , select 'Trip 2 (SBLOCA Trip)', and click OK.



3. Click  on Valve Table Indep. Var. <none> , select Signal Variable 1 (Problem Time) as the table signal, and click OK.



Time is the signal that will be used as the valve control signal. **Signal categories** are organized alphabetically in this dialog, so scroll down to the 'General Signals' section. The relationship between this signal variable and the valve area fraction is set in the next step.

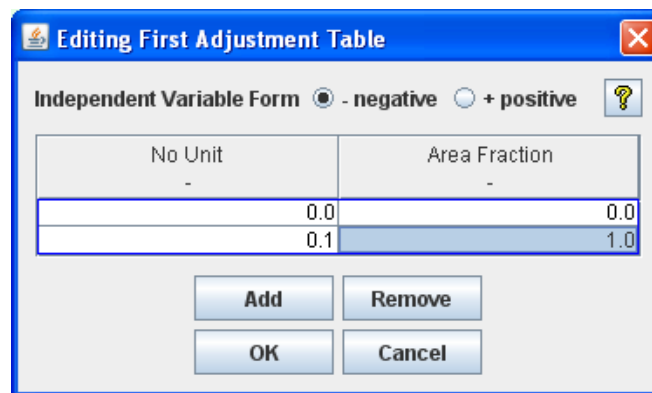
4. In the **Properties Window**, expand **First Adjustment Table** **Rows: 0** . Click



twice and set the values shown below. Above the table, set


Independent Variable Form ☒ - negative

and click




This 'Flow Area Table' option is actually a flow area fraction that ranges from 0 to 1. Make sure to use flow area fractions in this table rather than actual flow areas.



Independent Variable Form  - negative indicates that the table input is based on the change in the table signal from the time of the trip rather than the actual value of the table signal. For a time signal, this means that the time of the trip is the zero reference time for the table. This table causes the break to open over $1/10^{\text{th}}$ of a second after the SBLOCA trip is activated:

STEP 4. RUN SNAP 'MODEL CHECK' AND FIX ANY ERRORS

From the **Tools** menu select **Check Model**. This causes SNAP to review the model. An 'Error Report' dialog should pop-up that reports errors and warnings, and identifies the location of the problem. Errors are highlighted using the icon . If no errors are found



will be indicated in the top section of the dialog. Review and fix errors if any are found.

STEP 5. RUN A SHORT SBLOCA SIMULATION TO TEST THE BREAK

Step Summary:

If you have not finished adding the 3 inch break, close your model and open the SNAP file 'PWR4-BreakA.med'. This is a model that follows the detailed instructions to this point.

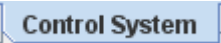
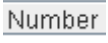


Run a short simulation (15 s) in order to see whether TRACE detects any errors and whether the break is behaving properly.

