PWR Model Setup – Exercise 3

OBJECTIVES

- Become familiar with the VESSEL, POWER and fuel rod HTSTR components .
- Renodalize the VESSEL and fuel rod HTSTR components to make them consistent with the TRACE guidelines for power plant models.
- Learn how to implement axial and core-wide radial power profiles.
- Understand the influence of the fuel rod gap conductance on the fuel rod temperatures.

OVERVIEW OF STEPS

1. Run a 200 s steady-state base calculation with the PWR model.

2. Vessel Component Renodalization.

3. Fuel Rod Heat Structure Renodalization

A) Radial Nodalization

B) Adding a Supplemental Hot Rod

4. Power Component Modifications

A) Total Core Power

B) Input a top-skewed axial power profile

C) Input a flat fuel pin radial power profile

D) Input a flat core-wide radial power profile

5. Obtain the Desired Fuel Center Line Temperature

STEP 1. RUN A 200 S STEADY-STATE BASE CALCULATION WITH THE PWR MODEL.

A steady-state base calculation will be made that will be used to compare the results of modifications made to the model in this exercise.

1. Locate and open the PWR model in the SNAP environment:

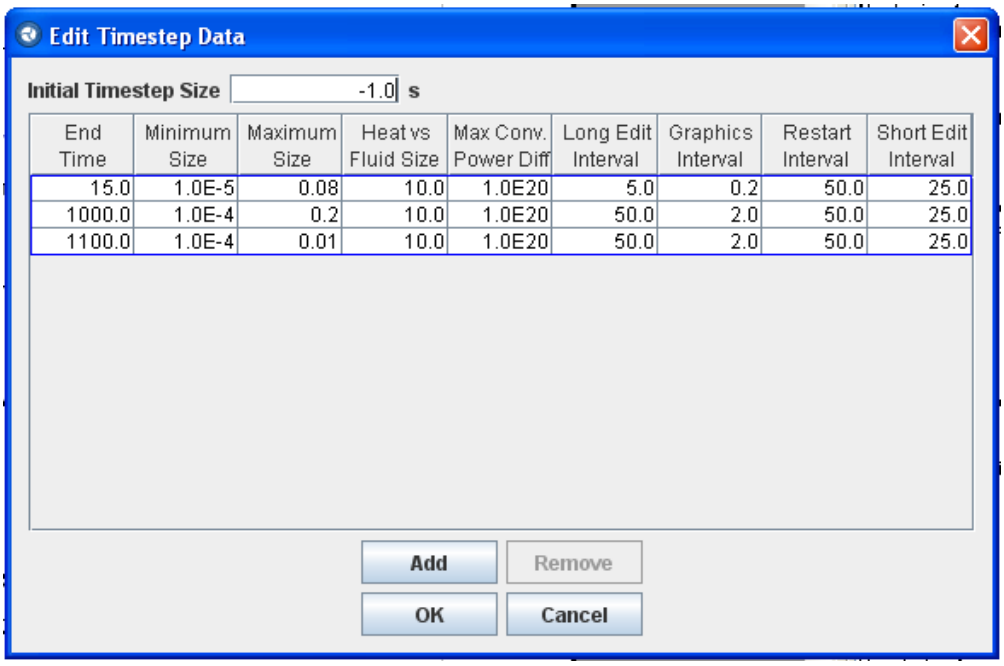
a) Go to the Day3/Morning/Plant_Exercise_Part_3 folder and double click on the PWR-SS-Ex3.med file. The SNAP model editor opens with the PWR steady-state model.

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.

2. Modify the end time.

a) Locate and click on "Model Options" in the Navigator Window.

b) In the Properties Window locate the "Timestep Data" box and expand  the box. The Edit Timestep Data popup window will appear.



The last data block of input information is the timestep data cards for controlling the calculation and output edits. The problem time span to be evaluated is separated into time domains. Each domain may have different minimum and maximum timestep sizes and output-edit time intervals. Any number of time domains may be input.

End Time (TEND) - End time (s) for this time domain.
Minimum Size (DTMIN) - Minimum timestep size (s) for this time domain.
Maximum Size (DTMAX) - Maximum timestep size (s) for this time domain.
Heat vs Fluid Size (RTWFP) - Ratio between heat-transfer and fluid-dynamics timestep sizes.
Max Conv. Power Diff (POWERC) - Maximum convection-power difference (W, Btu h-l) between what goes into the fluid and what comes from the wall in the convection heat-transfer calculation.
Long Edit Interval (EDINT) - Long-printout-edit time interval (s) for this time domain.
Graphics Interval (GFINT) - Graphics-edit time interval (s) for this time domain.
Restart Interval (DMPINT) - Dump/restart-edit time interval (s) for this time domain.
Short Edit Interval (SEDINT) - Short-printout-edit time interval (s) for this time domain.

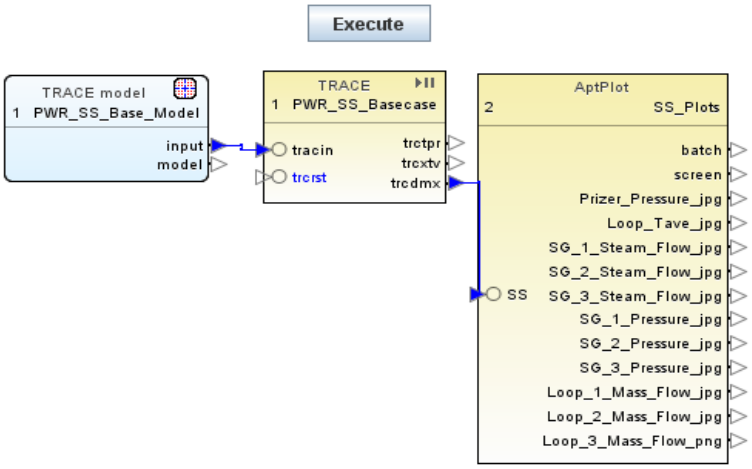
c) Click on the last End Time data entry row and click on the Remove button at the bottom of the popup window.

d) Change the End Time on the last row of time step data to 200.0. This change will end the calculation at 200 seconds.

e) Click on the OK button at the bottom of the window.

