MIT Pressurizer Insurge Test ST4 – Exercise Problem

Objectives

- Gain experience in building a complete TRACE model from start to finish.
- Learn how to add PIPE, FILL, and heat structure (HTSTR) components to a model.
- Become familiar with basic model options (simulation type, timestep control).
- Gain experience in adding control blocks to extract useful model information.
- Learn how to apply relevant boundary conditions.
- Assess whether important phenomena have been adequately modeled and make model adjustments as necessary.

Overview of Steps

1. Preliminary Setup (Start Model Editor)
2. Create MIT Pressurizer Experiment Model

   A) Create Pressurizer “PIPE” with Initial Conditions

   B) Create Pressurizer “FILL” to set Boundary Conditions

   C) Set Model Options
3. Add Control Systems to Calculate Values of Interest

   A) Pressurizer Liquid Level Signal

   B) Pressurizer Pressure Signal (in MPa)

   C) Experimental Pressure Signal (in MPa)
4. Open the Animation File
5. Run Simulation and Compare to MIT Experimental Results

6. Modify Model as Necessary and Repeat Step 5.

Step 1. Preliminary Setup (Start Model Editor)

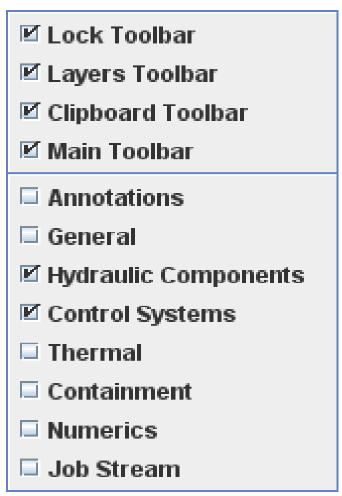
1. Open the SNAP Model Editor and (if questioned) answer “No” to “Open 1 model from the previous Model Editor Session?”

   a) Start the SNAP Model Editor.

   b) Under **Create a New Model** click Continue >> .

   c) Select TRACE model and click OK.

2. Right click on the right side portion of the Toolbar where there are no icons. This brings up the Toolbar menu. If the Hydraulic Components item doesn't have a check mark next to it, click on it to activate the 'Hydraulic Components' toolbar. Bring up the Toolbar menu again and activate the Control Systems toolbar if it is not active.



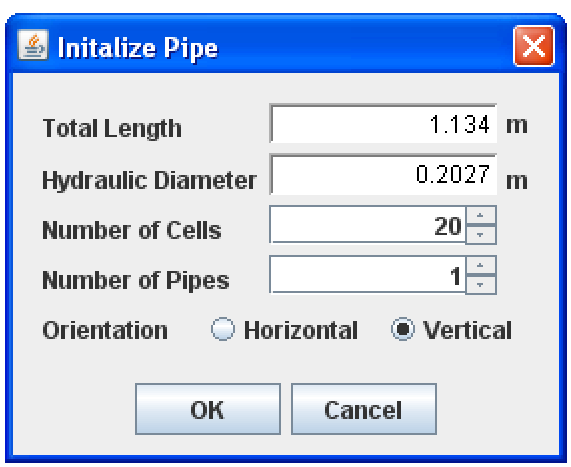
The Toolbar menus show icons for the various hydraulic components and control system components that are available in TRACE. If the model editor window is active, and if you hold the mouse pointer over a Toolbar icon, a label will appear that identifies what the icon represents. You can add a component by clicking on the icon, then clicking in the View Window. The Toolbar icons will be used in the instructions that follow.

Step 2. Create MIT Pressurizer Experiment 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.A) Create Pressurizer "PIPE" with Initial Conditions

1. Click on the PIPE component icon from the Toolbar and then click in the View Window. An "Initialize PIPE" dialog should pop-up.
2. Set the following properties in the pop-up window and click OK:



3. The pipe will be very small in the view. Click in the View Window away from the PIPE component so that the PIPE is not selected. This brings up the View Window properties. In the Properties Window set Pixels Per Meter 300.

> The 'Pixels per Meter' value defines the display size for components that are added to the view. By default the 'Pixels Per Meter' used in SNAP is 20, which is reasonable for medium-to- large TRACE facility models. For small TRACE facility models, the pixels per meter value needs to be adjusted in order for components to have a reasonable screen display size.

