

TRACE Workshop
Potomac, Maryland
March 2005

Symbolic Nuclear Analysis Package (SNAP) Tutorial

Prepared by:

Ken Jones
John Rothe
William Dunsford

Applied Programming Technology, Inc.
Bloomsburg, PA 17815

and

Chester Gingrich

USNRC
Office of Nuclear Regulatory Research
U.S. Nuclear Regulatory Commission

Table of Contents

Introduction.....	1
Exercise 1. Creating a Simple TRACE Model.....	8
Exercise 2. Submit the Model.....	19
Exercise 3. Creating an Animation Display.....	23
Exercise 4. Component Renodalization.....	27
Exercise 5. Working with View Templates.....	30
Exercise 6. Restarting a TRACE Calculation.....	32
Exercise 7. Using the jEdit Text Editor.....	34

Introduction

The Symbolic Nuclear Analysis Package (SNAP) consists of a suite of integrated applications designed to simplify the process of performing thermal-hydraulic analysis. SNAP provides a highly flexible framework for creating and editing input for engineering analysis codes as well as extensive functionality for submitting, monitoring, and interacting with the analysis codes. The modular plug-in design of the software allows functionality to be tailored to the specific requirements of each analysis code. SNAP currently supports the RELAP5, TRACE, CONTAIN and FRAPCON-3 analysis codes. Support for the MELCOR and FRAPTRAN codes is currently under development.

This tutorial is designed to provide an introduction to SNAP's basic functionality through a series of exercises using the TRACE thermal-hydraulic analysis code.

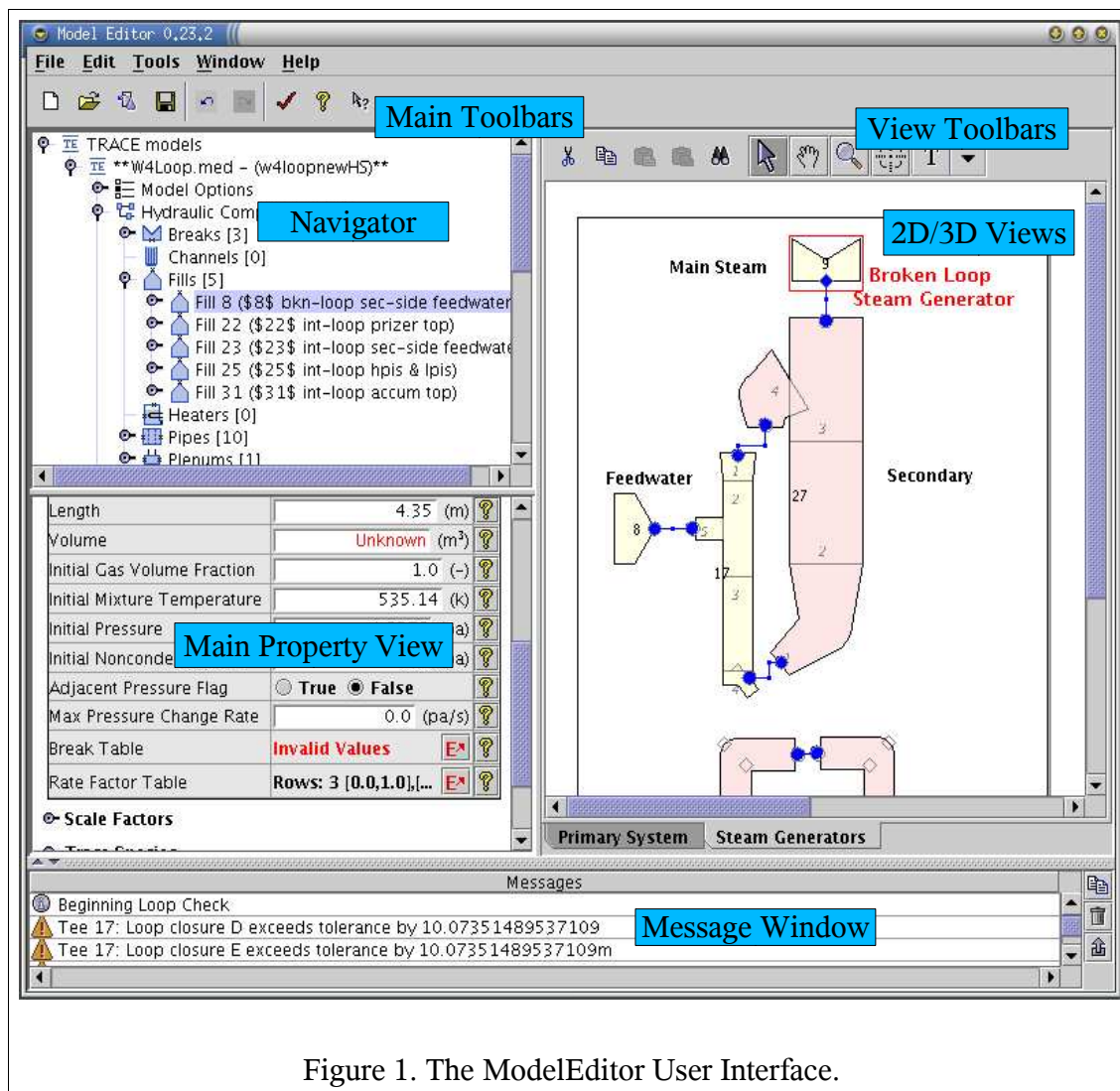
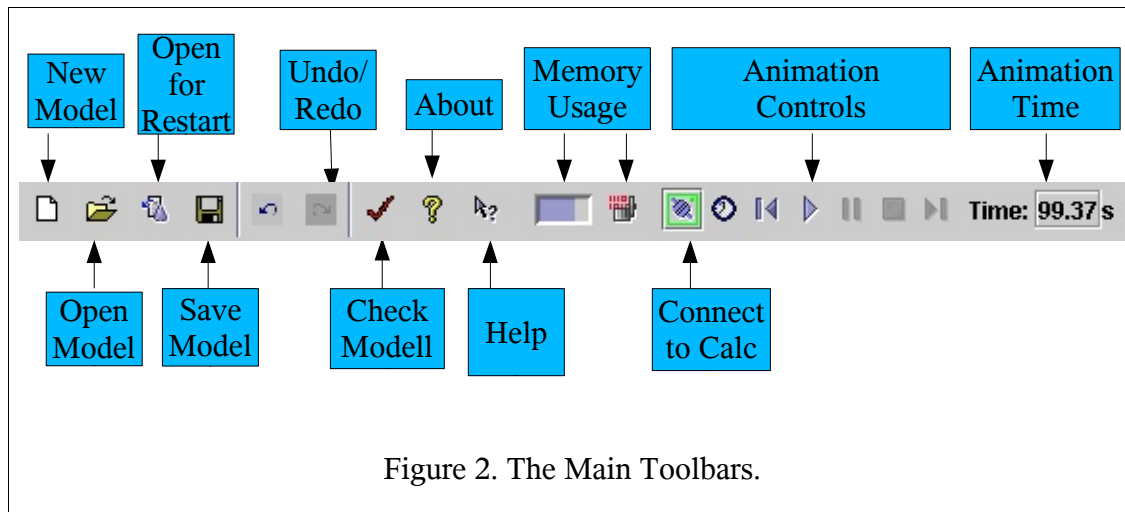


Figure 1. The ModelEditor User Interface.

SNAP's ModelEditor user interface is illustrated in Figure 1 with labels placed on it's primary components. A brief description of each of these components is provided below.

The Main Toolbars

The main toolbars include basic file and model operations, memory usage, animation controls and animation time toolbars. These toolbars can be individually enabled or disabled using a right-click popup menu located on the main toolbar panel.



Buttons for the basic file and model operations include New, Open and Save as well as shortcuts for Undo/Redo, Check Model, the About box, and context sensitive help. The memory toolbar displays the current memory usage. The animation toolbars provide feedback and control when working with animation models.

The Navigator

The navigator provides a logical hierarchical representation of the model's components and views as illustrated in Figure 3. The root of this tree is the Plug-in node. Any model that is currently open will appear underneath the appropriate plug in node. In Figure 3, note that the node labeled W4Loop.med – (w4loopnewHS) appears directly under the node labeled TRACE models. This indicates that only one TRACE model is open, and that model is named w4loopnewHS, and was opened from the MED file W4Loop.med.

Each model is broken down into categories of components. The majority of these categories are Plug-in specific, however some are shared between all plug-ins. The navigator provides access all of the model's components including non-visual elements such as global model options and CCFL models.

Pop-up menus located on the individual nodes of the Navigator can be used to add, delete or edit components as well as perform operations on the model.

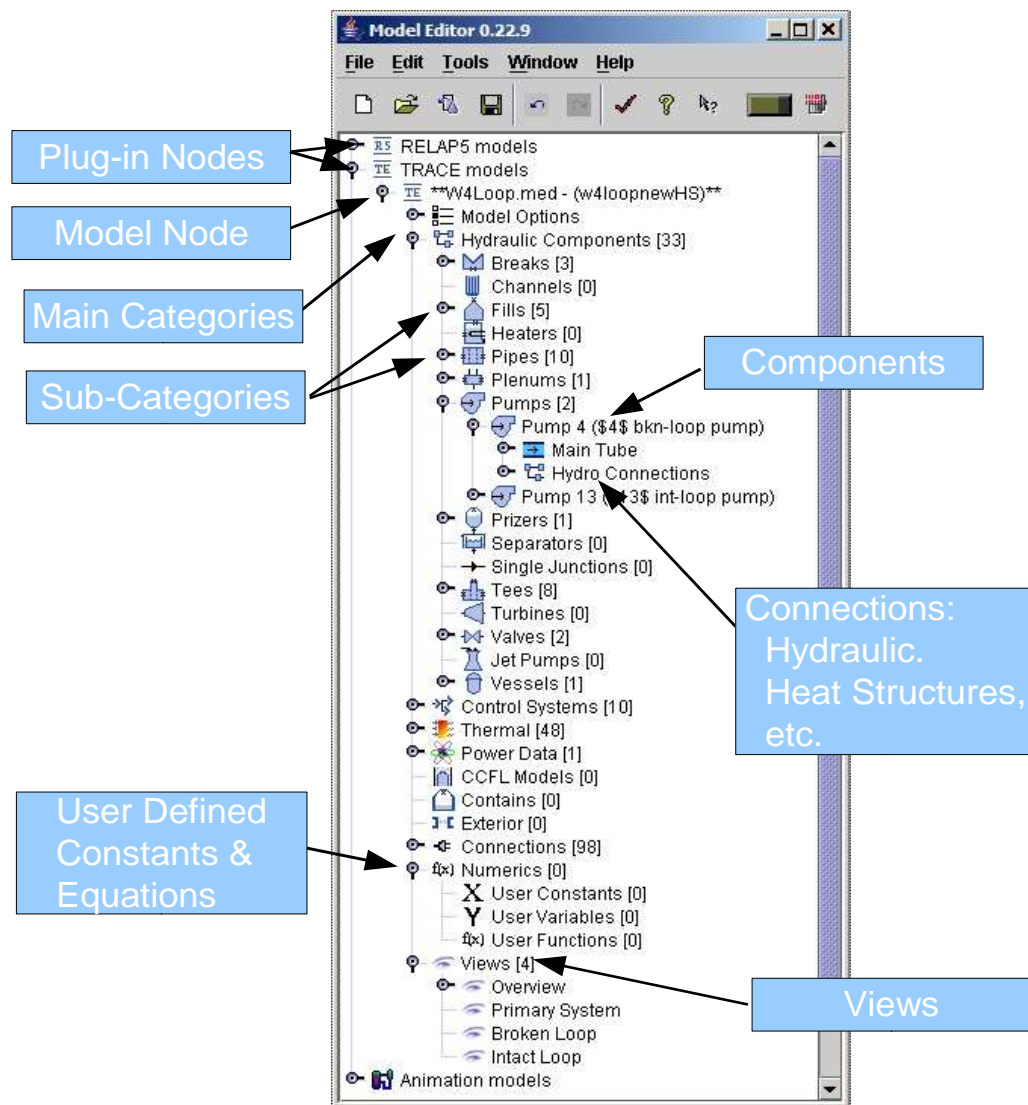


Figure 3. Component Navigator

NOTES:

- The number of components of that contained within a category or sub-category is displayed in brackets following the node name. Notice in Figure 3 that there are 33 hydraulic components, and two of those components are pumps.
- A model's name will appear surrounded by a pair of stars if that model has changes that need to be saved.

The Main Property View

The Main Property View provides the central point for viewing and editing properties in the ModelEditor. It displays the properties of the current selection in either the Navigator or a View. Changes to these properties will immediately reflected in all other open views (2D, ASCII, Property, etc.).

Property Views are made up of a set of Attribute Groups and a turn-key button to show and hide each. Beside each button is the name of the group.

Attribute Groups are used to organize the properties of a component. Each attribute group has a 3 column table of the properties included in the group (name, value and description). Some objects have only the *General* group whereas others have many different groups.