3. At the bottom of the View Window are several tabs. Each tab shows all or part of the PWR model. Additional tabs can be viewed by clicking on the left or right arrow buttons at the right hand side of the of the tabs.

4. Locate and click on the Job Stream tab. This view window shows the job stream used to submit the PWR model for a TRACE execution and to make plots of some of the key parameters.



Clicking on the execute Execute button submits the job when the SNAP view window is locked. The TRACE model box is the TRACE input file. The TRACE box contains information that connects the model to the code, allows the use of an animation model and demultiplexs the plot file when the calculation is complete. The AptPlot box shows the calculated parameters that have been included to be plotted.

5. Currently, the SNAP view window is not locked. Click on the execute button and do the following.

a) In the Properties Window, give the submitted job a name of PWR_SS_Step1 (that name may already be there, if so do nothing). The name can be unique to distinguish this run from other runs that are made.

b) Verify that the "View in Job Status" is marked Yes.

6. Click on the TRACE box in the View Window. In the Properties Window do the following:

a) Verify that V5840 is selected in the Applications box.

b) Verify that the "View in Job Status" is marked Yes.

c) Verify that the Demultiplex Box is checked Yes

d) The job stream is set up to use an animation file to view the status of the calculation. Verify that the Open Animation box is marked "immediately".

e) Verify that the Start/Paused button is turned on.

7. Lock the View Screen by clicking on the  located in the left-hand side of the Toolbar.

8. Click on the "Execute" button located in the Job Steam View window. This submits the job for execution. A Submit Stream popup window appears. Click on the OK button to continue.

9. The SNAP Job Status window will appear and the calculation will start the initialization process. The animation mask will appear as another session in the Model Editor Screen with a popup window that requests to Resume the Calculation. Click on the Yes button. The calculation will now proceed.

10. View the progression of the calculation using the System View tab and the Steady-State Plots tab located at the bottom of the Animation View Window in the Model Editor.

A base calculation has been run that will be used for comparisons later on in this exercise. This completes Step I. Continue to the next step.

Step 2. Vessel Component Renodalization.

This step will renodalize the core region of the VESSEL component to be consistent with TRACE modeling guidelines for transient simulations.

The VESSEL component should be divided into enough axial, radial and azimuthal cells to best describe the expected phenomena that occurs during a postulated accident scenario. For PWR modeling, there should be enough azimuthal sectors to provide one sector for each hot leg connection and one sector for each cold leg connection. For example, a three-loop power plant, the VESSEL component should contain six sectors. There should be enough radial rings to describe the vessel downcomer flow path and to capture the core-wide radial power profile. Typically, one radial ring is used to describe the downcomer flow path and three rings to describe the radial power profile in the core region. There should be sufficient axial levels to adequately describe the fluid behavior in the lower plenum, the core, the upper plenum and in the upper head. Typically, two levels below the core provide sufficient modeling of the turning of the downcomer flow below the elevation of the of the bottom of the downcomer skirt, losses of the radially-inward flow through the core support columns and losses of the upward flow through the lower core support plate. A moderately-detailed axial representation of the fluid behavior in the core region is accomplished with 12 one-foot vertical levels. Typically, two axial levels are placed between the top of the core and below the coolant loop nozzles to allow for fluid separation in that region. Sufficient levels in the vessel upper plenum and upper head to permit realistic modeling of the guide tube flows and the fluid temperature response.

For training purposes, the PWR VESSEL model being used in this workshop does not meet all of these detailed criteria. However, some modifications of the VESSEL are needed to better describe the fluid behavior in the core region for the transient calculation exercise to be performed later in the workshop.

1. Go to the PWR-SS-Ex3.med file in the Model Editor.

2. Unlock the View Window by clicking on the lock ICON located in the Toolbar on the left-hand side.

3. Click on the Hydro Comps tab at the bottom of the View Window. This view shows all of the hydrodynamic components used in the PWR model. Locate and click on the VESSEL component in the View Window.

4. In the Properties Window, scroll down to the Boundary Interfaces 

input section and expand it by clicking on the arrow on the left-hand side. The boundaries of the core region are defined:

a) Core Upper Interface equals 6 and defines the axial level whose upper bound is the top of the core,

b) Core Lower Interface equals 2 and defines the axial level whose upper bound is the core inlet.