4. Click on the PIPE component, and in the Properties Window set Component Number 333.

> An animation file has been created to help visualize the MIT pressurizer calculation results. The animation file looks up plot variables based on component numbers defined in the model. It expects the pressurizer to be component number 333, and other component numbers to be consistent with the numbers defined in these instructions.

5. On Initial Conditions | Unknown: TL TV P | E expand the initial conditions dialog by clicking E . Set initial conditions that follow and click Close:

Property	Value	Cells
Pressure (Pa)	4.923E5	All
Gas Volume Fraction	0.0	1-7
	0.44	8
	1.0	9-20

If you select all the pressurizer cells with the mouse before you enter the pressure, the value will be added to all the cells at once. Alternatively, you can enter the value in one cell, then copy the value (select the cell and type Ctrl-c) followed by selecting all the other pressure cells and pasting the value (Ctrl-v). The same idea applies to setting 'Gas Volume Fraction' values, but only select the set of cells you want to update.

Setting the pressure automatically sets the liquid and vapor temperatures to saturation values. Gas volume fractions must be set independently.

Step 2.B) Create Pressurizer "FILL" to set Boundary Conditions

1. Click on the FILL component icon from the Toolbar and then click in the View Window below the pressurizer PIPE.
2. In the Properties Window set Component Number 330.
3. We want to specify the flow vs. time for the pressurizer FILL in order to match experimental data, so set Fill Type [5] Mass Flow Table.
4. Set the following properties for the FILL:

Length	0.1	(m)
Volume	4.0E-3	(m^3)
Initial Gas Volume Fraction	0.0	(-)
Initial Liquid Temperature	294.4	(K)
Initial Vapor Temperature	294.4	(K)
Initial Pressure	4.923E5	(Pa)
Initial Noncondensible PP	0.0	(Pa)
Initial Coolant Mass Flow	0.412	(kg/s)

5. Configure the FILL mass flow table by doing the following:
   a) In the folder 'Day2\Morning\MIT_Pressurizer\' double click on 'FlowTable.inp' to

open it in a text editor. 'FlowTable.inp' holds the pressurizer inlet flow from the MIT Pressurizer experiment. Press 'Ctrl-A' to select the contents of the file, and 'Ctrl-C' to copy the contents to the clipboard.

b) Expand [Fill Table | Rows: 0] by clicking [E], and in the pop-up dialog, right click in the space below the 'Mixture Massflow' table header. Select 'Paste' to paste the copied data into the table.

c) Click [S] on [Independent Variable | <none> | S] and select 'Problem Time 1' from the pop-up dialog to set time as the mass flow table signal. Select [OK] on the selection dialog, then [OK] on the Fill Table dialog.

6. Connect the PIPE inlet to the FILL by selecting the 'Connection Tool' from the Toolbar. Click on the circular connection point at the bottom of the PIPE, then click on the connection point at the top of the FILL. A line connecting the PIPE to the FILL should appear.

Step 2.C) Set Model Options

1. In the Navigator Window, select [Model Options], and in the Properties Window, expand [Timestep Data | [1] Timesteps | E] and set the following timestep data. Values to change are highlighted below in blue. Click [OK]:

End Time	Minimum Size	Maximum Size	Heat vs Fluid Size	Max Conv. Power Diff	Long Edit Interval	Graphics Interval	Restart Interval	Short Edit Interval
60.0	1.0E-6	0.1	10.0	1.0E20	100.0	0.1	100.0	1.0

2. Expand [Namelist Variables | Active Entries: 2 | E]. In the Namelist Variables dialog, set [USE_IAPWS_ST - Steam table flag | ☑ | (•) True ( ) False] (search the 'General' section upward from the bottom) and click [X] to close the dialog box.

Including the steam table flag namelist option causes TRACE to use steam tables from the IF97 standard. Otherwise TRACE uses legacy internal curve fits to water property data that are less accurate. Using the IF97 standard is recommended.