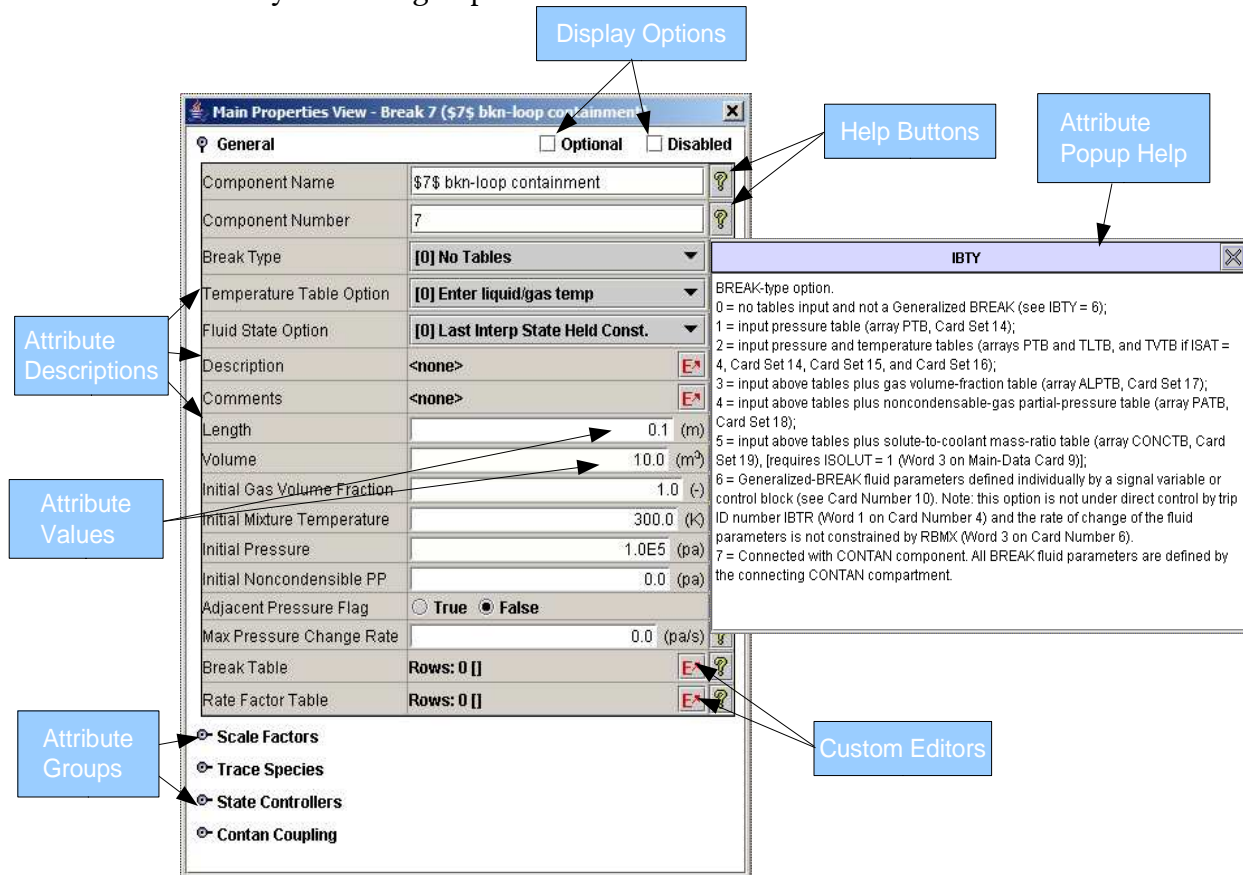


Figure 4. JavaBean Plug-ins Properties Views

Description Button - A description button appears next to each property. Pressing this button will show a more detailed description of the property being edited.

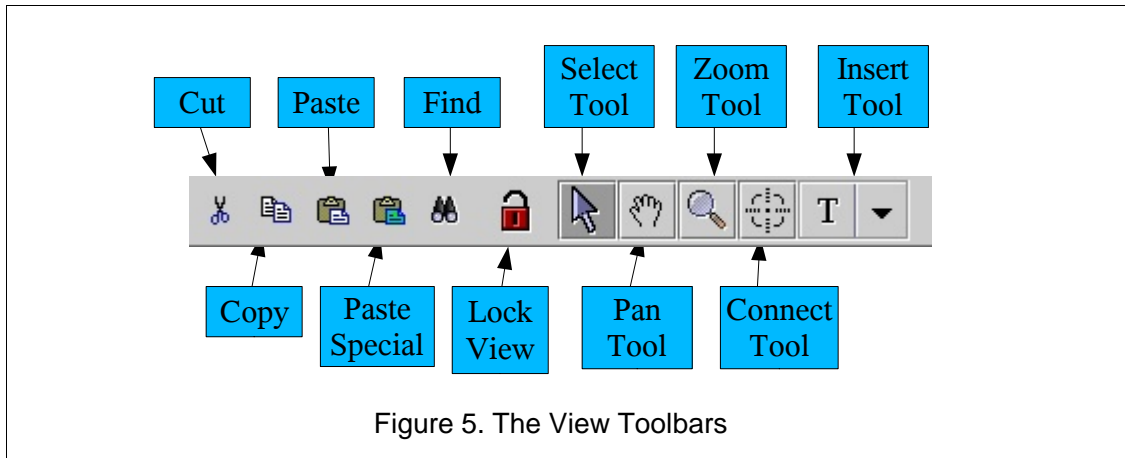
Optional/Disabled Checkboxes - These two checkboxes appear beside only the *General* Attribute Group button and are used to activate the display of optional

and disabled properties.

Custom Editor and Component Selection buttons are located adjacent to some attribute values. These buttons open detailed custom dialogs for editing components and for selecting other model components where appropriate. Buttons that contain a red **E** open editing dialogs while those containing a blue **S** open component selection dialogs.

View Toolbars

The view toolbars located above the 2D View include the main view toolbar, a tools toolbar, and optional plug-in specific toolbars used to select components to be inserted into the view. The main view toolbar contains buttons for cut, copy, paste, paste special and find operations. Paste special is used to paste multiple copies of a copied set or to change the component numbers of included components.



The tools toolbar contains buttons used for manipulating the View in various ways. These include:

- **Select Tool** - The select tool is used to select, move and manipulate elements of a View. Elements can be selected and dragged to new locations. Segments and points of a Line Annotation can be dragged to new locations.
- **Pan Tool** - The pan tool can be used to change the visible portion of a zoomed-in View.
- **Zoom Tool** - The zoom tool changes the zoom position of the View. Clicking in the View will zoom in a set amount. Holding the shift-key and clicking will zoom out the same fixed amount. Clicking and dragging to select a region (drawing a box) will zoom in to the selected region. Right-clicking in the View with the zoom tool will show the zoom menu for selecting a specific zoom position or fitting the entire View to the window.

- **Connect Tool** - This tool is used to create connections between components. For example: Connecting the outlet of a pipe to the inlet of another pipe would use this tool.
- **Insert Tool** - The insert tool includes a button to activate the tool and a drop down menu used to select the type of element to insert.

The Message Window

The Message Window displays a running list of error, warning, alert and notice messages. Processes such as saving a file or checking a model will produce messages in this window. Along the right side of this window are buttons to clear the window, export selected messages to a file or copy selected messages to the clipboard.

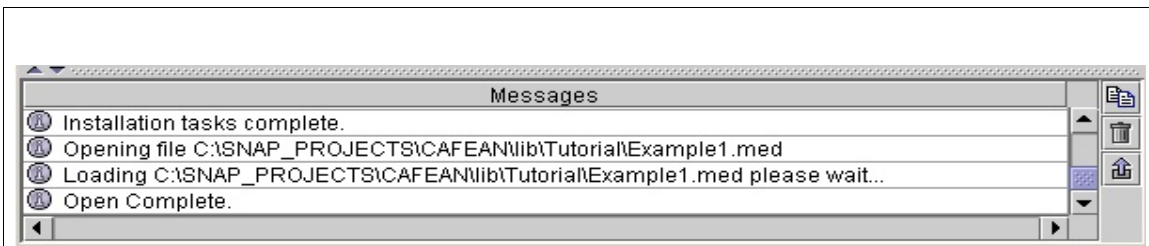
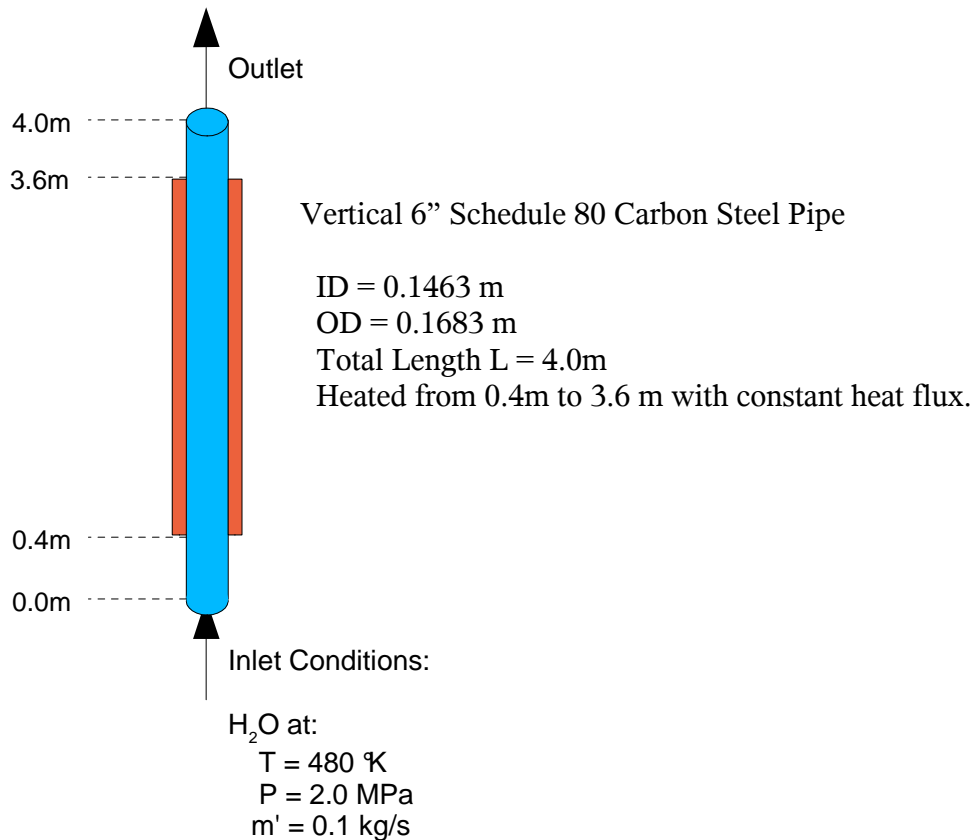


Figure 6. The Message Window

Exercise 1. Creating a Simple TRACE Model

This exercise is designed to familiarize the analyst with the basic functionality of SNAP's ModelEditor component. A TRACE model will be constructed and used to analyze the following system:



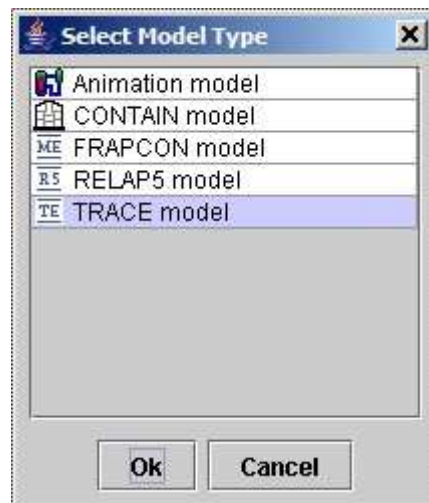
NOTE: This tutorial assumes you have properly installed and configured the SNAP software. Please refer to the installation instructions located on the SNAP website (<http://www.nrcsnap.com/snap/docs/quickstart.jsp>) if you have any questions concerning installation.

The following steps will guide you through creating a TRACE model for this system in SNAP.