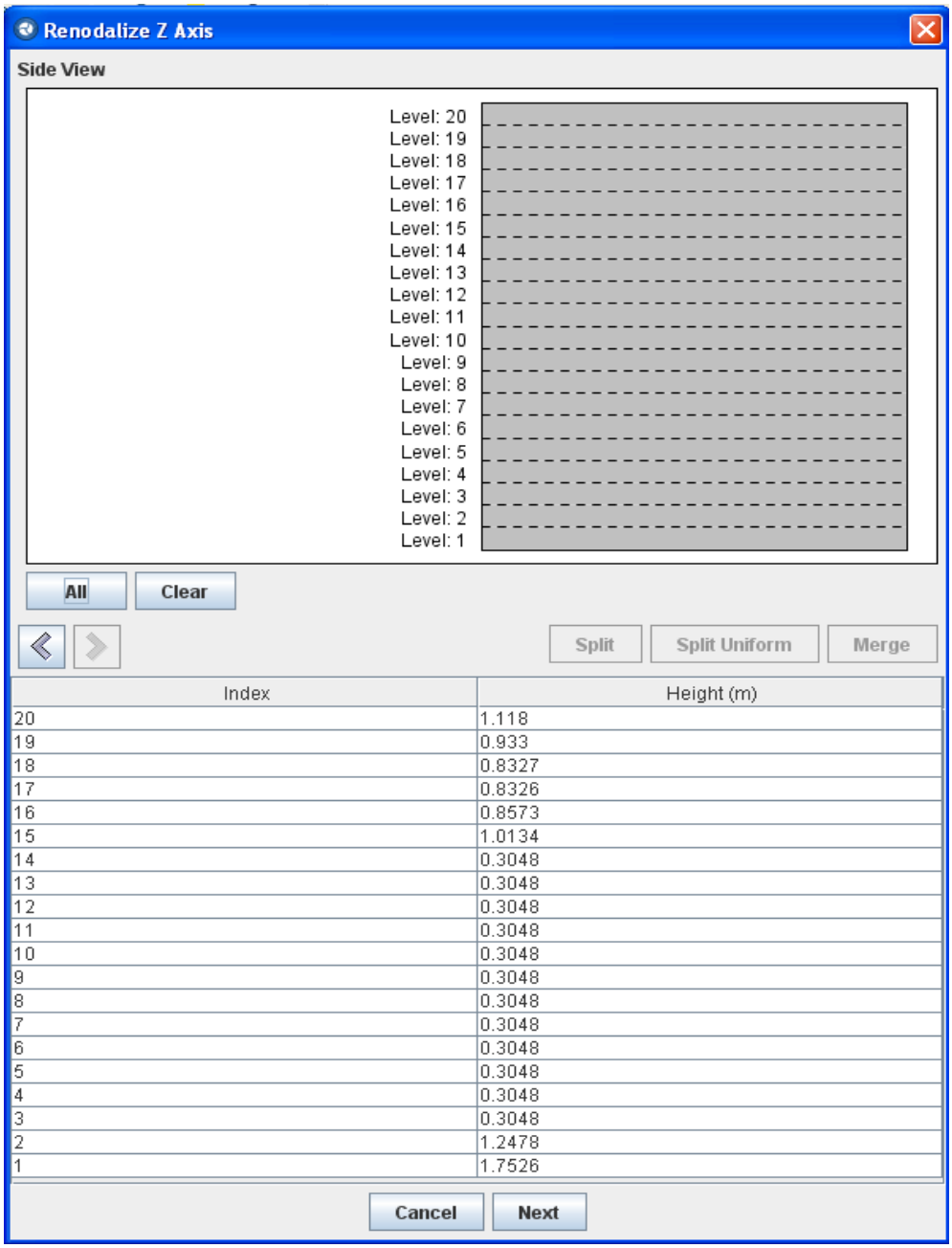
Thus the core region includes Axial Levels 3 through 6.

5. In the Properties Window scroll up to the top and expand the Geometry and Connections input. The lengths of the axial levels are given. The total core length is 3.6575 m (12 ft). The length of each axial level in the core region is 0.9144 m (3 ft). To be consistent with a typical core axial nodalization of 0.3048 m (1 ft) axial lengths, each of the four axial core levels will need to be sub-divided into three axial levels of 0.9144/3 = 0.3048 m (1 ft).

6. Close the 3D vessel window.

7. In the View Window, right-click on the VESSEL component and select "Renodalize Axial Levels.

8. In the popup window, click on the third axial level (Index) in the graphics display. Note that Level 3 in the lower part of the window is highlighted. Click on the "Split Uniform" button and split the level into three cells. Do the same thing for the other long length core levels. When finished, there should be 12- 0.3048 m (1 ft) axial cells in the core region as illustrated below.



9. Click on the "Next" button at the bottom of the window. A display is given that shows what changes were made to the axial levels. Click the OK button at the bottom of the window.

10. A Renodalization Report window will appear that summarizes the changes and checks the model for any loop closure issue that may exist from the renodalization. Click the Close button at the bottom of the window to complete the vessel modifications.

11. Save the model as PWR-SS-Ex3-Step2A.med.

✓ The VESSEL component has been modified to include a sufficient number of axial cells in the core region to provide a moderately-detailed representation of the fluid dynamics during a transient simulation. This completes Step 2. Continue to the next step.

STEP 3. FUEL ROD HEAT STRUCTURE RENODALIZATION

This step gives some experience with the fuel rod heat structures and provides some guidelines in modeling fuel rods. When modeling fuel rods it is a general practice to have an axial heat structure cell for every hydrodynamic cell. For example, if the active core region is modeled with 12 axial levels, then the fuel rod heat stucture will contain 12 axial cells so there is a one-to-one connection. In the initial model for this exercise there were four axial levels in the core region of the VESSEL component and there were four corresponding axial cells in the fuel rod heat structures (HTSTR 41, 43 and 45). In Step 2, the number of VESSEL core region levels was increased to 12, a level for every 0.3048 m (1ft). Since the fuel rod heat structures are tied to the VESSEL component, the number of axial heat structure cells also increased from 4 to 12 automatically. The next step will look at the radial nodalization of the fuel rods.

Step 3.A) Radial Nodalization

Using an acceptable fuel rod radial nodalization is important for predicting the correct initial fuel stored energy for transient simulations. If the radial nodalization is too course, then too much energy may be released unrealistically. Too many radial nodes may cost in computational time.

Past experience has shown that a typical radial nodalization of two equal-spaced intervals in the cladding, one interval in the gas gap and four equal-spaced intervals in the fuel pin yield reasonable results.

1. In the Navigator Window locate and expand the Thermal input section and then the Heat Structures section. Locate and click on Heat Structure 41. This heat structure is one of the three fuel rod heat structures.
2. In the Properties Window, locate and expand the Radial Geometry box. Note there are 3 radial intervals; one interval for the cladding, one interval for the gas gap and one interval for the fuel pin. Using the guidance presented above for radial nodalization, modify the fuel rod heat structure to be consistent a typical fuel rod radial nodalization.
3. Make the same radial modifications to Heat Structures 43 and 45.

Step 3.B) Adding a Supplemental Hot Rod

This step will add a hot rod calculation to the model.

Typically, the core fuel rods are modeled on an average basis. However, a hot rod can be included as a supplemental rod contained within a fuel rod heat structure. The supplemental rod does not affect the thermal-hydraulic behavior of the modeled

system. An input for the hot rod is the power peaking factor. This peaking factor is the ratio of the power of the hot rod to the power of the average rod. This peaking factor is a function of the peak axial power density, the zonal radial average power density and the FQ limit. The FQ limit is a maximum limit with respect to the linear power density at the highest power spot. A typical value for the FQ limit is 2.6. The FQ is the product of the radial power density, the peak axial power density and the hot rod-to-average rod power ratio.