Step 3. Add Control Systems to Calculate Values of Interest

Step 3.A) Pressurizer Liquid Level Signal

We would like to be able to see the collapsed liquid level as the simulation runs. In order to track the collapsed liquid level, we need to add a 'Collapsed Liquid Level' control signal as outlined below:

1. On the Toolbar, select the Signal Variables icon and click in the View Window. In the pop-up 'Select Variable Type' dialog select Volume Signal | 20 | Collapsed Water Level and click OK.
2. In the Properties Window, set Signal-variable ID 200.
3. Expand Signal Hydraulic: none and in the 'Signal Variable' dialog, expand Hydraulic none. Select 'Pipe 333' from the list and click OK. In the 'Signal Variable' dialog under 'Second Location', set Cell 20 of 20 and click OK.

The 'Collapsed Water Level Signal' calculates the liquid level in a specified component and cell range as if all the liquid contained in those cells were collapsed to fill the range from bottom to top. The input manual indicates that all cells within the range must be vertical.

Step 3.B) Pressurizer Pressure Signal (in MPa)

For convenience, it is useful to view the pressurizer pressure in units of MPa. The following control blocks convert the pressure at the top of the pressurizer to MPa:

1. On the Toolbar, select the Signal Variables icon and click in the View Window. In the pop-up 'Select Variable Type' dialog select Volume Signal | 21 | Pressure and click OK.
2. In the Properties Window, set Signal-variable ID 100.
3. Expand Signal Hydraulic: none and in the 'Signal Variable' dialog, expand Hydraulic none. Select 'Pipe 333' from the list and click OK. In the 'Signal Variable' dialog under 'Location' set Cell 20 of 20 to track the pressure at the top of the pressurizer.

The signal variable gives pressure in units of Pascals. We would like to convert these to MPa for plotting. There is not a way to do this through the signal variable itself. However, a signal variable can be connected to a dummy control block that is used to scale the value. This is done below.

4. On the Toolbar, select the Control Blocks icon and click in the View Window to the right of Pressure control 100 (i.e. 'P 100'). In the pop-up 'Select Control Type' dialog select Manipulation | 1 | Absolute Value and click OK.
5. In the Properties Window set the control block (component number) to: Component Number -101 and Gain 1.0E-6.

6. On the Toolbar, select the 'Connection Tool' and connect from control block 'P 100' to the input diamond of control block '-101 Abs'.

> The gain on control block '-101 Abs' is used to convert from a 'P 100' input signal of Pa to an output of MPa.

Step 3.C) Experimental Pressure Signal (in MPa)

The experimental results are added to the model via a control system function block for convenience in comparing the experimental results with the simulation results. Experimental pressure data is given as a function of time, so first a time signal will be added followed by a function block that receives the time signal and outputs the experimental pressure value. To add the experimental data do the following:

1. On the Toolbar, select the Signal Variables icon and click in the View Window. In the pop-up dialog, select General Signal | 0 | Problem Time.
2. In the Properties Window set Signal-variable ID | 300.
3. On the Toolbar, select the Control Blocks icon and click in the View Window. In the pop-up dialog, select Arithmetic | 102 | Function and click OK.
4. In the Properties Window set Component Number | -301.
5. On the Toolbar, select the 'Connection Tool' and connect from the problem time block 'TIMEOF 300' to function block '-301 Function' (click on any of the input diamonds).
6. In the folder 'Day3\Morning\MIT_Pressurizer\' double click on the file 'PressureData.inp' to open it in a text editor. 'PressureData.inp' contains the

experimental pressure data from the MIT Pressurizer experiment, in a text editor (data is given in MPa). Press 'Ctrl-A' to select the contents of the file, and 'Ctrl-C' to copy the contents to the clipboard.

7. Click on the '-301 Function' block, and in the Properties Window, expand Function Table Rows: ... . Right click below the 'unknown' table values and Select 'Paste' to paste the copied data into the table. (If a remaining row still indicates 'unknown', then select that row and click Remove.) Then click OK.

8. Save the MIT Pressurizer model as 'MIT-V1'.

Step 4. Open the Animation File

A SNAP animation file is available that is used to examine the simulation results for the MIT Pressurizer model. To open the animation, navigate to the folder 'Day2\Morning\MIT_Pressurizer\' and double click on the file 'MIT-Animation.med'. Note there are two Model Editors now active, one for the TRACE model (MIT-V1) and one for the Animation (MIT-Animation).

Step 5. Run Simulation and Compare to MIT Experimental Results - Iteration 1

1. To run the MIT Pressurizer simulation, first set up a Job Steam Window in the TRACE Model Editor with the appropriate links to the code and the model.

