Modeling Jet Pump Form Losses and Advanced Fuels

OBJECTIVE

The BWR PIRT identifies Jet Pump losses as having a High impact on Peak Cladding Temperature (PCT) for LBLOCA events based on the influence during the blowdown phase. The PIRT also identifies power distribution as having a high impact on the PCT. The power distribution in a bundle may be impacted by the advanced BWR fuels model, so including the advanced BWR fuel model will be included in the model.

The objective of this exercise is to add form loss factors to the jet pumps and add advanced fuel characteristics to the example BWR model.

PIRT INFORMATION

For LBLOCA events, the BWR PIRT[1] ranked jet pump losses and core power distribution as follows:

Phase	Blowdown			Refill/Reflood		
Phenomena ↓ BWR Type →	**2**	**3,4**	**5,6**	**2**	**3,4**	**5,6**
Core – Power Distribution: radial	M	M	M	H	H	H
Core – Power Distribution: axial	M	M	M	M	M	M
Jet Pumps – Flow: forward	0	H	H	0	L	L
Jet Pumps – Flow: reverse	0	H	H	0	L	L

Advanced fuels can affect both the axial and radial power distributions. Radial distributions are affected by the inclusion of water rods, and by the fact that power that would have been emitted by the upper part of the part length rods is distributed among the rods. Some of the differences are exhibited in the fuel bundle profiles that must be provided by measured bundle data.

1 “BWR PIRT and Assessment Matrices for BWR LOCA and NON-LOCA Events” by M. Straka and L. W. Ward, SCIE-NRC-393-99, Contract NRC-04-96-060 Task 002, BWR_PIRT.pdf (on workshop CD)

Setup

You can either continue with the model from the previous exercise, or open the SNAP model named 'PBTT_SS4.med' in the folder 'Day4\Afternoon\BWR\' under the workshop main folder.

Configuring Jet Pump Form Losses

Form losses in general help define the flow rates through the system, and the path that coolant will take. During blowdown the jet pump form losses affect the flow of coolant to the break.

The diffuser and nozzle losses are calculated using Idel'Chik irreversible loss factors for contraction and expansion through a smooth change of area. The equations are:

$$\begin{aligned} &\text{Expansion Losses: } K_E = C_E(\tan\alpha)^{1.5}(1-A^*) \\ &\text{Contraction Losses: } K_C = C_C\sin\alpha(1-A^*) \end{aligned} \qquad (1)$$

$K_E \& K_C$ = Expansion & Contraction K Loss Factors
$C_E \& C_C$ = Expansion & Contraction Constants (TRACE Inputs)
α = Expansion or Contraction Angle
A^* = Area Ratio of Outlet (A_0) to Inlet (A_i) Flow Areas where $A_0 \le A_i$ (i.e. $A^* \le 1$)

If HD_{j+1} and HD_j are the hydraulic diameters of the inlet and outlet of the nozzle or diffuser and DX_j is the cell length. The expansion/contraction angle can be calculated using the following formula:

$$\alpha = \arctan\left(\frac{|HD_{j+1} - HD_j|}{2\text{DX}_j}\right) \qquad (2)$$

To configure the form losses for jet pumps 65 and 69, select them one at a time and do the following:

1. In the Properties Window locate the forward and reverse form loss input boxes for the Diffuser and Nozzle.

2. To simplify the process for this exercise, TRACE has default values that will be used for input. These default values are obtained by clicking on the help icon for each of the form losses. Note the default values and enter them into the appropriate input box. For example, the Forward Diffuser Formloss help string indicates the following (Default = 5.5) :

   - Forward Diffuser Formloss (Diffuser C_E value)
   - Reverse Diffuser Formloss (Diffuser C_C value)
   - Forward Nozzle Formloss (Nozzle C_E value)
   - Reverse Nozzle Formloss (Nozzle C_C value)
   - Suction Formloss
   - Discharge Formloss

> ! Note that formloss can be specified independently from K losses at edges in the jet pump (indicated in the 'Friction' dialog). K losses for internal edges that are specified in the 'Friction' dialog will be ignored. K losses can be specified for boundary edges that connect to other components however.

3. Add K loss factors at the jet pump boundaries by doing the following:

   a) Expand the Friction properties dialog in the Properties Window.

   b) In the 'Friction' dialog, set the following K losses at the three boundary edges. Remember that these K losses are in addition to the suction and discharge form K losses. Note that ALL the internal edge K factors in the jet pump component are ignored.

   - Jet pump suction: apply a forward flow loss of 0.1 and a reverse flow loss of 0.2.
   - Jet pump discharge: apply a forward flow loss of 0.2 and a reverse flow loss of 0.1.