For example, if a core has three radial power zones with average radial power densities (in the outer, middle and inner zones) of 0.8, 1.0 and 1.2, a peak axial power density of 1.6 in each zone and a FQ limit of 2.6, then the ratio of hot rod-to-average rod powers needed for TRACE input are calculated as follows.

Outer zone: 2.6/(1.6*0.8)

Middle zone: 2.6/(1.6*1.0)

Inner zone: 2.6/(1.6*1.2)

The radial power profile for the PWR model used in this exercise is assumed to be flat. The axial power peaking factor is assumed to be 1.6. The FQ limit is assumed to be 2.6. Calculate the ratio of the power in the hot rod to the power of the average rod and use this value in the supplemental rod input.

1. The hot rod peaking factor is (calculated value):

2. Locate and click on Heat Structure 41 in the Navigator Window.

3. In the properties window locate the Supplemental Rods box and expand  the box.

4. In the popup window click on the Add button to add a supplemental hot rod.

5. In the lower part of the popup window input the calculated power peaking factor in the appropriate box.

6. Click on the OK button at the bottom of the window.

7. Repeat the above steps for Heat Structures 43 and 45.
8. Save the Model as PWR-SS-Ex3-Step3A

This completes the fuel rod renodalization step. Go to the next step.

STEP 4. POWER COMPONENT MODIFICATIONS

Power to the fuel rod heat structures is provided by the POWER component. The POWER component contains the following information:

- Identifies the HTSTR components into which power is deposited
- Identifies the power distribution among the core heat structures and within each rod heat structure.
- The total core power

Options for simulating the core power are:

- Power can be specified as a constant.
- Power can be obtained from a table that is a function of time.
- Power can be specified from the output of a control system that includes influences of parameter changes elsewhere in the model
- Power can be specified from the TRACE point reactor kinetics model
- Power can be obtained from a coupled PARCS three-dimensional reactor kinetics model

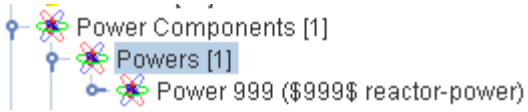
The POWER component can specify a heat structure power shape that changes as a function of time.

The PWR model uses the TRACE point Kinetics model. The power option type is set to 4 which is point-reactor kinetics with an initial constant programed reactivity (REACT) and trip-initiated table lookup of programmed reactivity. This exercise demonstrates the use of the point kinetics model but a detailed description of it is beyond the scope of this workshop.

Step 4.A) Total Core Power

The total core power for this PWR model is 2346 MW. Verify the total power used in the model is at the correct value.

1. Locate the POWER component in the Navigation window of the Model Editor. Open the power component input until "Power 999" is shown:



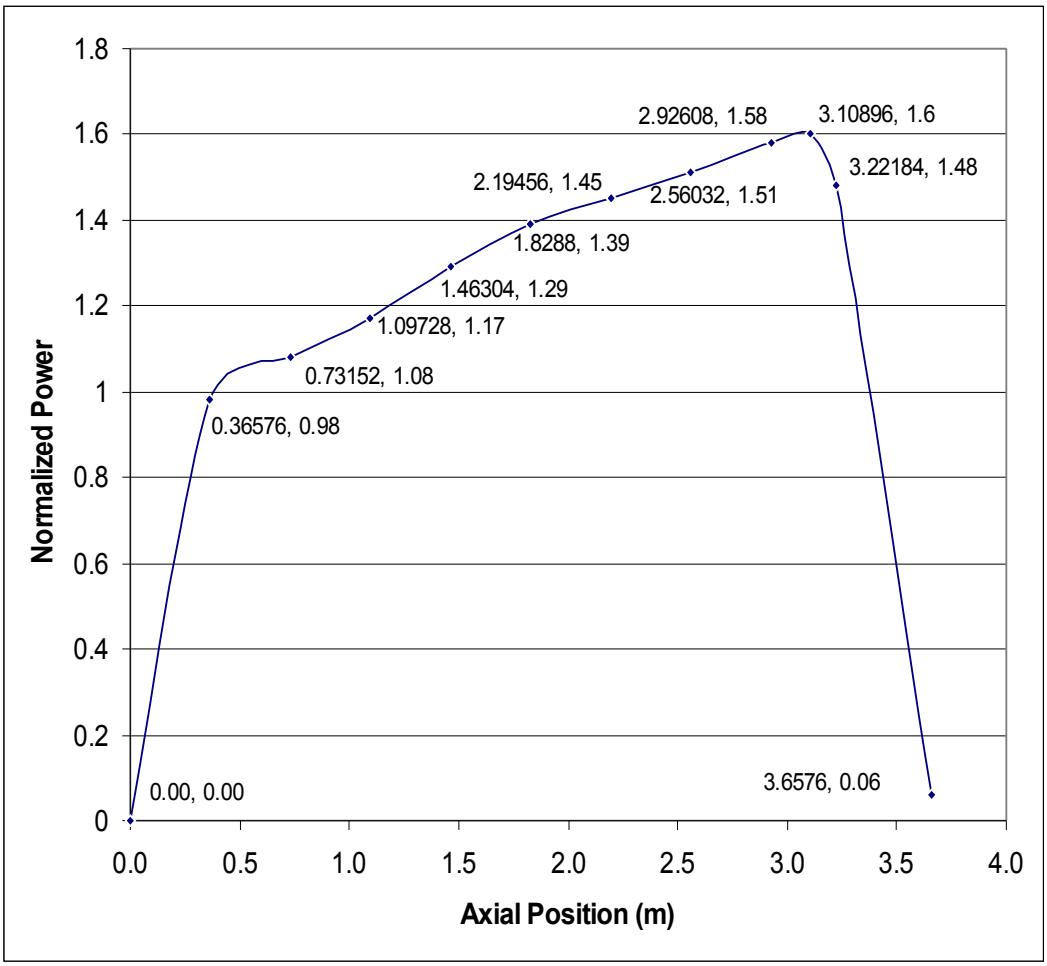
2. Click on "Power 999"
3. The input for Power 999 is entered in the Properties Window. In the Properties Window locate the Initial Power box and verify the input for the initial power is at 2346 MW. Make the necessary changes if needed.