2. In the TRACE Model Editor Navigator Window, locate 'Views' and right-click. In the popup window that appears select 'New'. In the Properties Window rename the new window to "Job Stream".

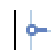
3. In the Navigator Window locate 'Job Streams [1]' and open it by clicking on the key to the left of the 'Job Streams' name. Left click on the 'Single_Step' and drag it out onto the Job Stream View Window. In the Properties Window, rename the

'Single_Step' button to 'Execute-MIT'.

4. Open the 'Execute-MIT' folder in the Navigator Window. Left click on the 'Stream Steps' and drag it onto the Job Stream View Window. Two boxes appear, one titled TRACE and the other titled AptPlot. Separate the boxes. Click on the TRACE box and in the Properties Window select 'V5840' for the application (this is the TRACE code executable).

5. Locate 'Model Nodes' box under Job Streams in the Navigator Window. Left click and drag it onto the Job Stream View Window. This is the TRACE model input file.

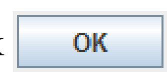
6. In the Main Tool Bar above the View Window is a red padlock icon. Click on it to "lock" the view (in the locked condition, the padlock is green). Re-save the MIT pressurizer TRACE model as 'MIT-V1', then in the View Window click on the Execute-MIT button to submit the simulation job. A Job Status Window will appear, indicating the progress of the calculation.

7. In the Animation Model Editor Navigator Window, click on  from 

to expand the data sources list.

8. Click on 

, and in the Properties Window expand 

. In the 'Select Data Source' dialog, expand the 'local' box, select 'Execute-MIT', and from the list pick 'Base_Job'. Click 

.

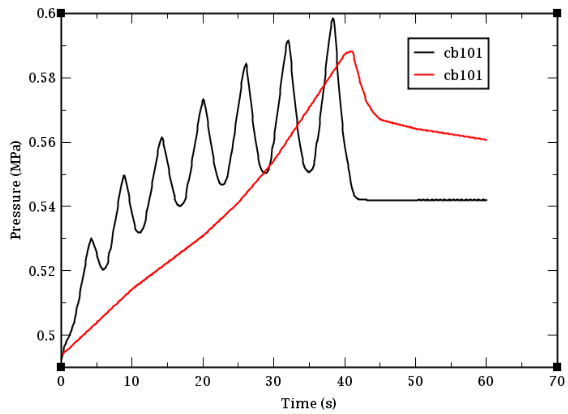
9. From the icons in the Main Tool Bar at the top of the screen select  to connect to the TRACE simulation chosen in the previous step to the animation screen. If the animation successfully connects, the icon should change to . Lock the animation file by clicking on the red padlock located in the top left corner of the View Window.

10. Select the play icon  to start the animation.

11. Examine the results. Does the simulation match the experimental data reasonably well?

STEP 6. MODIFY MODEL AS NECESSARY AND REPEAT STEP 5. - ITERATION 2

Your comparison should look similar to this:



A non-physical jagged pressure response is predicted with TRACE. This behavior appears to be caused by liquid crossing cell boundaries, resulting in a two-phase fluid state in the cell containing the liquid level as the pressurizer fills. Apparently the heat transfer between the fluid and liquid, as calculated by TRACE, varies significantly with the amount of water contained in the cell, causing condensation rates to vary significantly. This in turn causes the jagged pressure response.

A TRACE 'level tracking' option is available that: (1) permits tracking of the water level within a column of cells and (2) divides the cell containing the water level into two sub-volumes, one below the water line and one above the water line. Fluid properties, such as void fraction, are tracked independently in each sub-volume.

Activate 'level tracking' by doing the following, then perform Step 5 Iteration 2:

1. Select the Model Editor window that contains the TRACE MIT pressurizer model. In the Navigator Window, select Model Options .
2. In the Properties Window, expand the 'Namelist Variables' item. In the 'Namelist Variables' dialog box that appears, set parameter “NOLTD1” to “[-1] On In All 1D Cells”, then close the dialog box.
3. At the top of the Properties Window, change the name of the model to MIT-V2. Then on the Main Tool Bar, select “File”, “Save As”, and enter

MIT-V2.med.