1. Open the ModelEditor. Under Windows, select the “All Programs ► SNAP ► Model Editor” option from the Start menu. This will open the ModelEditor application and display initial popup dialog as shown below:

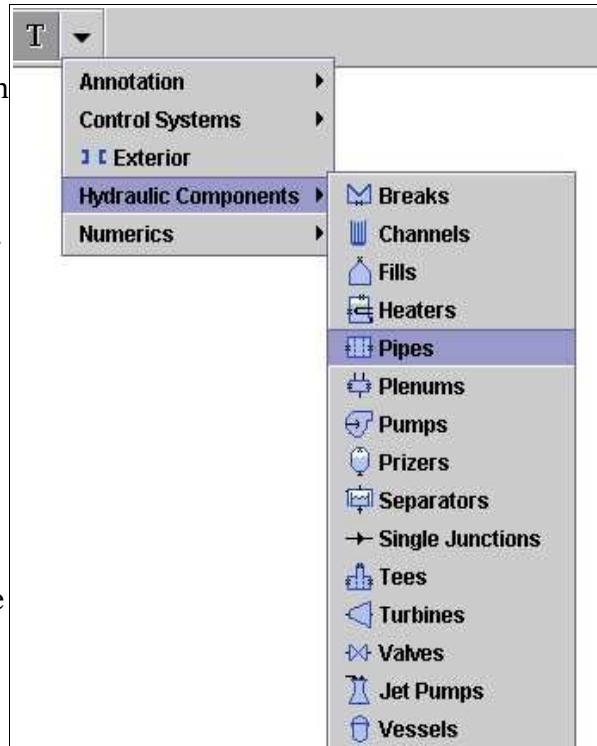
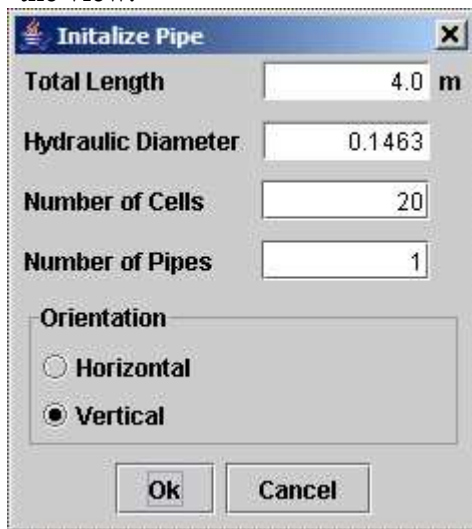


2. Select the “Create a New Model” option and the click the OK button. You will next be prompted for the type of model to be created:



Select TRACE from the list and press the OK button. This will open the ModelEditor with an empty TRACE model.

3. SNAP provides several methods of creating and editing components. We will use the component insertion button located above the 2D view in this example. Select the pipe component from the “Hydraulic Components ► Pipes” dropdown menu. The cursor will change to a crosshair when a component type is selected and the component's icon will be displayed in the component insertion button. Move the cursor to the 2D View and press the left mouse button. This will open a completion dialog for the pipe component. Enter the total length of the pipe (4.0 m), the diameter (0.1463 m), the number of cells (20) and set the orientation to Vertical. Press the OK button, and the pipe should appear in the view.



NOTES:

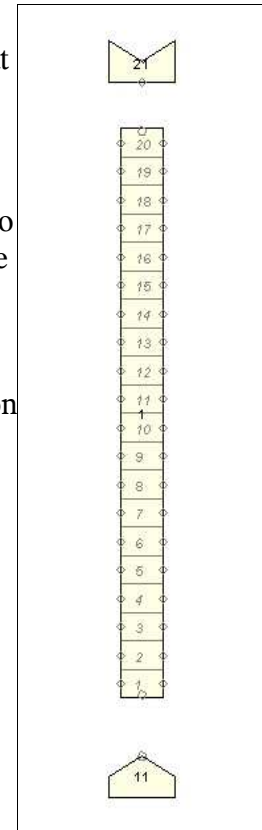
- SNAP will remember data values entered into the completion dialogs. These values will serve as the defaults the next time a component is created.
- To place multiple components on the canvas, hold the control key down while clicking on the canvas.

4. The 4.0m pipe created in the last step may appear very small on the View. Hydraulic components are drawn to scale based on their physical dimensions. The scaling of hydraulic components placed on the View is controlled by the “Pixels Per Meter” property of the view. The initial default value of 20 pixels/m is designed to display complete hydraulic nodalization diagrams for large full plant models. In our example problem, we want to increase the relative size of the hydraulic components within the view. A value of 100 pixels/m should work well for our test case. If the Views properties are not already displayed in the Main Property View, click anywhere on the View's background. Locate the “Pixels Per Meter” property in the Main Property

View and select it with the mouse. Change its value to 100 and press the enter key. Note that the pipe now appears larger in the view.

NOTES:

- Boundary Condition components such as Breaks and Fills, and Control System elements are not effected by the scaling factor.
 - Drawn components may be scaled individually using the “Scale Drawing” menu item available on the components right-click pop-up menu.
5. Although the pipe now appears larger it may still be difficult to distinguish individual cells and their connection points. The relative width to height scaling of hydraulic components placed on the View is controlled by the “Width Scale Factor” property of the view. Locate the Views “Width Scale Factor” property in the Main Property View and select it with the mouse. Change its value to 2.0 and press the enter key. Note that the width of the drawn pipe increases making it easier to distinguish individual cells and their connection points.
 6. Now add a fill and a break component using the same process that was used to create the pipe. The model should now appear as shown to the right:
 7. The next step is to connect the outlet of the fill to the inlet of the pipe. Select the connection tool button. The cursor will change to a target. Hover the cursor over the connection point located at the outlet of the fill. Notice that a set of crosshairs will appear in the target indicating that a connection can be made from this point. Click the left mouse button while the cursor is on the connection point. A rubber band line will then be created from the connection point to the cursor. Notice that a filled blue circle will appear when the cursor hovers over a valid connection point. Move the cursor to the the inlet connection point of the pipe and click the left mouse button to create the connection. A completion dialog will be displayed which prompts for the connection type. Select “Hydro Connection (JUN1)” and press the OK button. The connection should now be drawn between the two connection points.



NOTES:

- While a connection is being made, left clicking anywhere other than a valid connection point will create intermediate line segments for the connection.
- Right clicking while creating a connection will remove the last intermediate line segment or will cancel the connection if there are no intermediate segments. The escape key can also be used to cancel a connection.
- Connections have a right-click pop-up menu that can be used to add or remove

points, to remove the connection from the view (cut) or to delete the connection from the model.

8. Create a connection from the outlet of the pipe to the break component using the same process described above.
9. Switch the the “Selection Tool” and select the fill component. Notice that it's properties will be displayed in the main property view window. Enter the following values for the fill component:

Fill Type: [2]	Constant Mass Flow
Length:	0.2 m
Volume:	3.362084E-3 m ³
Liquid Temp:	480.0K
Vapor Temp:	480.0K
Pressure:	2.0E6 Pa
Initial Coolant Mass Flow:	0.1 kg/s

10. Select the break component and enter the following values for its' properties:

Break Type:	[0] No Tables
Length:	0.2 m
Volume:	3.362084E-3 m ³
Mixture Temp:	480.0K
Pressure:	2.0E6 Pa

The fill and break properties should now appear as shown below:

General ☐ Optional ☐ Disabled

Component Name	unnamed	?
Component Number	11	?
Description	<none>	E?
Comments	<none>	E?
Fill Type	[2] Constant Mass Flow	?
Length	0.2 (m)	?
Volume	4.449263E-3 (m³)	?
Initial Gas Volume Fraction	0.0 (-)	?
Initial Liquid Temperature	480.0 (K)	?
Initial Vapor Temperature	480.0 (K)	?
Initial Pressure	2.0E6 (Pa)	?
Initial Noncondensable PP	0.0 (Pa)	?
Initial Coolant Mass Flow	0.1 (kg/s)	?
Max Flowrate Change	1.0E20 (kg/s²)	?
Rate Factor Table	Rows: 0 []	E?

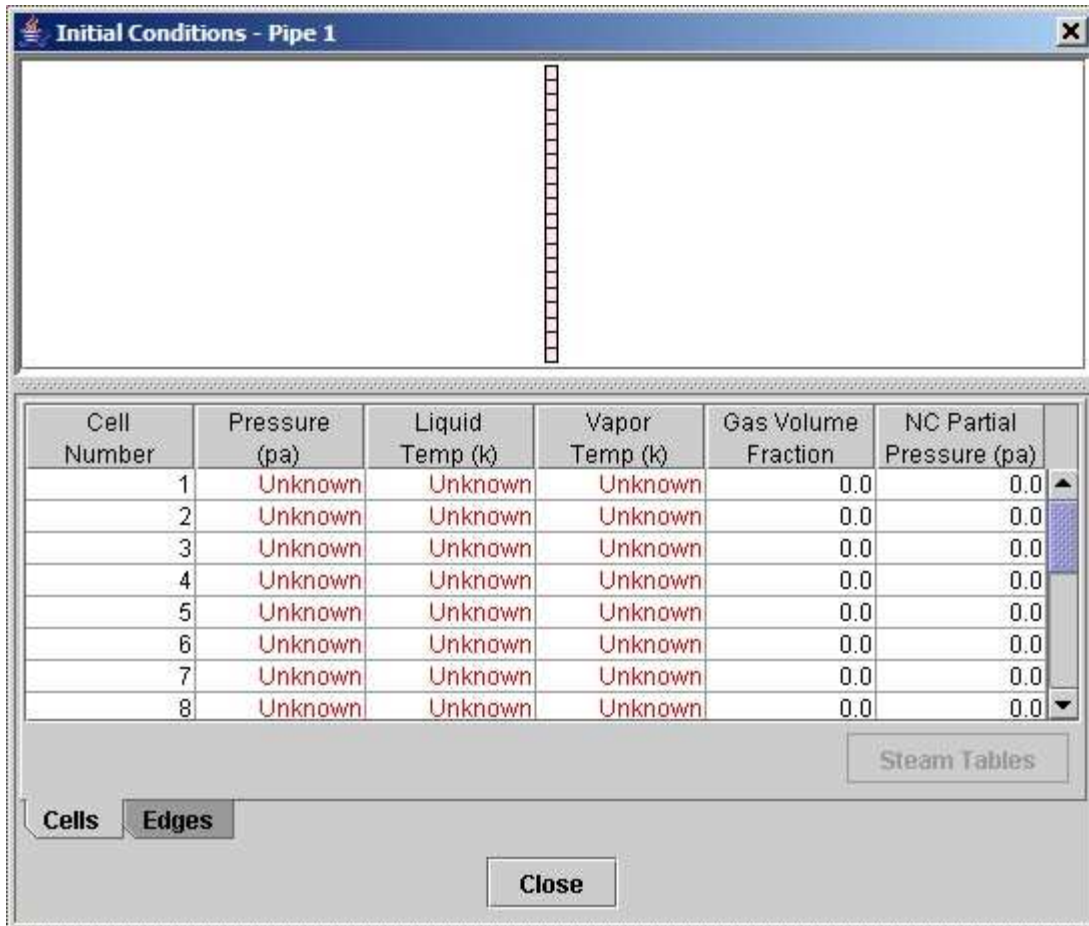
General ☐ Optional ☐ Disabled

Component Name	unnamed	?
Component Number	21	?
Break Type	[0] No Tables	?
Temperature Table Option	[0] Enter liquid/gas temp	?
Fluid State Option	[0] Last Interp State Hel...	?
Description	<none>	E?
Comments	<none>	E?
Length	0.2 (m)	?
Volume	4.449263E-3 (m³)	?
Initial Gas Volume Fraction	0.0 (-)	?
Initial Mixture Temperature	480.0 (K)	?
Initial Pressure	2.0E6 (Pa)	?
Initial Noncondensable PP	0.0 (Pa)	?
Adjacent Pressure Flag	<input type="radio"/> True <input checked="" type="radio"/> False	?
Max Pressure Change Rate	1.0E20 (Pa/s)	?
Break Table	Rows: 0 []	E?
Rate Factor Table	Rows: 0 []	E?

11. Now select the pipe component. Notice that the initial condition property is shown in red. This indicates that there is invalid or missing data for this property. Right click on the pipe to open its' popup menu and select "Show ASCII." This will open an ASCII view of the component's input. Note that the initial conditions (TL, TV, P) are all output as Unknown.

Open the Initial Conditions Editor for the pipe. This will display the dialog shown below. Select the pressure entry for the first cell and enter 2.0E6 Pa. Note that the liquid and vapor temperatures which were Unknown have been set the the saturation conditions.

Most tables in SNAP will allow you to edit multiple rows of a column in one step. To accomplish this, click and hold the left mouse button down on the first cell of the column to be edited and then drag the mouse down to the last cell to be edited and release it. The cells to be edited should all be highlighted. Type in the value and press the enter key to update all of the cells. Note: The first cell must not be selected prior to starting the multi-row selection process. Enter 2.0E6 Pa for the pressure in the remaining cells.



Note that the ASCII view will automatically update to reflect these changes.

12. The boundary condition for the outer surface of the heat structure will be controlled by an interactive input to the model. Using the component insertion tool, create an interactive variable by selecting the “Control Systems ► Control Blocks ► Controller Blocks” menu item and clicking on the canvas. Edit the properties of this interactive control block as follows:

Component Name: Interactive Heat Flux Controller

Constant One: $-1.E5 \text{ w/m}^2$

Variable Name: HeatFlux

The properties should appear as:

General ☐ Optional ☐ Disabled

Component Name	Interactive Heat Flux Controller	?
Control Block ID	-1	?
Type	[-9] Interactive Variable	?
Description	<none>	E ?
Comments	<none>	E ?
Gain	1.0 (-)	?
Minimum	-1.0E20 (w/m ²)	?
Maximum	1.0E20 (w/m ²)	?
Constant One	-1.0E5 (w/m ²)	?
Variable Name	HeatFlux	?

NOTE: Click on the units button (-) located adjacent to the value to change the units.

13. The next step is to add a heat structure to the pipe. To accomplish this, use the pipe's pop-up menu and select "Edit Heat Structures ► New." This will open a completion dialog for the heat structure. Enter the following values into the completion dialog:

Plane: Along Z
 Surface: Inner
 Thickness: 0.0220 m
 Temperature: 480.0 K
 Radial Nodes: 5

Note that the Material property has a selection button. Press the material selection button, select "Material 9 (Carbon A508)" from the list, and press the OK button to set the material type.