Step 4.B) Input a top-skewed axial power profile

Axial power profiles are cosine shaped and can vary from bottom-skewed to top-skewed depending on the life of the fuel and the control rod positioning during normal operation. For PWR small-break loss-of-coolant-accidents (SBLOCA), the limiting axial power profile (i.e. resulting in the highest peak clad temperature) is typically top-peaked. The last workshop exercise is to perform a SBLOCA simulation.

The axial power profile implemented in the PWR model is bottom-skewed, meaning there is more power deposited in the fuel rods at a lower elevation. The effort here is to

implement a top-skewed axial power profile (more power deposited in the fuel rods at a higher elevation). The following diagram shows a top-skewed power profile



Using the xy data pairs given in the above diagram implement a top-skewed power profile in the PWR model.

1. In the Properties Window locate the Power Shape ▸ Power Shape box and expand it by clicking on the arrow at the left-hand side of the box. Note the following for this POWER component input for the power shape:

a) The power shape uses a 1-D axial table (default). Another option is inputting a 2-D axial-r or axial-x table.

b) The axial integration option uses a histogram with step changes midway between

the axial locations where the axial-power shape's relative power densities are defined. Other options are histogram with step changes at the axial locations where the axial-power shape's relative power densities are defined and trapezoidal integration. **For this PWR plant model exercise use the trapezoidal integration.**

2. Locate the "Axial Shape Locations" box and expand the view. This popup window allows the input of the axial locations along the rod for each of the power densities for an axial power profile. Using the axial power profile shown in the figure above, enter the axial location for each of the defined normalized power densities. When the axial locations are input, click on the OK button.

> The last input for the axial location is the length of the fuel rod and must be the same length as the heat structure designated to be powered by the POWER component. For instance, if the heat structure defining a fuel rod is 3.6576 m (12 ft) long, then the last entry for the axial location must be 3.6576 m. TRACE will check to make sure the two lengths match exactly. If not an input error will result.

3. Locate the "Power Shape Table" box and expand the view. In this popup window, several inputs are entered, including a time varying power shape. For purposes of this exercise, only one power shape is used. For Power Shape #1, note the axial locations that were entered in Part 2 above. Using the axial power profile shown in the figure above, enter the corresponding normalized power density for each axial location. When the normalized power densities are entered, click the OK button.

STEP 4.C) INPUT A FLAT FUEL PIN RADIAL POWER PROFILE

In the model it is assumed the power density within the fuel pin is flat. In Step 3 above, the fuel rod heat structure was radially divided into seven intervals (8 radial interval points). The fuel pin radial power density is input in the Radial Power Shape input in the Properties Window.

1. Locate and expand the Radial Power Shape box.

2. In the popup window, the 8 radial interval points are shown on the left-hand side and the radial power densities are input on the right-hand side. The first five points (0.0 through 4.5275e-3) are the fuel pin. The power in the fuel pin is assumed to be evenly distributed radially. For these five points, input the value 1.0 for a flat distribution.

3. For the last three radial points (the gas gap and the cladding) input 0.0 since there is no power deposited in these sectors. Click the OK button.

STEP 4.D) INPUT A FLAT CORE-WIDE RADIAL POWER PROFILE

The POWER component identifies what heat structures (HTSTR) are to be powered. In the PWR model, the core fuel rods are modeled with heat structures 41, 43 and 45. These heat structures reside in Ring 1 of the VESSEL component. HTSTRs 41, 43 and 45 are connected to Azimuthal Sectors 1, 2 and 3 of the VESSEL component, respectively.

1. In the Properties Window, locate the "Powered Components" box and expand  the view.

The Powered Components popup window is divided into two sections, unpowered components and powered components. All heat structures are listed under the unpowered components. Those heat structures that are to be powered by this POWER component are moved to the Powered Component section.

2. Verify the fuel rod heat structures that are powered with this POWER component are listed in the Powered Component section.

3. Associated with each Powered Component is a power fraction. This input is used to setup the core-wide radial power profile. Since there is only one ring that models the core region in this PWR model, by definition the core-wide radial power profile is flat. Note the power fractions for each of the three powered heat structures. Make the necessary modifications to obtain a flat radial profile. Click the Close button.

The radial power fractions account for the radial power distribution in the core. The power fractions when normalized sum to 1.0. For example, if the radial power distribution is the same core-wide then inputting a 1 for each powered HTSTR would yield a flat profile.

4. Save the model as PWR-SS-Ex3-Step4A.med.

This step is now completed. The input file has been modified to model the desired power, axial power profile and radial power profile.