- Jet pump nozzle inlet: apply a forward loss of 0.2 and a reverse flow loss of 0.1.

The PIRT indicated that Jet Pump losses can have a high impact on the PCT based on the influence during the blowdown phase. Since we have no specific data, the Jet Pumps were configured to use the default loss values. The model should now capture Jet Pump loss phenomenon.

Configure Advanced Fuels – Water Rods

Advanced fuel designs include partial length rods and water rods which can help shape the power profile to increase the stability in the core and improve cold shutdown margins and the fuel efficiency. Water rods and fuel rods may be partial length rods.

For purposes of the workshop, advanced fuel options will only be configured on the hot bundle (CHAN 1). First we will add a water rod group with **2 water rods**. In order to define the water rods we will need to choose a diameter, the water initial temperature and some other parameters. Water rods sometimes take up the space of more than one rod, but for this exercise, we assume that the water rod is the same diameter as the fuel rods. We will use an initial temperature consistent with fluid temperatures in the CHAN fluid cells. To configure the water rods do the following:

1. Select CHAN 1.
2. E xpand the initial conditions box in the Properties Window. Note the initial temperatures in the rod range from 551 K to 558 K. Also note that most of the temperatures are about **558** K. We will use this as the initial temperature for fluid in the water rod.
3. Open the ▾ Fuel Rods section and locate the outer radius of the modeled fuel rod. Note the outer radius is **6.2611E-3** m. Twice this value will be used for the

diameter of the water rods.

4. Note that SNAP uses a toggle switch to activate the advanced BWR fuel model. This toggle switch is located in the Properties Window for each modeled CHAN. Locate the toggle switch and assure it is set to true. Turning on the advanced BWR fuel model activates additional input required for the CHAN component input. Part of this additional input is for implementation of water rods.

5. Locate and Expand the Water Rods box in the Properties Window. In the 'Edit Water Rods' dialog, add a water rod group.

> Note the water rods group that is created is called 'Water Rod: 1'. Recall that in a previous exercise, three 'Non-Average Rod' groups were added (these groups can be seen by clicking E on 'Non-Average Rods' in the Fuel Rods section of the CHAN 1 properties). In the 'Rod Locations' dialog (which we will open later), the 'Water Rod: 1' group is identified as group 5. Group 1 is the default average rod group. Groups 2, 3, and 4 are the 'Non-Average Rod' groups created in a previous exercise. In the 'Rod Locations' dialog, the water rod groups are numbered after the 'Non-Average Rod' groups. Since we have 1 water rod group, it is group 5.

6. For this water rod in this exercise a cylindrical geometry will be assumed.

7. In the dialog box, locate and Expand the initial temperature box. In the "Edit Initial Temperature" dialog, click on one of the unknown temperature values and press 'ctrl-a' on the keyboard to select all the cells. Enter the temperature of the fluid inside the water rod (see Step 2 above). Press the enter key to set the fluid temperature to all cells in the water rod.

8. The remaining input data for the water rods includes the water rod outer diameter, the wall thickness the inlet and outlet forward and reverse flow losses and the surface multiplier. The basis for the water rod diameter is given in Step 3 and a water rod wall thickness of 7.0e-4 m is assumed. A discussion of the flow losses is given in the information blocks below. Recall that two water rods are being

modeled. From this information, complete the input data in the 'Edit Water Rods' dialog.

While K losses are not usually provided for the water rods, a target flow often is available. K losses are typically adjusted in order to achieve the target flow through the water rods. In this case, we will choose K loss factors 10 times the anticipated losses.

The inlet forward direction is oriented INTO the pipe, which would be a contraction for K loss purposes, so the anticipated K loss value would be about 0.4. A value of 4 will be used to simulate intentional resistance to tune flow. The inlet reverse direction is oriented OUT of the pipe, which would be an expansion so the anticipated K loss value would be 1.0. 10 was used to to simulate water rod tuning. Similarly, the outlet forward direction is oriented OUT of the pipe (an expansion), while the outlet reverse is oriented INTO the pipe (a contraction).

Note that the surface multiplier is set to 2. This indicates that the space of two rods will be taken up by water rods, and NOT that there are necessarily two water rods in the bundle. If each of the 2 water rod took up the space of 4 rods in the bundle, the surface multiplier would be 8. However we are assuming the water rods only take the space of 1 rod in this case, so the surface multiplier is 2.

Now that the water rod properties have been configured, we need to specify the location of the water rods. In this case, SNAP needs to know the X and Y location of the center of the rod. The origin of the coordinate system used to locate the center of the rods is at the upper left hand corner of the inside of the channel box as shown in Figure 1. The values of X and Y are specified in meters and the X coordinate for the center of the water rod will be a positive value, while the Y coordinate will be a negative value.