Select From Available Materials

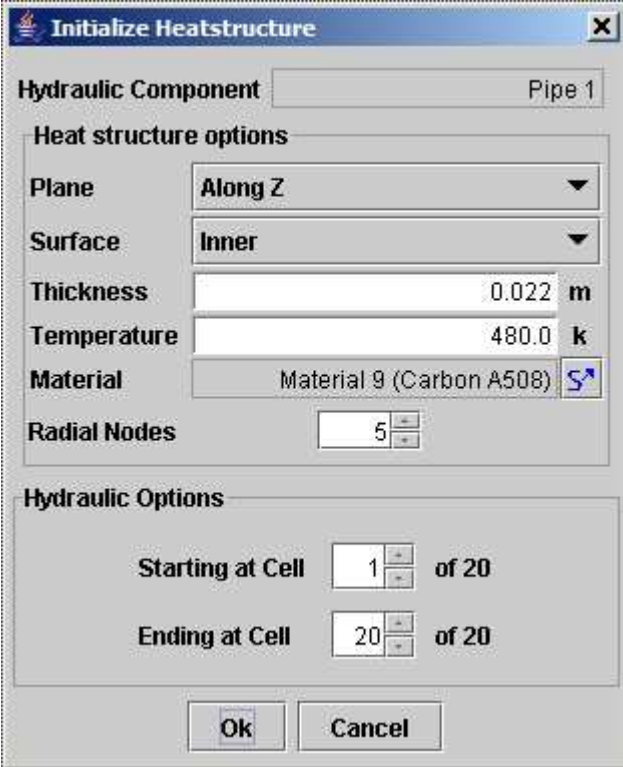
Category	Number	Component
Builtin	1	Material 1 (Mixed Oxide)
Builtin	2	Material 2 (Zircaloy)
Builtin	3	Material 3 (Gap Gases)
Builtin	4	Material 4 (Boron-nitride)
Builtin	5	Material 5 (Constantan/Nichr...
Builtin	6	Material 6 (Stainless 304)
Builtin	7	Material 7 (Stainless 316)
Builtin	8	Material 8 (Stainless 347)
Builtin	9	Material 9 (Carbon A508)
Builtin	10	Material 10 (Inconel 712)

Ok Cancel

Press the OK button at the bottom of the completion dialog. This will create the heat structure, and open it's properties dialog. Note that the color of the pipe changes to indicate that it includes a heat structure.

NOTES:

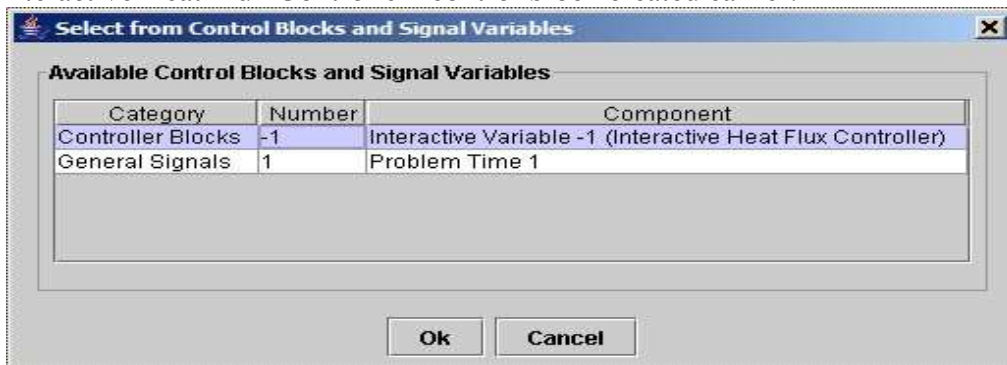
- By default, the heat structure will be connected the entire length of the pipe. The starting and ending cells for the heat structure can be set in the completion dialog.



The **Initialize Heatstructure** dialog box is shown. It has a title bar with a close button. The **Hydraulic Component** is set to **Pipe 1**. Under **Heat structure options**, **Plane** is **Along Z**, **Surface** is **Inner**, **Thickness** is **0.022 m**, **Temperature** is **480.0 k**, **Material** is **Material 9 (Carbon A508)**, and **Radial Nodes** is **5**. Under **Hydraulic Options**, **Starting at Cell** is **1 of 20** and **Ending at Cell** is **20 of 20**. At the bottom are **Ok** and **Cancel** buttons.

14. Now that the heat structure is created, the next step is to impose a constant heat flux boundary condition over the heated length of the pipe. To accomplish this, locate the “Axial Nodes / Surface BC's” property of the heat structure and open it's property editor by selecting the edit property button. Note that the inner surface of the heat structure is connected to the pipes cells and that the outer surface is adiabatic with a constant heat flux of 0.0. Using the mouse, select the outer surface boundary conditions for cell 3 and drag the mouse down to highlight the outer surface boundary conditions through cell 18.

With the outer surface boundary condition for cells 3 through 18 selected, change the boundary condition from “[0] Constant Heat Flux” to “[10] Controller Heat Flux” in the lower panel. Use the Control Signal property selection button to select the “Interactive Heat Flux Controller” control block created earlier:

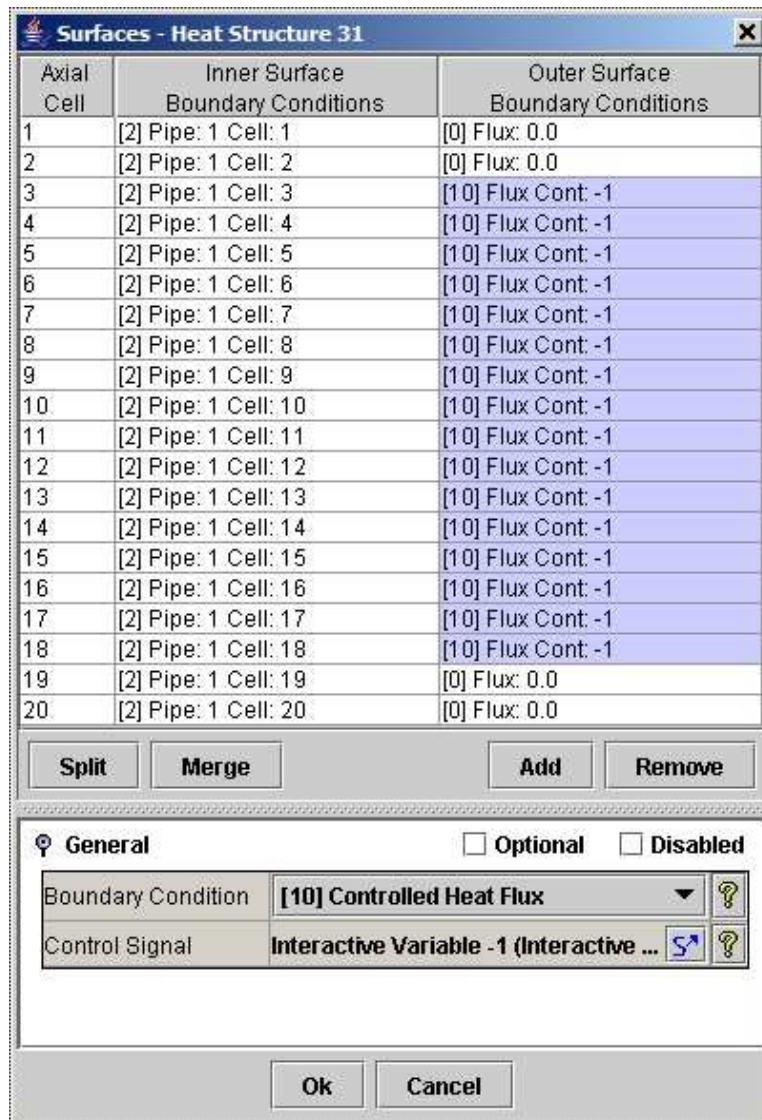


The **Select from Control Blocks and Signal Variables** dialog box is shown. It has a title bar with a close button. Under **Available Control Blocks and Signal Variables**, there is a table with the following data:

Category	Number	Component
Controller Blocks	-1	Interactive Variable -1 (Interactive Heat Flux Controller)
General Signals	1	Problem Time 1

At the bottom are **Ok** and **Cancel** buttons.

The surface boundary condition editor should now appear as follows:



Press the OK button to update the heat structure and close the editor.

15. At this point, the model should be essentially complete. To verify this, select the Check Model button on the main toolbar. The message window should display the following message:

“Error check complete. No errors found.”

If errors were found, correct any problems indicated in the error messages before continuing.

NOTES:

- Double clicking on an error message will select the problem component in the navigator.
- Model validation tests such as the elevation checker may be enabled and disabled by editing the Model Validation property of the Model Options navigator node.

16. Save the model using the “File ► Save” menu item. When prompted for a file name, enter Exercise1.med.

NOTES:

- ModelEditor files have a default extension of “.med”.

Optional Exercises:

1. Use the insertion tool to add text and/or graphic annotations to the display.
2. Change the inlet fill type to “[10] Gen State CS” where generalized-state parameters can be defined individually using control blocks. Create 7 interactive variables representing Pressure, Void Fraction, Liquid Mass Flow, Vapor Mass Flow, Liquid Temperature, Vapor Temperature, and NC partial pressure, respectively. Modify the fill's state controller to use these interactive control blocks.
3. Export the model to jEdit using the “Tools ► Export to jEdit” menu item.
4. Open the TRACE User's Manual from SNAP. Select a component on either the 2D View or in the navigator. Using the right-click pop-up menu, select the “Reference Docs ► User's Manual” item. The TRACE user's manual should open to the section describing input requirements for the selected component.

Exercise 2. Submit the Model.

This exercise is designed to familiarize the analyst with the basic functionality of SNAP's runtime. In this exercise we will submit the model created in the previous exercise to SNAP's runtime and use the Job Status application to view the output of the calculation as well as delete the calculation. This exercise requires the SNAP Calculation server to be running. If the server is not setup to start automatically as a Windows Service or using a UNIX init script, start the Server using the Configuration Tool prior to starting this exercise.

1. Open the model created in the last exercise using the “File ► Open” menu item (or use the Exercise1.med file provided with this tutorial). The model should open, displaying the view displayed when the model was closed.
2. Before we submit the run, we need to check the time step data to ensure we have reasonable values. To do this, select the “Model Options” node for this model in the navigator. Locate the “Timestep Data” property in the main properties window (near the bottom of the list) and open its' editor. Verify that the values match those shown below:

End Time	Minimum Size	Maximum Size	Heats Fluid Size	Max Conv. Power Diff	Long Edit Interval	Graphics Interval	Restart Interval	Short Edit Interval
100.0	1.0E-6	1.0	10.0	0.0	100.0	1.0	100.0	1.0

3. We are now ready to run the calculation. Select the “Tools ► Submit Job” menu item. This will open the adjacent dialog:

Verify that the appropriate executable is selected. Each plug-in may have several executables defined, this setting determines which of the defined executables will be used for this run. Also verify that the “Start Paused” run option is set to false and that the “View Console Output” option located near the bottom of the dialog is selected.

4. Press the Submit button at the bottom of the dialog. This will close the submit dialog and open a console window that will echo the console output generated by the calculation.