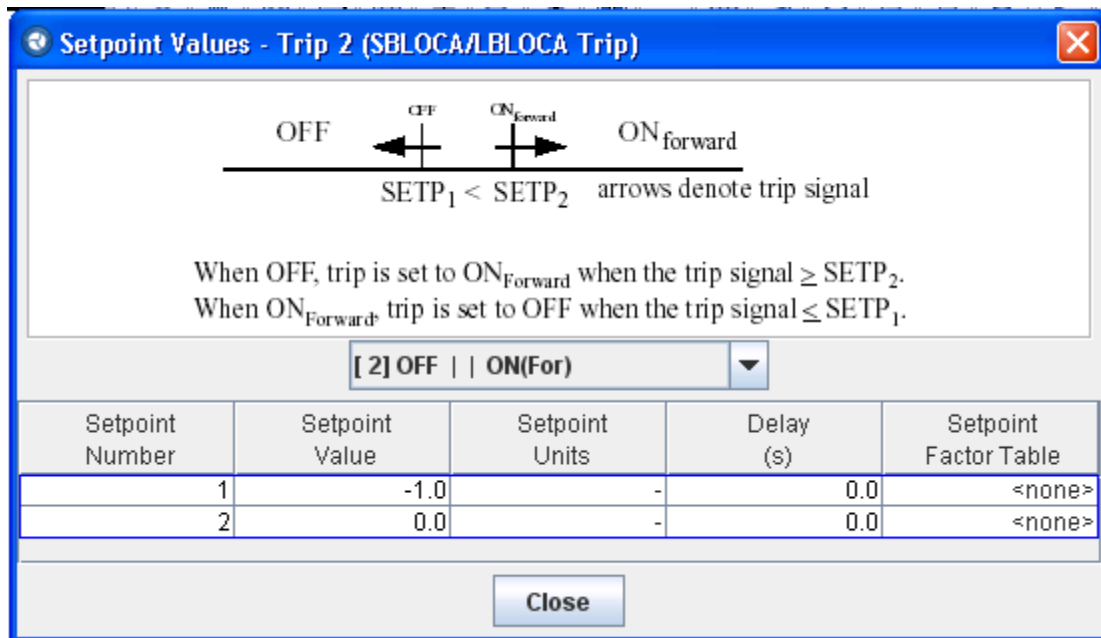


The model initial conditions should be relatively close to steady state conditions when running a quick test case. Otherwise errors may arise due to ill-conditioned initial conditions. Initial conditions can either be imported into the model after a steady state is run and the test run directly, or the simulation can be run as a restart from a current steady state simulation. For this model, the initial conditions have been imported, so the model can be run directly.

1. Configure the SBLOCA trip (Trip 2) to activate at time zero.
2. In model options, configure the model to run as a transient simulation.
3. In model options, adjust the timestep.
 - End Time: 15 (s)
 - Maximum Size: 0.1 (s)
 - Graphics Interval: 0.2 (s)
4. Start the simulation using the name 'BreakTest'.

Detailed Steps

1. Configure the SBLOCA trip (Trip 2) to activate at time zero.
 - a) Select the 'Control System' view by clicking on the  tab below the **View Window**.
 - b) Type 'Ctrl-F' to bring up the 'Find Component' dialog. Click on the  column to sort by component number and scroll to find Trip 2. Select this row and click . The properties for Trip 2 should now be visible in the **Properties Window**.
 - c) The signal for Trip 2 is time. The SBLOCA begins when time crosses setpoint 2. In the **Properties Window**, locate and expand  the “Setpoint Data” section. Set the “Setpoint Value” to 0.0 for “Setpoint Number” 2 as shown below then click on the close button.



2. In model options, configure the model to run as a transient simulation.

- a) In the **Navigator Window**, click on **Model Options**. In the **Properties Window**, set **Transient Calculation** to **[1] Transient**.



Trips are not evaluated in the Steady-State mode unless the trip number is preceded by the negative sign. Trip 2 is a positive trip. Therefore in order for the trip to function, the transient mode must be selected.

3. In model options, adjust the timestep.

- a) In the **Properties Window**, find **Timestep Data** [1] Timesteps and click on to expand the 'Edit Timestep Data' dialog. Set:


- End Time: 15 (s)
- Maximum Size: 0.1 (s)
- Graphics Interval: 0.2 (s)

Click .







A maximum timestep size of 0.1 s is a common maximum size to use for a LOCA transient. Sometimes the timestep size has to be reduced by one or two orders of magnitude in order for the model to avoid timestep failures.



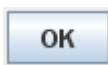
4. Start the simulation.



- a) Locate and click on the “Job Stream” tab at the bottom of the View Window. A job stream has already been set up to run this exercise.
- b) Lock the View Window by clicking on the padlock icon  located in the Toolbar on the left-hand side. When locked, the padlock is closed.
- c) Click on the “Execute” button in the View Window. This submits the job for execution.

STEP 6. CHECK RESULTS. MAKE CORRECTIONS IF NECESSARY.