Step 5. Run Simulation and Compare to MIT Experimental Results - Iteration 2

1. If it is still open, close the "SNAP Job Status 2.2.7" window from the previous calculation.
2. In the "Job Stream" tab View Window click the "Execute-MIT" button to submit the Iteration 2 simulation job. A Job Status Window will appear, indicating the progress of the calculation.
3. To view the animation results for Iteration 1, repeat the process described above for Step 5, Iteration 1, Items 7 through 11.

> If an animation is connected to a simulation, and a new simulation is submitted over the top of the previous one, the animation will automatically disconnect, so each time you submit a model you need to reconnect the animation.

4. Examine the Iteration 2 results. Does the simulation match the experimental data reasonably well? Are there important effects that were not included that could be added to the model?

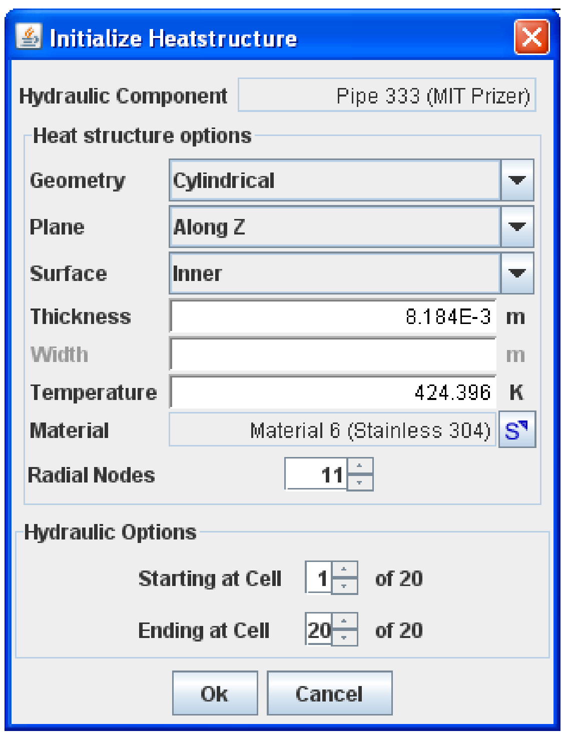
Step 6. Modify Model as Necessary and Repeat Step 5. - Iteration 3

In the Iteration 2 TRACE simulation, the pressure increases much more rapidly than in the experiment and the Iteration 1 TRACE simulation. One of the reasons for this difference is that the wall of the MIT Pressurizer were neglected in the TRACE model. The metal wall acts as a heat sink for the fluid inside the tank and can have a great impact on condensation processes. The compression of the steam in the tank causes the saturation temperature to rise, thereby increasing the heat transfer from the steam to the cooler tank wall. For Iteration 3, we will add heat structures to representing the pressurizer wall by doing the following:

1. Select the 'Model Editor' window that contains the MIT Pressurizer model. Click on the 'Default' view tab, then unlock the model by clicking on the green padlock at the top left corner of the View Window. Right click on Pressurizer Pipe 333, and from the pop-up menu select 'Edit Heat Structures→New'.

2. Set the heat structure properties for the entries shown below and click OK. Click Close on the heat structure properties view window that pops up.

3. In the Navigator Window, select Model Options and at the top of the Properties Window, change the name of the model to MIT-V3. Relock the model by clicking on the red padlock. Then on the Main Tool Bar, select “File”, “Save As”, and enter MIT-V3.med.



Step 5. Run Simulation and Compare to MIT Experimental Results - Iteration 3

1. If it is still open, close the “SNAP Job Status 2.2.7” window from the previous calculation.
2. In the “Job Stream” tab View Window click the “Execute-MIT” button to submit the Iteration 3 simulation job. A Job Status Window will appear, indicating the progress of the calculation.
3. To view the animation results for Iteration 3, repeat the process described above for Step 5, Iteration 1, Items 7 through 11.
4. Examine the Iteration 3 results. Does the simulation match the experimental data? Have any important effects been neglected?