Submit Calculation

Executable: V4150

Run Options

Name: ☒ Exercise2

Priority: 5

Start Paused: ☐ True ☒ False

No CPU: ☐ True ☒ False

No Dif: ☐ True ☒ False

No Rand: ☐ True ☒ False

Run Stats: ☐ True ☒ False

Parametric

☐ Enabled

First: < none >

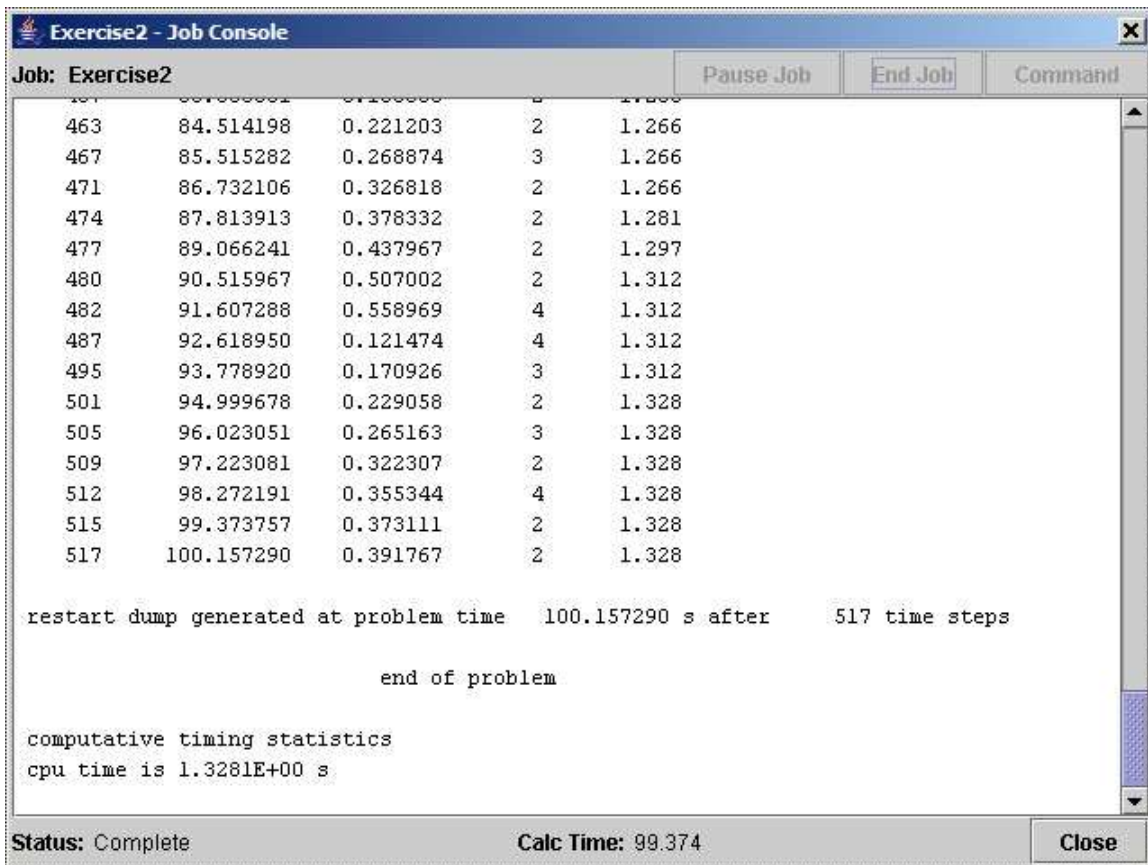
Second: < none >

Restart

Restart File:

☒ View Console Output

Submit **Cancel**



Exercise2 - Job Console

Job: Exercise2

Pause Job End Job Command

463	84.514198	0.221203	2	1.266
467	85.515282	0.268874	3	1.266
471	86.732106	0.326818	2	1.266
474	87.813913	0.378332	2	1.281
477	89.066241	0.437967	2	1.297
480	90.515967	0.507002	2	1.312
482	91.607288	0.558969	4	1.312
487	92.618950	0.121474	4	1.312
495	93.778920	0.170926	3	1.312
501	94.999678	0.229058	2	1.328
505	96.023051	0.265163	3	1.328
509	97.223081	0.322307	2	1.328
512	98.272191	0.355344	4	1.328
515	99.373757	0.373111	2	1.328
517	100.157290	0.391767	2	1.328

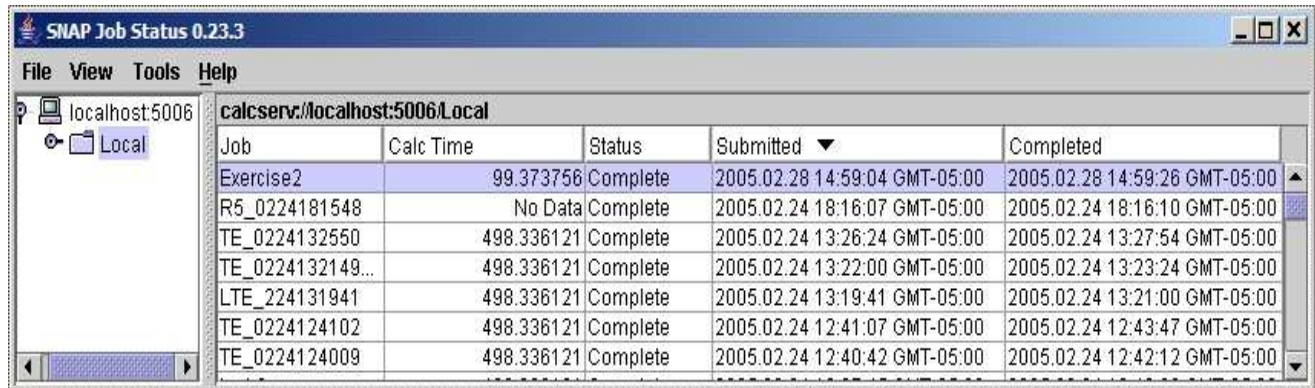
restart dump generated at problem time 100.157290 s after 517 time steps

end of problem

computative timing statistics
cpu time is 1.3281E+00 s

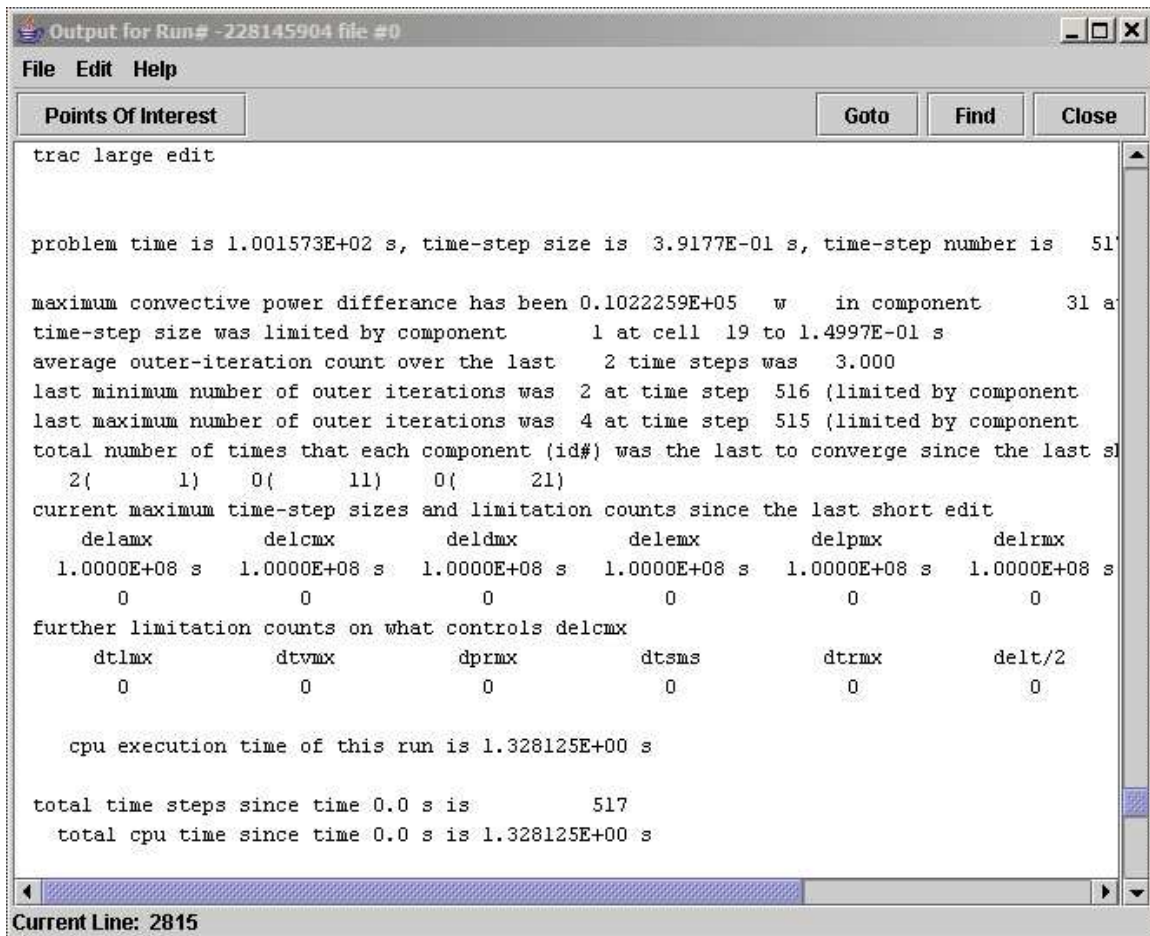
Status: Complete **Calc Time:** 99.374 **Close**

- Close the console window when the calculation completes.
- Now open the Job Status application by selecting “SNAP ► Job Status” from the Windows Start menu:



Job	Calc Time	Status	Submitted	Completed
Exercise2	99.373756	Complete	2005.02.28 14:59:04 GMT-05:00	2005.02.28 14:59:26 GMT-05:00
R5_0224181548	No Data	Complete	2005.02.24 18:16:07 GMT-05:00	2005.02.24 18:16:10 GMT-05:00
TE_0224132550	498.336121	Complete	2005.02.24 13:26:24 GMT-05:00	2005.02.24 13:27:54 GMT-05:00
TE_0224132149...	498.336121	Complete	2005.02.24 13:22:00 GMT-05:00	2005.02.24 13:23:24 GMT-05:00
LTE_224131941	498.336121	Complete	2005.02.24 13:19:41 GMT-05:00	2005.02.24 13:21:00 GMT-05:00
TE_0224124102	498.336121	Complete	2005.02.24 12:41:07 GMT-05:00	2005.02.24 12:43:47 GMT-05:00
TE_0224124009	498.336121	Complete	2005.02.24 12:40:42 GMT-05:00	2005.02.24 12:42:12 GMT-05:00

- Locate and select the calculation that was just submitted. Select “View Output ► Output File” from the right-click pop-up menu. This will open the output viewer shown below.



```

trac large edit

problem time is 1.001573E+02 s, time-step size is 3.9177E-01 s, time-step number is 51

maximum convective power difference has been 0.1022259E+05 w in component 31 a
time-step size was limited by component 1 at cell 19 to 1.4997E-01 s
average outer-iteration count over the last 2 time steps was 3.000
last minimum number of outer iterations was 2 at time step 516 (limited by component
last maximum number of outer iterations was 4 at time step 515 (limited by component
total number of times that each component (id#) was the last to converge since the last sl
2( 1) 0( 11) 0( 21)
current maximum time-step sizes and limitation counts since the last short edit
delamx delcmx deldmx delemx delpmx delrmx
1.0000E+08 s 1.0000E+08 s 1.0000E+08 s 1.0000E+08 s 1.0000E+08 s 1.0000E+08 s
0 0 0 0 0 0
further limitation counts on what controls delcmx
dtlmx dtvmx dprmx dtsms dtrmx delt/2
0 0 0 0 0 0

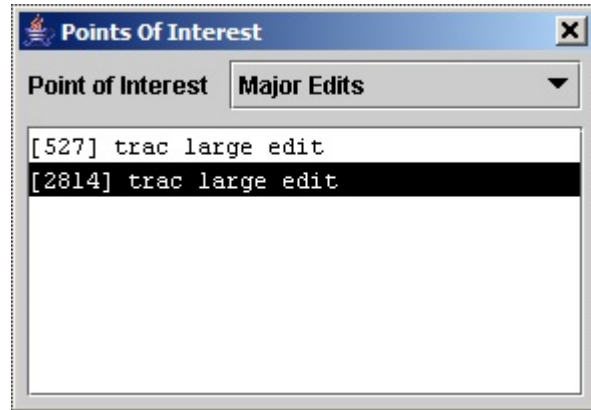
cpu execution time of this run is 1.328125E+00 s

total time steps since time 0.0 s is 517
total cpu time since time 0.0 s is 1.328125E+00 s

```

Current Line: 2815

8. Press the “Points of Interest” button and select “Major Edits” from the list. This will display a list of Major Edits. Select specific major edits in the list and verify that the output viewer repositions to those locations.



9. Close the Output Viewer.
10. Using the right-click pop-up menu for the calculation, select the Delete menu item. This will prompt you to confirm that you wish to delete the calculation. Press OK and notice that the calculation is removed from the list of calculations.
11. Resubmit the calculation. Notice that it appears in the Job Status application.