STEP 5. Obtain the Desired Fuel Center Line Temperature

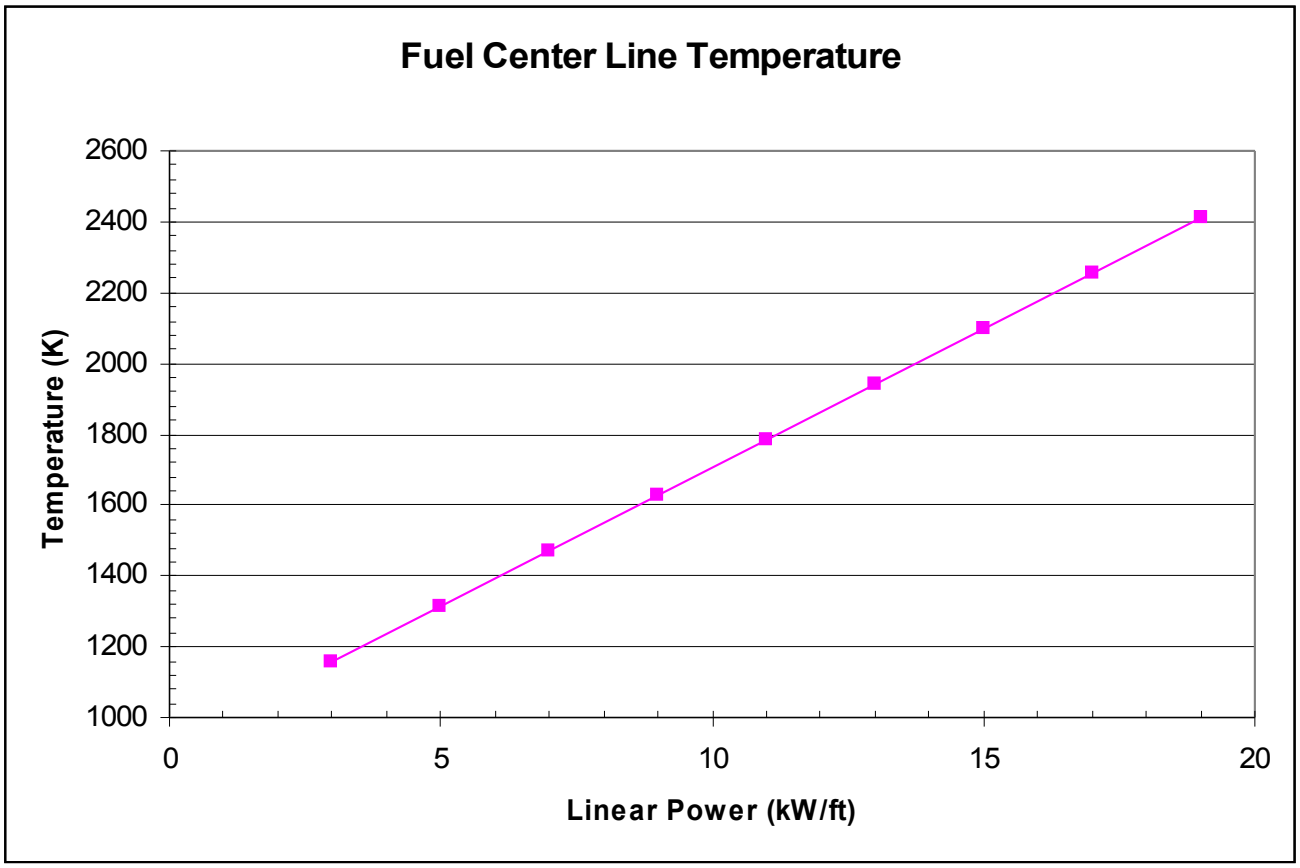
This last step is centered around obtaining a desired fuel rod center line temperature. Getting the right center line temperature is important in transient simulations since it is directly related to the fuel stored energy. Fuel rod center line temperatures can be obtained from refueling analysis.

The fuel rod center line temperature is a function of the linear heating rate. A center line temperature as a function of the linear heating rate is shown in the figure below. Given the total core power, the number of fuel rods and the heated fuel rod length, an average linear heating rate can be calculated by dividing the core power by the number of rods and the heated rod length. The center line temperature is affected by the gas gap conductance, an input into the fuel rod heat structure component.

1. Calculate the linear heating rate (kW/ft) given the steady-state power, the total number of fuel rods and the heated rod length.

- Core Power: 2346 MW – this value is the "Initial Power" input to the POWER component.

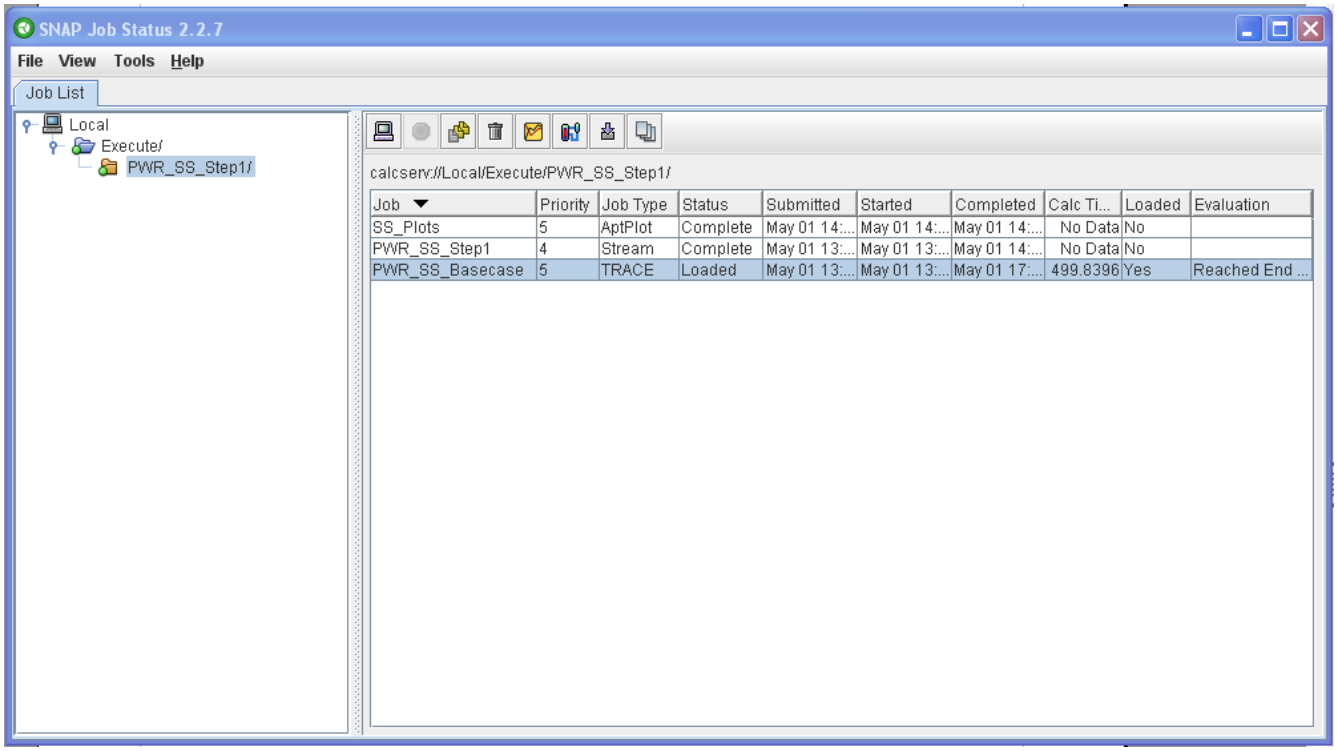
- Total Number of Fuel Rods: 10676 rods per modeled fuel rod – This value is the "Surface Multiplier" input for each of the modeled fuel rods (HTSTRs 41, 43 and 45).
- Heated Rod Length: 12 ft (3.6576 m) – This is the sum of the axial cell lengths describing the fuel rod heat structure. The input for the heat structure cell lengths is located in the "Axial Nodes/Surface BCs" expanded box in the Axial Cell column.
- The linear heating rate is: _____