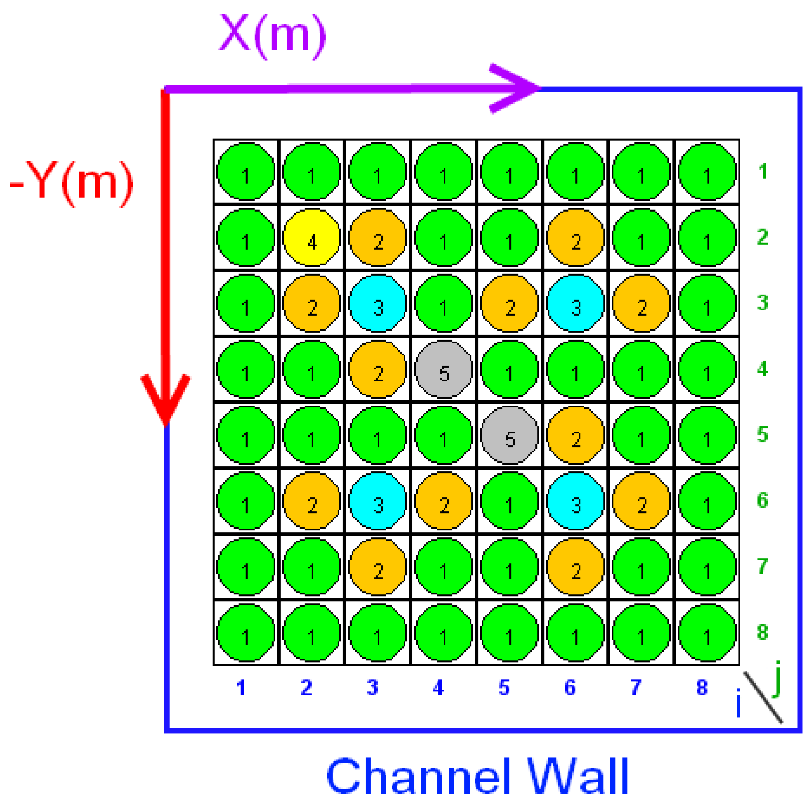

Figure 1: Rod Groups with Water Rods Shown as gray circles with a 5 in the center.

Along with the location of the center of the rod, the specific rod slots that are occupied by the water rods need to be identified. Rod slots are identified by an (i,j) integer pair, where i indicates the location along x, and j the location along y. We will be assigning location (4,4) and (5,5) as water rods as shown in Figure 1 above.

You can see the rod diagram by Expanding 'Rod Locations' in the ▼ General section of the Properties Window, but the gray circles will not be visible yet because the rod slot haven't been identified for the water rods. To set the location, we first need to calculate the x and y position of the rods. This can be calculated if we have the width of the Channel box, the radius of the rods, and the rod pitch. The table below shows where to find the relevant information in the Properties Window, and shows the calculations to get the water rod locations:

Property (var. name)	Value (m)	Property Location/Calculation
Channel Perimiter (L)	0.53624	▼ Canister Wall Width

Property (var. name)	Value (m)	Property Location/Calculation
Channel Width (w)	0.1341	L/4
Rod Radius (r)	6.2611E-3	▸ Fuel Rods Rod Thickness **(Note this is NOT rod WALL Thickness)**
Rod Diameter (D)	1.2522E-2	r*2
Rod Pitch/Diam. (R)	1.2982	▸ Fuel Rods Rod Ratio
Rod Pitch (P)	1.6256E-2	R*D
Rods Per Bundle (n)	8	▾ General Rods per Row
Bundle Width (w_b)	1.2632E-1	P*(n-1)+D
Wall to Bundle (d)	3.8718E-3	$(w-w_b)/2$

If the rods are numbered from 1 to 8 as in the diagram, and *i* indicates the number of the rod of interest, the distance from the wall to the rod center line can now be calculated by the formula:

$$D_i = d + r + P(i-1)$$

Based on this formula, calculated the center locations for the water rods at (4,4) and (5,5)

D_4 =

D_5 =

To set the position of the water rods:

1. xpand 'Water Rods' box in the Properties Window.

2. Select 'Water Rod: 1' and xpand 'Locations' in the water rod properties.

3. In the 'Edit Water Rod Locations' dialog, add two water rods (upper add button).

> When entering water rod location, the ' Center Along X' and 'Center Along Y' values indicate the center of the water rod. X **must** be positive or 0 and **Y must be negative** or 0. SNAP doesn't warn you if there is an error. If both values are set to zero, the manual indicates that TRACE will calculate the X and Y value for you based on the rod slots that this water rod occupies (defined in step 5 below). However, tests using (0,0) causes the simulation to fail.