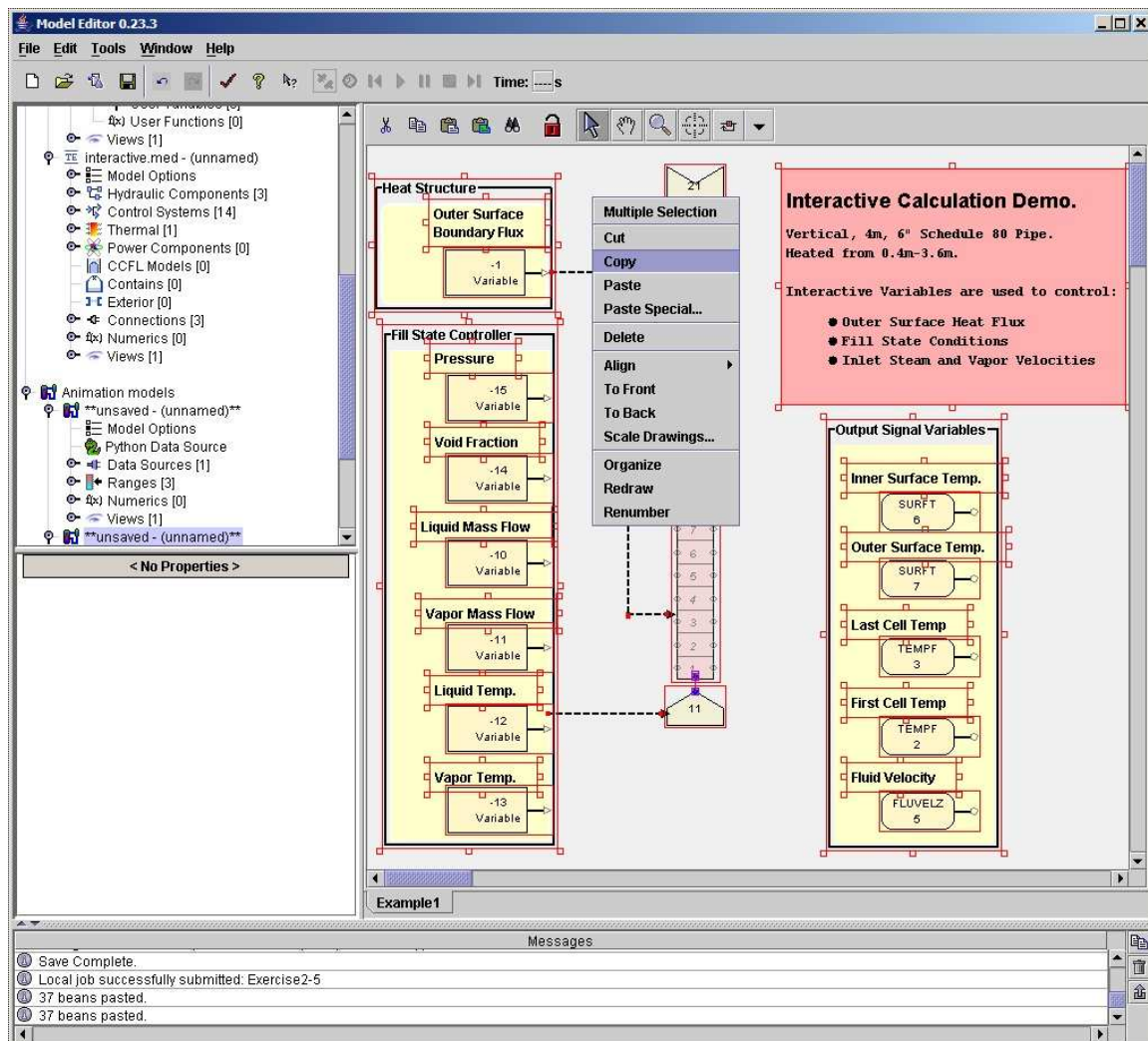
Optional Exercises:

1. Submit the calculation with “Start Paused” set true. Resume and pause the calculation using the buttons located at the top of the job console window.
2. Plot the results of the calculation. Select the calculation in Job Status and select Plot from the right-click popup menu. This will open AcGrace with the calculation's graphics file loaded. Select the “Edit ► TRAC Data” menu item in AcGrace. This will open a list of data channels that may be selected for plotting.

Exercise 3. Creating an Animation Display.

This exercise is designed to familiarize the analyst with the basic functionality of SNAP's interactive and post-processing capability. The following steps will guide you through creating an animation display using the model created in exercise 1:

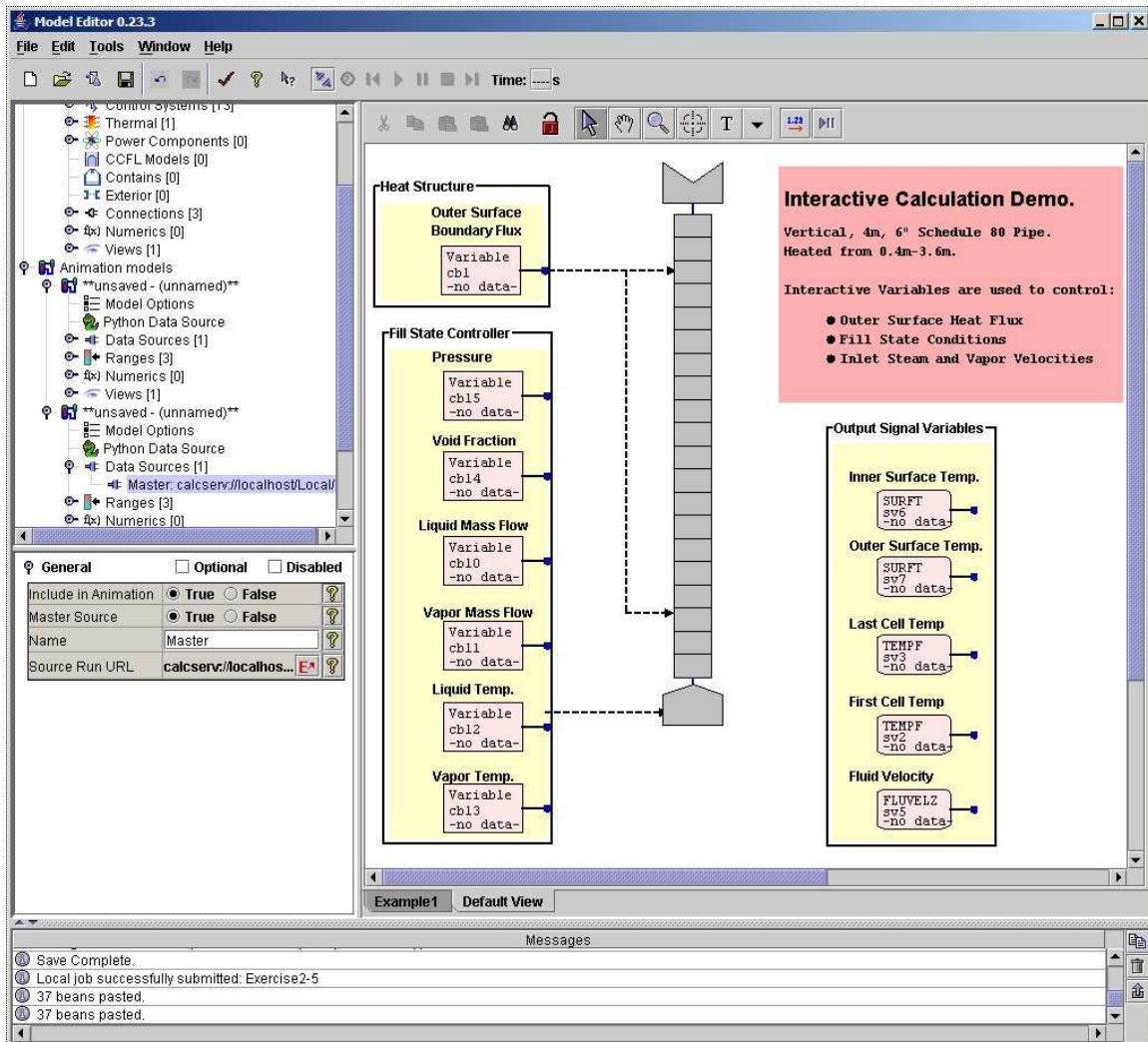
1. Open the model created in exercise 1 (or use the Exercise1.med file provided with this tutorial) using the “File ► Open” menu item. The model should open displaying the view displayed when the model was closed.
2. Select all of the components in the view using the right-click “Select ► All” menu item.
3. With all components in the view selected as shown below, use the view's right-click “Copy” menu item to copy the components.



4. Create a new model using the “File ► New” menu item. Select “Animation Model”

when prompted for the model type press the OK button to create a new Animation model.

- When the empty view appears, paste the contents of the TRACE model into the animation using the view's right-click Paste menu item. This will create the initial animation view.



Any number of additional items can be added to enhance the view including data values, interactive controls, strip charts, ranges, etc. These can all be added using the insert tool as we did in exercise 1 to add components to the TRACE model.

- First we need to connect to the calculation that was submitted in Exercise 2. This will provide us with a working set of data channels and interactive variables for the calculation and help simplify development of the interactive view. Locate and expand the Data Sources node of the animation model in the navigator. Select the data source labeled "Master:calcserv://localhost/Local."

Locate the “Source Run URL” property for this data source and open its editor. Select the Local folder under the localhost:5006 node. This should display a list of the calculations similar to those shown in the Job Status application. Select the calculation that was submitted in Exercise 2 and press the OK button.

7. Press the “Connect to Data Sources” button located on the main toolbar to activate the connection.
8. Using the insertion tool, select “Interactive ► Playback Controls”. The cursor will change to crosshairs. Click on the view to place the interactive controls. The controls can be easily repositioned using the select tool after placement on the view.
9. Next use the insertion tool to place an “Interactive Value” onto the view directly under the display bean for the heat flux control block (cb1). The interactive value control is used to change the value of interactive control blocks during a calculation. Modify the following properties of the interactive value using the selection editors for each:

Channel Name:	cb1
Variable Name:	cb 1:[HeatFlux]

10. SNAP uses Ranges to map components to map component data and Thermal-Hydraulic conditions to color. Expand the ranges node for the animation model. Select the fluid condition range and examine its' properties. The component provides a mapping of thermal hydraulic condition over the subcooled, saturated and superheated regions. The colors at the ends of each region are supplied along with the minimum subcooled temperature and the maximum superheat temperature.

Using the right-click pop-up menu for the fluid condition range in the navigator, select “Add to View ► Default View”. This will add the range to the view. Resize and reposition it appropriately.

11. Using the insertion tool, add a flow indicator to the display. Place it at between the outlet of the pipe and the inlet of the break. Set its Channel Name property to liquid velocity at the exit of the pipe (i.e. “vln-0010021”).
12. Using the insertion tool, add a data value (located under the indicators menu) to the display and set its properties as follows:

Channel Name:	time
Numerical Format:	%10.2f

NOTE: Data value use the C-language scanf standard for numerical formats.

13. Add a strip chart to the display and set its properties as follows:

Graph Title:	Void Fraction
Time Data Channel:	time
Data Channel 1:	alpn-001001
Data Channel 2:	alpn-001010
Data Channel 3:	alpn-001020

Show Scrollbar: False
Time Base: 10 min.

14. Test the animation using the animation controls on the main menu bar.
15. Locking an animation view will prevent inadvertent repositioning of components and will activate any interactive controls contained in the view. Lock the view using the view's lock button. Test the animation controls contained in the view.
16. Save the animation model using the "File ► Save" menu item. When prompted for a file name, enter Exercise3_anim.med.

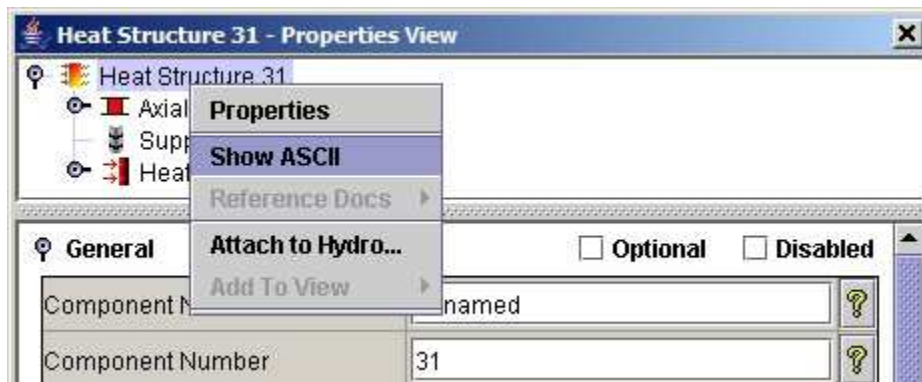
Optional Exercises:

1. Use the insertion tool to add text and/or graphic annotations to the display.
NOTE: Text annotations can include HTML formatting tags. This can be useful to create multiline annotations and to display sub/super-scripting.
2. Add an interactive pop-up command menu to the display bean for Control Block cb1.
3. Change time step data in the TRACE model to give it an end time of 1.e6 seconds and submit the calculation with "Start Paused" set true. Connect to the calculation with the animation model and vary the interactive heat flux. Observe changes to the axial void profile. Terminate the calculation interactively using the time step controls.

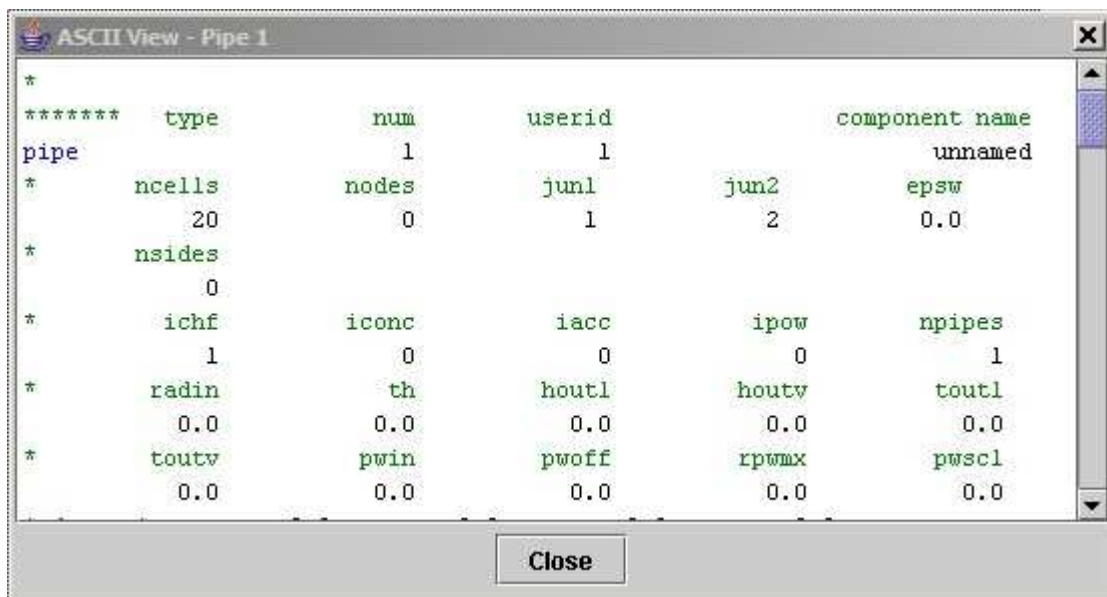
Exercise 4. Component Renodalization.