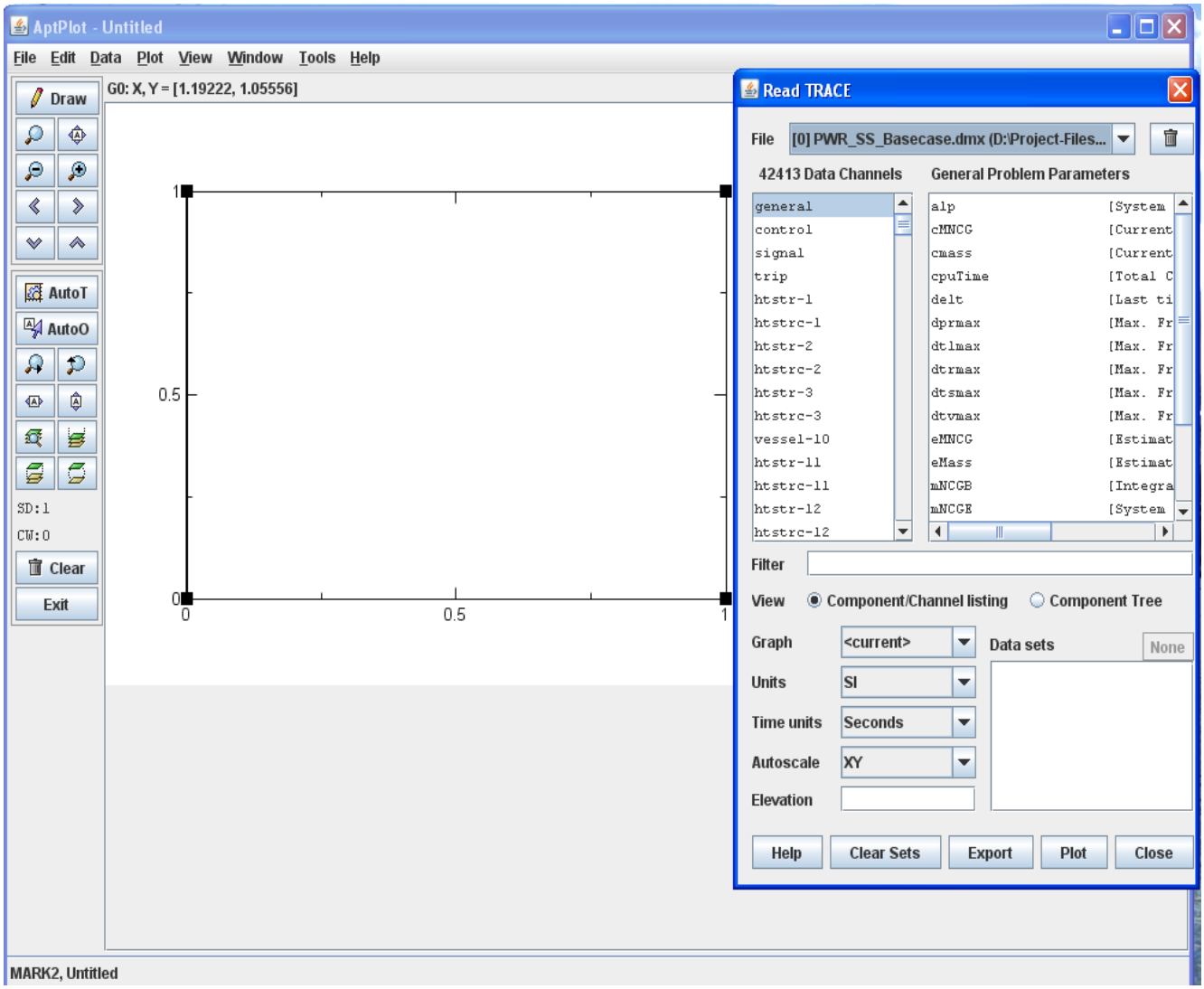
2. Using the calculated linear heating rate determine the center line temperature from the figure. We see that with the calculated linear heating rate, the fuel rod center line temperature should be about 1400 K. For steady-state, the fuel rod center line temperature should be around this value at the peak axial power density location.

3. Compare the center line temperature to the calculated center line temperature from the initial calculation made at the beginning of this exercise. This comparison will be used as a basis for the following steps of the exercise. AptPlot will be utilized for this segment.