1. Using 'Windows Explorer', go to the folder 'Day4/Morning/PWR4_Break_Model' and double click on the animation file 'PWR4-Anim.med' to open it in SNAP.
2. If the simulation was successful, connect the animation to the simulation:
 - a) In the **Navigator Window**, click the  on  **Data Sources [1]** to expand that data sources.
 - b) Select  **Master** under 'Data Sources', and in the **Properties Window**, expand

Source Run URL	calcserv://localhost/...	
----------------	--------------------------	---

 to bring up the 'Select Data Source' dialog.
 - c) Toggle the open key  next to 'Local' and then toggle the open key  next to 'Execute'. Locate and click on 'PWR4-Break'. On the right-hand side, click on 'PWR4-Break:trextv' and then click the  button at the bottom of the window.

- d) Click the connect icon  to connect to the PWR4-Break simulation, then click on the play  button.

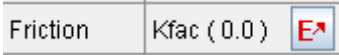
3. Check whether break flow is reasonable

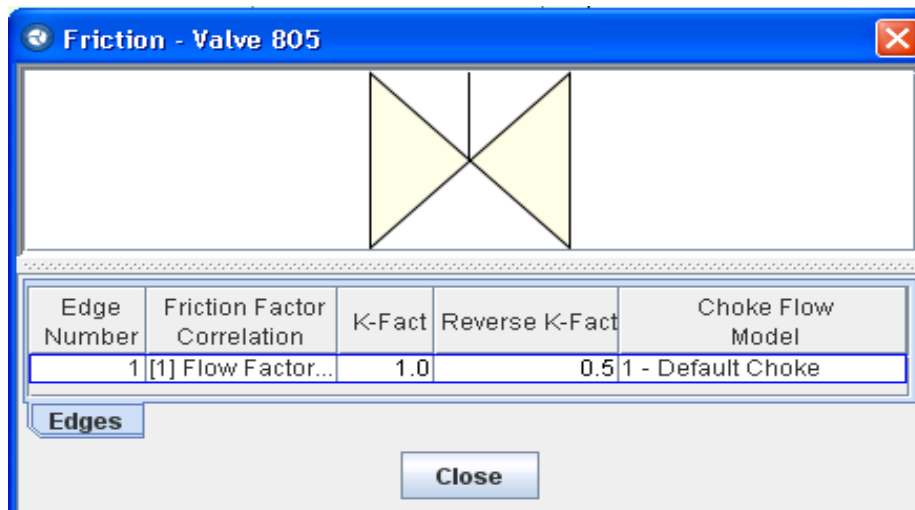
- a) Break flow is an important factor in LOCA events. To determine if it is reasonable, compare the TRACE calculated break flow velocity against the reference critical flow velocities given below.

PRESSURE	TEMPERATURE	CRITICAL LIQUID VELOCITY
15.5 MPa	559 K	155 m/s
10 MPa	551 K	107 m/s



Checking important model responses against reference values is a useful way to determine if there are modeling errors that need to be addressed. When possible, it is helpful to do a simple test that verifies that the system is behaving as expected.

- b) If you followed the detailed instructions above, the calculated break flow velocity should be significantly higher than the velocities shown in the table above. This difference is because setting the choked flow flag in the break VALVE component was intentionally omitted from the instructions to emphasize the importance of checking the model results to make sure that reasonable values are being obtained.
4. To set the choking flag, go to the 'PWR4-Break.med' model editor (or the PWR4-BreakB.med file in the Day4/Morning/PWR4_Break_Model folder) and click on the break valve (VALVE 805). (You will need to unlock the View Window)
5. In the **Properties Window**, locate and expand .
6. The geometry at the break can be considered an infinite expansion, for which the forward flow loss coefficient is about 1.0 and the reverse flow loss coefficient is about 0.5. In the 'Friction' dialog box, set the following values for the K-factors and choking model selection, then click on the Close button at the bottom of the window.



7. If you have not completed the steps to this point, close your model and open the SNAP file 'PWR4-Break-Final.med' located in the Day4/Morning/PWR4_Break_Model folder.

STEP 7. RERUN THE SHORT SBLOCA SIMULATION AND RECHECK THE RESULTS.

Rerun the simulation (Step 5, Item 4), then check the results again by replaying the simulation in the animation med file (Step 6, Item 2d). Verify that the break flow is reasonable (Step 6, Item 3).