This exercise is designed to familiarize the analyst with the renodalization capabilities of SNAP. In this exercise we will renodalize the model created in exercise 1.

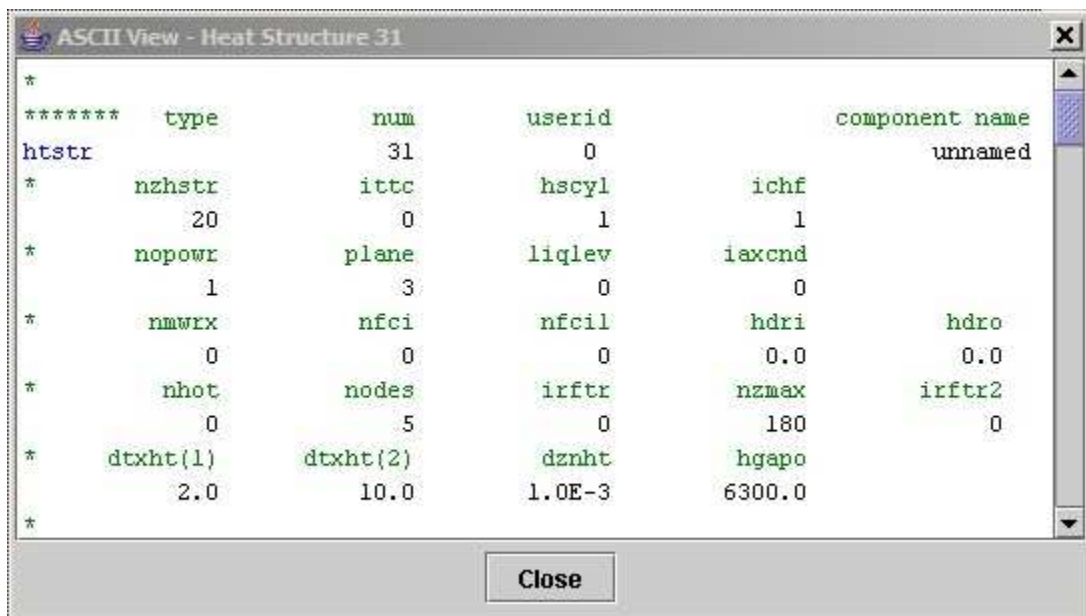
1. Open the model created in exercise 1 (or use the Exercise1.med file provided with this tutorial) using the “File ► Open” menu item. The model should open displaying the view displayed when the model was closed.
2. Open the ASCII view of the pipe using the right-click “Show ASCII” menu item. Also open the heat structure editor using the right-click “Edit Heatstructures ► Heat Structure xx” menu item. Open the ASCII view of the heat structure using the right-click “Show ASCII” menu item located on the mini-navigator at the top of the heat structure property view:



3. The ASCII views for the pipe should appear as:

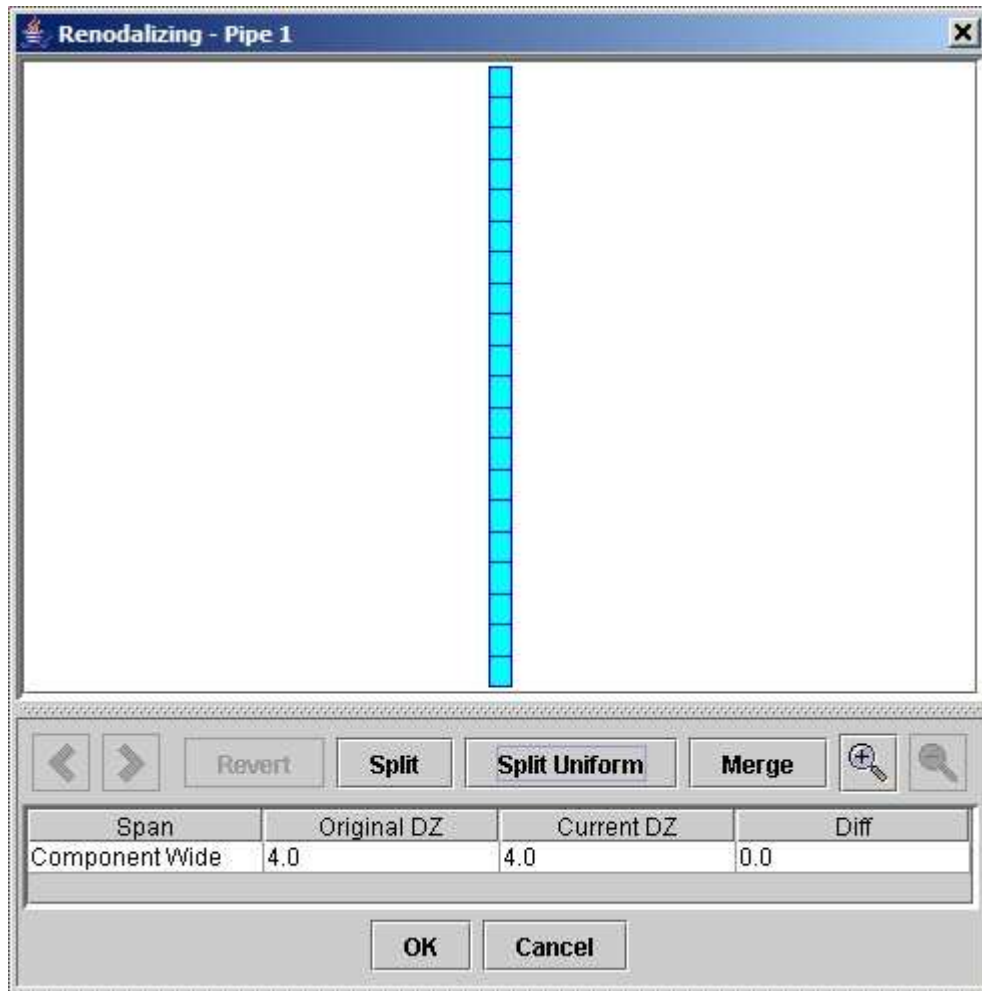


and the ASCII views for the heat structure should look something like:



Note that the heat structure and pipe both contain 20 axial cells or nodes. Review the ASCII input to confirm that the heat structure is connected to the pipe as expected.

4. Select the “Renodalize” menu item from the pipe's right-click pop-up menu.
5. Select the top cell of the pipe with the cursor and then select the bottom cell of the pipe while holding down the shift key. This will highlight the entire pipe as shown in the following dialog:



6. Press the “Split Uniform” button and enter 2 into the following dialog when prompted then press the OK button.
7. Press the OK button located on the renodalization dialog to complete the process. Review the ASCII views of the pipe and heat structure. Verify that both the components are renodalized as expected.

Optional Exercises:

1. Update the animation display from Exercise 2 to reflect the new nodalization.
2. Submit the renodalized model.

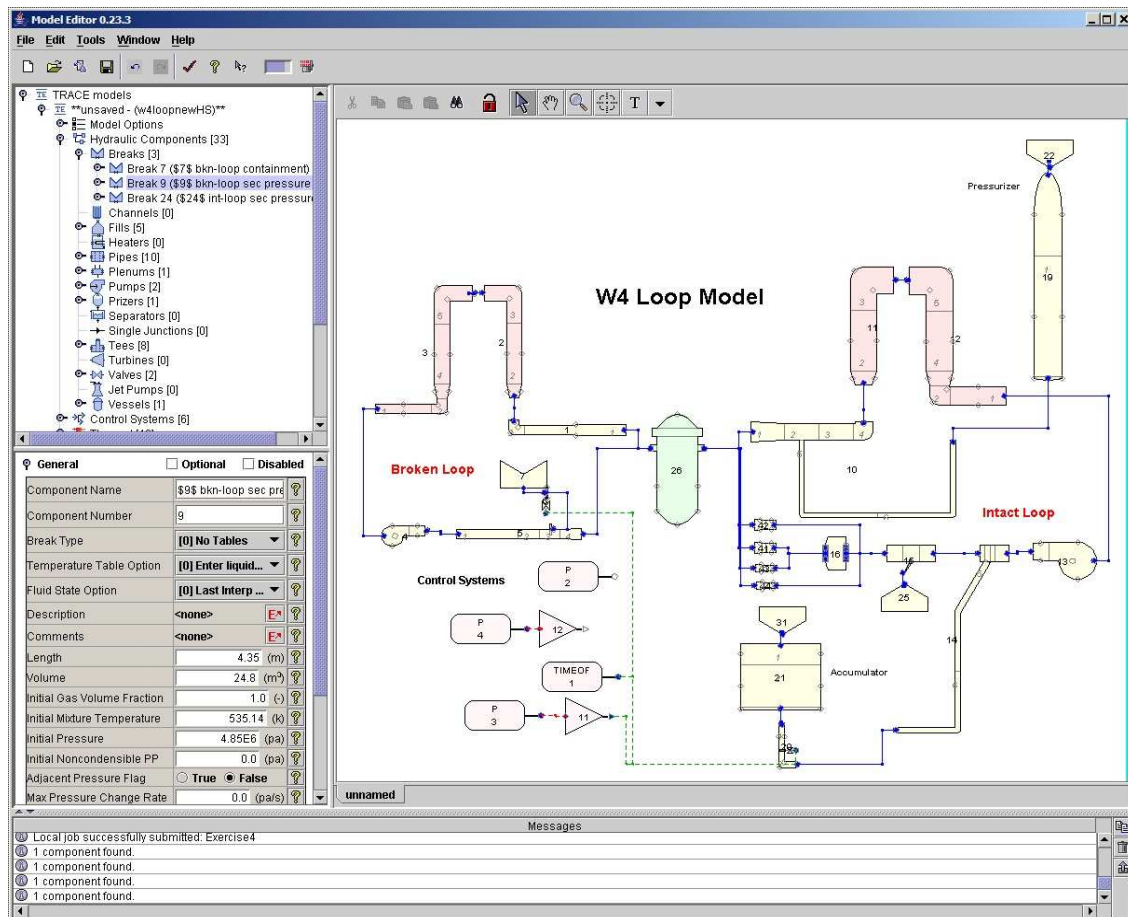
Exercise 5. Working with View Templates

This exercise is designed to familiarize the analyst with importing existing models and View Templates. View Templates provide a means of saving a layout of a view that may then be applied to a similar model.

In this exercise the analyst will use a sample View Template to layout an imported model.

1. Import the sample TRACE input deck w4loop.inp provided with this tutorial by using the "File ► Import ► TRACE ► ASCII" menu item.
2. When prompted, close the Create Views dialog without creating Views by either using the close dialog button for the window or by pressing the Escape key.
3. Create a new empty View by right-clicking on the Views node in the Navigator and selecting the "New" menu item. This should create and open a new View with the name "unnamed" and display its properties in the Main Property View.
4. Next, import a View Template by opening the right-click pop-up menu in the View and selecting the "Tools ► Import View Template" menu item. Select the file Primary.mvt provided with this tutorial and press OK.

This will apply the template to the view which should now appear as shown below:



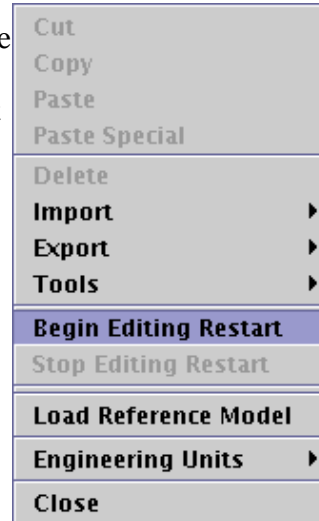
NOTES:

- To export a View Template of an existing view, use the "Tools ► Export View Template" item of the right-click pop-up menu in the View.