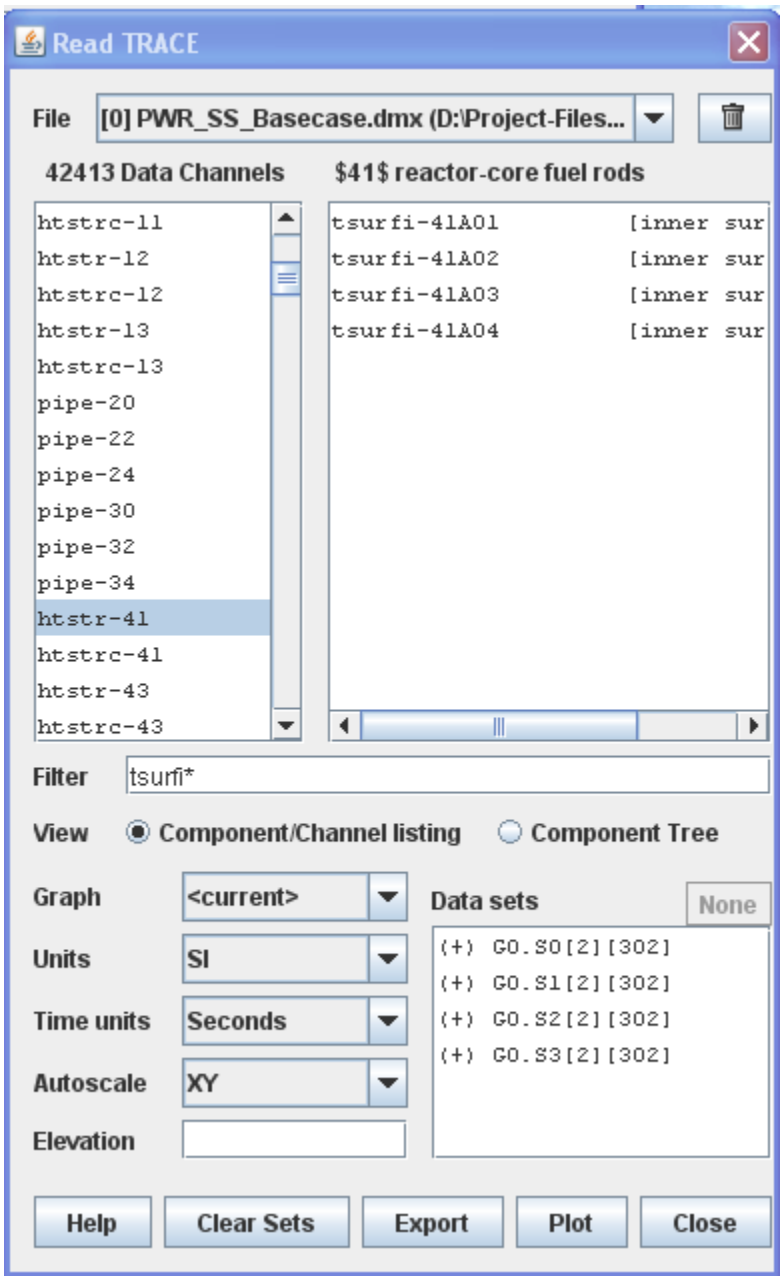
A) Using the SNAP Job Status Window (this window should already be available from the initial calculation that was made at the beginning of this exercise), click on the Job List tab located in the upper left-hand side of the window. Open the Local folder and then the Execute folder. Click on the PWR_SS_Step1 folder. In the Job Status View Window click on the "PWR_SS_Basecase" row to highlight it.



B) In the Toolbar click on the AptPlot ICON to initiate the plotting routine. The AptPlot plotting window will open up as well as the "Read TRACE" plot variable window.



C) In the left-hand pane of the Read TRACE window locate “htstr-41” (one of the three modeled fuel rods) and in the filter box type in tsurfi* and press the enter key. All of the tsurfi plot variables will appear in the right-hand pane. tsurfi is the heat structure inner surface temperature or in this case the modeled fuel rod center line temperatures at the four axial locations. Because of symmetry, the results for HTSTR 43 and HTSTR 45 are expected to be similar to those of HTSTR 41.



D) Select all of the tsurfi plot variables in the right-hand pane and click on the Plot button at the bottom of the Read TRACE window.

E) Make note of the maximum center line temperature (~ 874.7 K) and its axial location.

F) Return to the Model Editor and in the Navigator Window locate Heat Structure 41 and in the Properties Window locate and note the Gas Gap HTC (17,000.0

W/m^2/K).

Compared to the desired center line temperature we see that initial calculated center line temperature is much lower than we want. In the process of this exercise several changes have been made to this model that may affect the center line temperature. We will rerun the the calculation and examine the resultant center line temperature.

4. Rerun the Step 4 calculation using similar steps taken as outlined in STEP 1 above. In the Properties Window, give the submitted job a name of PWR_SS_Step4A to keep this calculation separate from the initial run.

5. When the calculation is finished plot the center line temperatures as outlined above. Note the maximum center line temperature. How did the model modifications affect the center line temperature and its location. Is the new calculated temperature closer to the desired temperature.

Discussion: In the course of the model modifications the VESSEL core region was renodalized into finer axial levels, a top-skewed axial power profile was included in the POWER component as well as a flat core-wide radial power profile, the radial renodalization of the fuel rod heat structures included more detail and a supplemental hot rod was included in fuel rod heat structure. All of these changes are expected to affect the transient results. Changing the axial power profile from bottom-skewed to top-skewed changed the location of the maximum center line temperature. However, changing the total core power was the driver of the higher center line temperature (1320.6 K). Although much closer, the calculated center line temperature is not yet at the 1400 K desired value. One thing that is generally adjusted to bring the center line temperature in line with a desired temperature is the fuel rod gas gap conductance. The larger the gap conductance, the lower the center line temperature. The next steps are used to iterate on the gas gap conductance in order to get the correct fuel rod

center line temperature.

6. For each of the three fuel rod heat structures (HTSTRs 41, 43 and 45):

- Locate and change the Gas Gap HTC from 17,000 W/m^2/K to 8,000 W/m^2/K.
- Using the procedures outlined above for submitting a job, make a short 200 second calculation.
- Note and record the center line temperature in the table below. Use the procedure outlined above for using AptPlot.
- Repeat the above for Gas Gap HTCs of 10,000 and 9,000.

Gas Gap HTC (W/m^2/K)	center line Temperature (K)
17000	1320.6
8000	
10000	
9000	

From the iterative process we see that a gas gap conductance of 9000 W/m2/K produces the desired center line fuel temperature at the peak power density.

Points to Consider

- This exercise gave experience with the VESSEL, POWER and HTSTR components. The model was modified to include a more detailed nodalization of

the VESSEL component based on user guidelines. This was done to provide a more accurate model for making transient calculations. When performing transient analysis, consider the transients to be performed and use good modeling practices to achieve reasonable results.

- A top-skewed axial power profile was added to the POWER component. This addition affected the location of the maximum fuel rod center line temperature. For transients of interest, consider the axial power profile and its effects on fuel temperature during a calculation.
- The fuel rod heat structure radial nodalization was modified to provide more detail in the cladding and fuel pin. A fuel rod that is coarsely noded radially can lead to a change in the fuel temperature that is not realistic.
- The gas gap conductance was changed to get the correct fuel center line temperature. Having the correct center line temperature defines the stored energy in the fuel rod that is important during transient calculations.