4. Select 'Water Rod 1' and set the 'Center Along X' and 'Center Along Y' values to those calculated for D_4 above.

> We now have set the location of the rod center in the bundle, but we need to identify which rods in the bundle are part of this water rod group. A single water rod may take up the space of multiple fuel rods. We click the bottom **Add** button once for each fuel rod slot that the individual water rod occupies. In our case the water rod only takes up one slot.

5. Click the bottom **Add** button once and set both the 'X Axis' and the 'Y Axis' values to **4**.

> The 'X Axis' and 'Y Axis' values are the rod identification per Figure 1. For example (1,1) is the upper left hand corner rod and (8,8) is the lower right hand corner rod. This water rod is located at (4,4). If the water rod was large enough to take up the space of multiple fuel rods, we would need to add the number of fuel rod slot occupied and set the grid location of each water rod in the 'X Axis' and 'Y Axis' values in the list.

6. Select 'Water Rod 2' and set the 'Center Along X' and 'Center Along Y' values for D_5 calculated above. Add in the rod identification location for this water rod (5,5) as was done in Step 5 above.

7. Expand 'Rod Locations' in the General tab and verify that the location of the water rods matches Figure 1.

Since we have replaced average rods with water rods, we need to reduce the physical rod multiplier value for the average rods indicated in the CHAN properties by 2 (since each of the 2 water rods take the place of one rod in the bundle).

8. Update the number of physical rods in the average rod group, in the CHAN 1 Properties Window, under the ▸ Fuel Rods tab.

Configure Advanced Fuels – Fuel Rod Lengths

> **!** When partial length rods are added, TRACE checks to make sure that the partial length rod does not extend above the heated portion of the core. If it does, TRACE will indicate an input error and stop the simulation. All Non-Average rods are assumed to be partial length rods when advanced fuel is enabled.

Before we set the partial rod lengths, lets look at what the maximum length of the rod can be. To do this, locate and Expand the "Axial Heatstructure Properties" dialog box in the Properties Window and click on the "Power Peaking Factors" input.in the pop up window. Note the powered cells range between 3 and 26. Therefore, the partial length rods cannot extend above level 26.

> **!** The partial length rods can't extend above level **26** (in this case). This is the last heated level in the CHAN, so partial length rods CANNOT extend above level 26.

The lengths of the partial length rods are set in the "Non-Average Rods" dialog under the Fuel Rods section of the Properties Window. For this exercise, we will assume the "Non-Average Rod: 1" length ends at cell 20, the "Non-Average Rod: 2" length ends at level 22, and the "Non-Average Rod: 3" length ends at level 26.

1. In the CHAN 1 Properties Window, Expand the 'Non-Average Rods' dialog under the ▸ Fuel Rods section. Locate the "Last Axial Cell" input and make the

above changes to the non-average rod lengths.

The advanced fuels have now been configured for the **hot bundle**.

Perform a Steady State Run

Rerun a steady state simulation by doing the following:

1. Save the SNAP model as PBTT-SS4-Ex4.med.
2. Submit the steady state simulation using the Job Stream View tab located at the bottom of the View Window in the Model Editor.
   a) Click on the Execute button and change the name to reflect this exercise (for example BWR-Steady_State-ex4). Note that changing the name of the SNAP job submission will not over-write the calculations made previously.
   b) Click on the lock button in the upper left-hand corner of the View window and then click on the Execute button to submit the job.

From the animation window review the steady state target values again. Did any values change significantly?

Parameter	Target/Expected Value	Actual Value
Turbine Stop Valve Pressure		Old: New:
Steam Dome Pressure		Old: New:
Downcomer Level		Old: New:
Core Mass Flow		Old: New:
Total Jet Pump Mass Flow	None Given	Old: New:

Parameter	Target/Expected Value	Actual Value
Core Inlet Temperature		Old: New:
Feedwater Flow	None Given	Old: New:
Steam Line Flow	None Given	Old: New:

OPTIONAL EXERCISE

For the Jet Pumps, the nozzle and diffuser losses are not direct K losses, but rather coefficients to the expansion and contraction irreversible loss equations shown in (1). The default expansion loss coefficient (C_E) that was used is **5.5** and the default contraction loss coefficient (C_C) used was **0.38**. Calculate what the actual nozzle and diffuser K losses are based on these values using the equation (1) on page 2 and equation (2) on page 2. Some of the equation parameters will have to be taken from the jet pump 'Component Geometry'.

Name	Calculation/Value
Nozzle Expansion Loss	
Nozzle Contraction Loss	
Diffuser Expansion Loss	
Diffuser Contraction Loss	