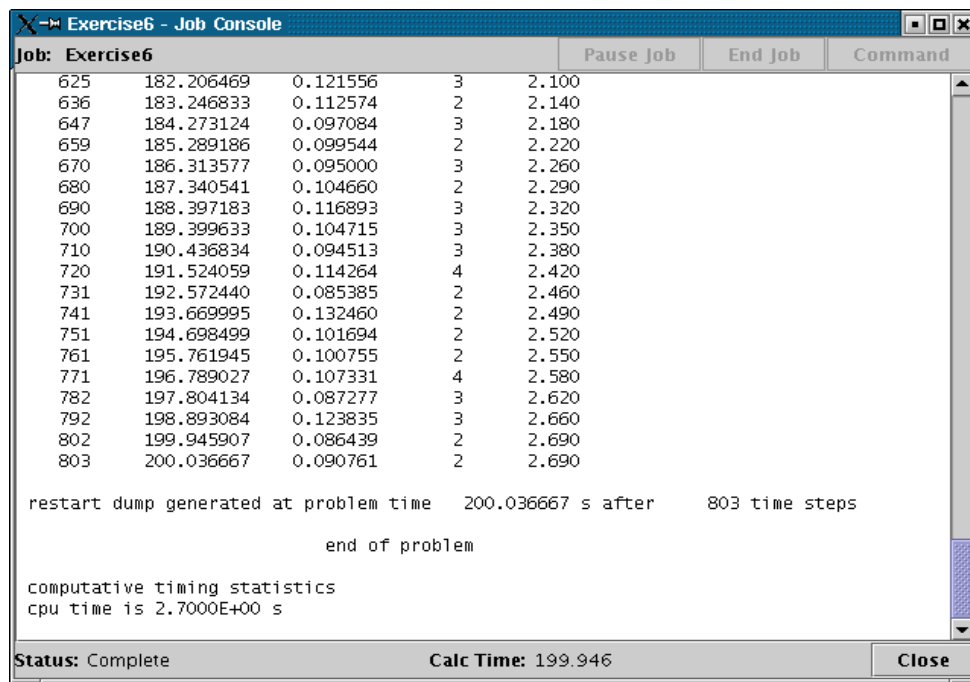
Exercise 6. Restarting a TRACE Calculation

This exercise is designed to familiarize the analyst with the process of creating a restart case and submitting the restart to the calculation server. In this exercise we will modify the model created in exercise 1 and submit the changes as a restart to the job run in exercise 2. The following steps will guide you through modifying a model and submitting the changes as a restart.

1. Open the model created in exercise 1 (or use the Exercise1.med file provided with this tutorial) using the “File ► Open” menu item. The model should open, displaying the view displayed when the model was closed.
2. Right click on the model node in the navigator and select the "Begin Editing Restart" item as shown to the right. The text color of the model node will change to red to indicate that it is currently editing a restart. Editors for properties that cannot be changed for a restart will be disabled.
3. Select the renodalize option from the right-click pop-up menu of pipe 1.
4. Select all of the cells inside the pipe by clicking on the top cell, holding the shift key down and clicking on the bottom cell.
5. Press the “Split Uniform” button and enter 2 into the following dialog when prompted then press the OK button.
6. Review the ASCII views of the pipe and heat structure. Verify that both the components are renodalized as expected.
7. Observe that in the Navigator, the text of the "Model Options" node and the node for pipe 1 has changed to red. The heat structure, and any signal variables attached to either the pipe or the heat structure should also appear red in the navigator. This indicates that there are restart changes to these components.
8. Extend the end time of the model. To do this, select the “Model Options” node for this model in the navigator. Locate the “Timestep Data” property in the main properties window (near the bottom of the list) and open its editor. Add 100 seconds to the "End Time" value.
9. Locate the “Restart Number” property in the main properties window and set its value to -1 to choose the last restart dump.
10. Select the “Tools ► Export to jEdit” item from the from the right-click popup-menu of this model's node in the Navigator. Observe that only the input required for a restart and those components with restart changes have been exported.



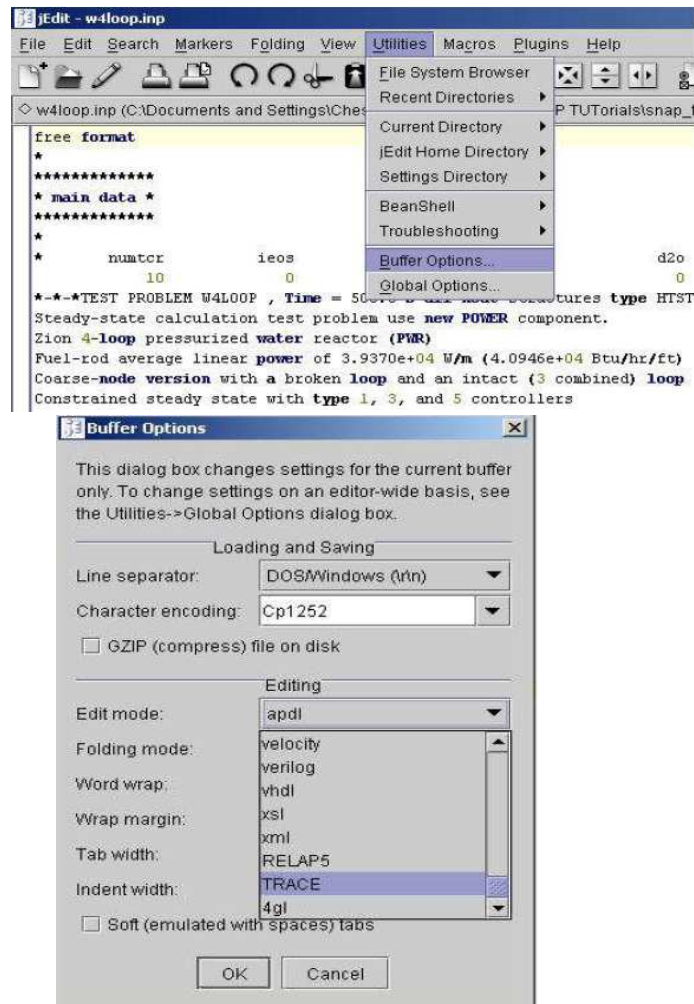
11. We are now ready to run the restart calculation. Select the “Tools ► Submit Job” menu item. This will open the adjacent dialog:
12. Verify that the appropriate executable is selected. Each plug-in may have several executables defined, this setting determines which of the defined executables will be used for this run. Also verify that the “Start Paused” run option is set to false and that the “View Console Output” option located near the bottom of the dialog is selected.
13. Select the restart file. Press the selection button adjacent to the Restart File text box and select the trcptpr file produced when the job was submitted in exercise 2. This file will be copied over with the tracin file and renamed to trcrst.
14. Press the Submit button at the bottom of the dialog. This will close the submit dialog and open a console window that will echo the console output generated by the calculation.



Exercise 7. Using the jEdit Text Editor

This is an optional exercise designed to demonstrate the use of the jEdit text editor with SNAP. Jedit is a very powerfull (and free) text editor available on many computer platforms. Like SNAP, jEdit uses plugin modules to support various features. A SNAP plugin for jEdit is included with the SNAP distruibution. Installation of jEdit is described in the installation instructions included with SNAP. To get full use of the features discussed in this demonstration, it is very important to follow the installation instructions carefully!

1. Start jEdit.
2. Open an exisiting TRACE deck. The file “w4loop.inp” is provided with this tutorial, open “w4loop.inp” in jEdit.
3. If jEdit has been configured correctly, you will notice that jEdit highlights comments and TRACE keywords automactically. If jEdit is not highlighting the TRACE keywords you need to change the “buffer option edit mode”, as shown in the following figures.



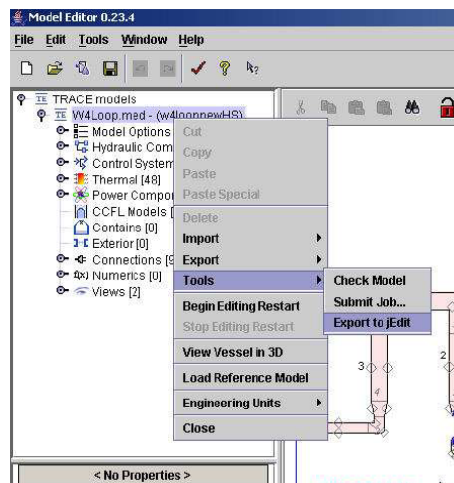
Once “TRACE” is selected in the preceding dialog, click the “OK” button. The jEdit window showing the w4loop.inp deck should now be using TRACE specific highlighting, as shown in the following figure.

```

free format
*
*****
* main data *
*****
*
*      numctr      ieos      inopt      nmat      dzo
*      10          0        1          0          0
*
*--*TEST PROBLEM W4LOOP , Time = 500.0 s all heat structures type HISTR
Steady-state calculation test problem use new POWER component.
Zion 4-loop pressurized water reactor (PWR)
Fuel-rod average linear power of 3.9370e+04 W/m (4.0946e+04 Btu/hr/ft)
Coarse-node version with a broken loop and an intact (3 combined) loop
Constrained steady state with type 1, 3, and 5 controllers
Two Stgen components replaced by four Htstr, Pipe, and Tee components
Based on an isothermal (primary, secondary, & ecs temperatures differ),
isobaric (pressurizer pressure), no-flow, and no-power input condition
The solution estimate is initialized by hydraulic-path steady-state data
*
* #####          #####          #####
* # fill #        # fill #        # fill #
* # 23 #          # 22 #          # 8 #
* #####          #####          #####
*      24          23              9
* #####          #####          #####
* # tee #          #prizer#        # tee #
* # 18 #          # 19 #          # 17 #
* ##### rods      #####          rods #####

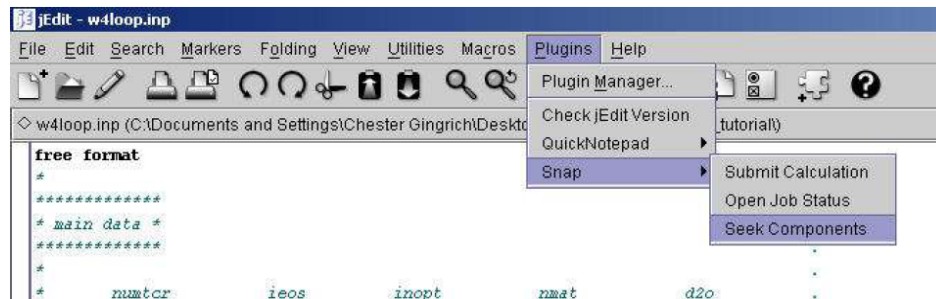
```

- Note that the SNAP model editor also provides a direct “export to jEdit” feature, accessible from the right-click menus of the model editor's navigator and the 2D Views, as shown in the figure below.

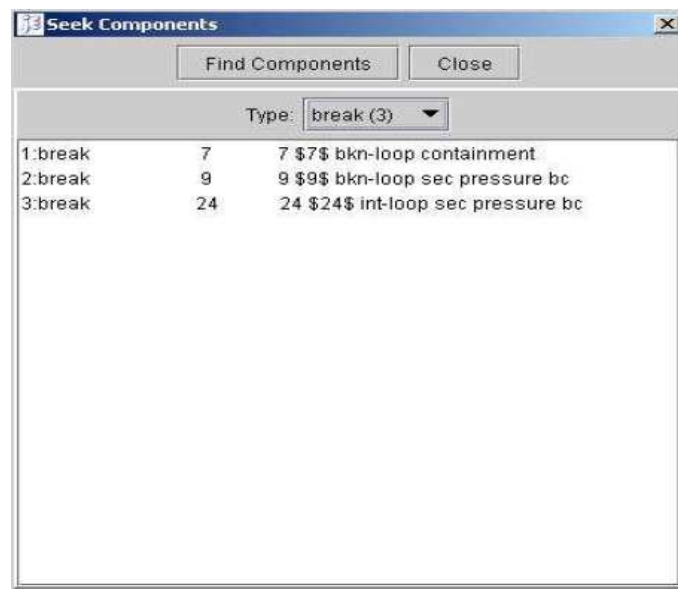


(Note that while the model editor allows you to modify the deck in the exported text file, any changes you make to the text file are NOT automatically re-introduced into the model editor's version of the model!)

5. Find components feature in jEdit. The jEdit SNAP plugin provides several special features available to TRACE and RELAP5 code users. To access these features, use the “plugins -> SNAP” menu item as shown in the figure below.



6. Select the “Seek Components” menu item. A window like the one shown below will appear.



7. By selecting the “Type” button in the dialog shown in the above figure, the user will be presented with a list of the available components of that type in the current deck. The user may select the specific component of interest from the displayed list and jEdit will automatically move the view and cursor position to that component in the deck!
8. Once modifications are made to the input deck the user may submit a job to the calculation server, just as was done in Exercise 2 of this tutorial. The “Submit Calculation” menu item under the “Plugins-> SNAP” menu accesses this feature.
9. 7. “Job Status” is also available through the jEdit SNAP plug-in, providing a

convenient way of monitoring and viewing results of a calculation.

10. Jedit has many features, well worth exploring on your own. Noteworthy features include:

- Split windows. Windows inside the jEdit view can be split horizontally and/or vertically. Useful for comparing different sections of a deck, or comparing two different decks side by side.
- Hypersearch. This feature allows the user to search a whole directory for files containing specified words. The locations of the words found are displayed in a window and the user may then select which locations he wishes to view and/or edit.
- Folding. Allows the user to “fold” or hide a portion of the text but doesn't remove the text from the document. Folds may be nested within each other. Folding permits a user to write in an “outline” style, accessing details by expanding a fold as needed.