Step 6. Modify Model as Necessary and Repeat Step 5. - Iteration 4

The calculated pressure is somewhat higher than the measured pressure. Heat loss to the environment is neglected in the model. For the next iteration, add a constant heat flux of 2,000 W/m^2 from the outside of the pressurizer wall to the ambient environment simulating heat loss by doing the following:

1. Select the 'Model Editor' window that contains the MIT Pressurizer model. Click on the 'Default' view tab, then unlock the model by clicking on the green padlock at the top left corner of the View Window. Right click on Pressurizer Pipe 333, and from the pop-up menu select 'Edit Heat Structures→Heat Structure 10'.
2. In the pop-up properties dialog, expand Axial Nodes / Surface BCs | 20 Axial Cells. Left click on the top 'Outer Surface Boundary Condition', and while holding the mouse button down, drag down through all 20 boundary condition cells to select them. At the bottom of the dialog box, set Heat Flux | 2000.0 (W/m^2) and click “OK”. All the outer surface boundary conditions are now a heat flux of 2000 W/m^2. Close the Heat Structure 10 Property View Window.

3. In the Navigator Window, select Model Options and at the top of the Properties Window, change the name of the model to MIT-V4. Relock the model by clicking on the red padlock. Then on the Main Tool Bar, select "File", "Save As", and enter MIT-V4.med.

Step 5. Run Simulation and Compare to MIT Experimental Results - Iteration 4

1. If it is still open, close the "SNAP Job Status 2.2.7" window from the previous calculation.
2. In the "Job Stream" tab View Window click the "Execute-MIT" button to submit the Iteration 4 simulation job. A Job Status Window will appear, indicating the progress of the calculation.
3. To view the animation results for Iteration 4, repeat the process described above for Step 5, iteration 1, Items 7 through 11.
4. Examine the Iteration 4 results. Does the TRACE simulation now match the experimental data well?

For this experiment, the simulation of heat transfer from the fluid to the walls of the pressurizer is shown to be fundamental for accurately predicting the system pressure response. This heat transfer results primarily from the high compression of the fluid during the liquid insurge. The resulting increase in saturation temperature, causes increased wall condensation. It is noted that heat structures are not typically included on the pressurizers of plant models used for LOCA simulations because pressurizer insurge is not experienced. However, the behavior observed in the MIT pressurizer insurge experiment is relevant for non-LOCA plant models used for simulating pressurization events,

Step 6. Modify Model as Necessary and Repeat Step 5. - Iteration 5

A reasonable match of the data was observed in Iteration 4. This iteration will examine the calculated response based on time step size.

1. Select the 'Model Editor' window that contains the MIT Pressurizer model. Click on the 'Default' view tab, then unlock the model by clicking on the green padlock at the top left corner of the View Window.
2. Using Item 1 of Step 2.C, modify the maximum time step size (Maximum Size) from 0.1 to 0.2.
3. In the Navigator Window, select **Model Options** and at the top of the Properties Window, change the name of the model to MIT-V5. Relock the model by clicking on the red padlock. Then on the Main Tool Bar, select "File", "Save As", and enter MIT-V5.med.
4.

Step 5. Run Simulation and Compare to MIT Experimental Results - Iteration 5

1. If it is still open, close the "SNAP Job Status 2.2.7" window from the previous calculation.
2. In the "Job Stream" tab View Window click the "Execute-MIT" button to submit the Iteration 5 simulation job. A Job Status Window will appear, indicating the progress of the calculation.
3. To view the animation results for Iteration 5, repeat the process described above for Step 5, iteration 1, Items 7 through 11.
4. Examine the Iteration 5 results. How does increasing the time step size affect the calculated results. Does the TRACE simulation match the experimental data better?
5. Repeat Step 6 and 5, Iteration 5, for time step sizes of 0.05 and 0.025 seconds.

6. What does making the time step smaller do for the calculated results.

Points to Consider

- This example exercise shows that simulation results can deviate significantly from physical results if the relevant phenomena have not been modeled. Therefore, it is important to carefully consider and be aware of the relevant physical phenomena and include models for their simulation.

> For many accident types, a PIRT (Phenomenon Identification Ranking Table) has been developed by field experts, that helps identify phenomena that are expected to be important for an accident type.

- If boundary conditions are known, or can be derived from a larger model, simple separate effects models (such as used in this exercise) can be very helpful for identifying the phenomena important for plant safety analysis.
- The time step size determines how well the solution converges on the desired parameters of interest. Sensitivity studies on time step size should be made to determine a time step size that gives a reasonable response compared to the measured data.