TRACE Component Introduction – Part 1

Information Systems Laboratories, Inc.

Presented at

Nuclear Regulatory Commission
TRACE/SNAP User Workshop
Idaho Falls, Idaho
September 30 – October 3, 2014

Objective

Provide basic information about the TRACE components used for building system models

This information is needed for developing and applying TRACE models and is presented in three parts

At the end of each part, exercises are included to demonstrate the development and application of TRACE models

Outline of Information Discussed in Part 1

- Global TRACE Modeling Options
- Modeling Features Common to Multiple TRACE Component Types
- PIPE Component
- FILL Component
- HTSTR Component
- Material Properties Data
- Control System Input: Signal Variables and Control Blocks
- Time Step Data Input

Global TRACE Modeling Options

There are a couple of TRACE global modeling options that the user should become familiar with:

- NAMELIST Variables
- Equation-of-State

Global TRACE Modeling Options

NAMELIST Variables:

The input instructions for the TRACE component models contain several conditional data arrays that are either entered or not entered depending upon the presence (or lack thereof) of certain global modeling flags/options, normally provided in Fortran namelist format.

Example: Namelist Variable IKFAC defines whether loss coefficients or K loss factors are to be entered

Namelist variables are entered in a specific format:

- The list of variables must begin with a **$(or &)INOPTS** beginning in column 2.
- The list of variables must end with a **$(or &)END(or end).**

SNAP automatically sets the correct format for the NAMELIST variable input

Global TRACE Modeling Options

NAMELIST Variables (continued):

The use of NAMELIST variables is activated with the **Namelist Option** (INOPT). This flag is located in the Properties Window under the Model Options (Navigator Window) dialog in SNAP. The **Namelist Options** are:

0 = NAMELIST input data is omitted
1 = NAMELIST input data is entered
-1 = NAMELIST input data is obtained from the restart file

SNAP only uses options 1 and -1 with 1 as default

Global TRACE Modeling Options

NAMELIST Variables (continued):

The list of NAMELIST variables is extensive and should be thoroughly reviewed and understood (TRACE User's Manual, Volume 1: Input Specification).

NAMELIST variables have default settings which are typically used for TRACE calculations.

However, there are some commonly used variables that need particular attention. These variables affect the behavior of one-dimensional components, the VESSEL component and the HTSTR component.

Global TRACE Modeling Options

NAMELIST Variables (continued):

One-Dimensional Component NAMELIST Variables:

- FLUIDS – Working fluids available in TRACE
- ICFLOW – Choked flow model option
- IELEV – Option to specify whether gravity terms (GRAV), cell centered elevations (ELEV) or full-cell elevation changes (DELEV) are to be input in the component data
- IGAS – Non-condensible-gas type option
- IKFAC – Defines whether additive loss coefficients (FRIC) or K factors are input
- IOFFTK – Invokes the offtake model
- MWFL – Option for input specifying wall-to-liquid wall-friction multiplicative design factors
- MWFV– Option for input specifying wall-to-gas wall-friction multiplicative design factors
- NCGASSPECIES – Provides capability to have a mixture of non-condensible gases
- NFRC1 – Option to allow separate input for forward and reverse flow loss coefficients
- NOLT1D – Option controlling the use of the 1D level tracking model
- USESJC – Option to allow single junction components and side junctions to 1D components

The use of these NAMELIST variables applies to all one-dimensional components in an input file

Global TRACE Modeling Options

NAMELIST Variables (continued):

Vessel Component NAMELIST Variables:

- FLUIDS – Working fluids available in TRACE
- IELEV – Option to specify whether gravity terms (GRAV), cell centered elevations (ELEV) or full-cell elevation changes (DELEV) are to be input in the component data
- IGAS – Non-condensible gas type option
- IKFAC – Specifies whether additive loss coefficients (FRIC) or K factors are input
- MWFL – Option for input specifying wall-to-liquid wall-friction multiplicative design factors
- MWFV– Option for input specifying wall-to-gas wall-friction multiplicative design factors
- NCGASSPECIES – Provides capability to have a mixture of non-condensible gases
- NFRC3 – Option to allow input of separate forward and reverse loss coefficients for all 3D vessel components
- NOLT3D – Option controlling the use of the level tracking model in the VESSEL component
- NVGRAV – Option for the user-specified orientation of 3D vessel components with respect to the gravity-vector direction
- USEVESSHS – Option that controls use of simplified VESSEL heat structure input

The use of these NAMELIST variables applies to all VESSEL components in an input file

Global TRACE Modeling Options

NAMELIST Variables (continued):

HTSTR Component NAMELIST Variables:

- NRSLV – Axial-direction conduction heat transfer calculation numerics in all HTSTR/CHAN components having IAXCND = 1
- ITHD – Option allowing specification of heated diameters which differ from hydraulic diameters
- nEnclosure – Number of radiation heat transfer enclosures in the model
- MHTLI – Option for specifying wall-to-liquid heat transfer multiplicative design factors for the inner surface of all HTSTR components
- MHTLO – Option for specifying wall-to-liquid heat transfer multiplicative design factors for the outer surface of all HTSTR components
- MHTVI – Option for specifying wall-to-gas heat transfer multiplicative design factors for the inner surface of all HTSTR components
- MHTVO - Option for specifying wall-to-gas heat transfer multiplicative design factors for the outer surface of all HTSTR components

The use of these NAMELIST variables applies to all HTSTR components in an input file

Global TRACE Modeling Options

NAMELIST Variables (continued):

The NAMELIST variables are exercised in the Properties Window when "Model Options" is selected in the SNAP Navigator Window.

The NAMELIST variables are listed under 7 categories in SNAP:

1. General
2. Default Initial Conditions
3. Choke Flow Options
4. Output Options
5. Diagnostics
6. Design Factors
7. RELAP5 Options

Global TRACE Modeling Options

Equation-of-State:

TRACE currently offers the choice of two different equation-of-state models:

1. The legacy built-in curve fit formulation inherited from TRAC-PF1
2. An interpolation scheme based on the 1997 International Association for the Properties of Water and Steam (IAPWS) Industrial Formulation (IF97) standard

Curve fit formulation:

- Currently performs somewhat faster (~ 10-15%) than the IAPWS standard
- The speed advantage comes at the cost of lower accuracy in certain regions of the phase diagram for water
- Significant impact on the predictions of certain reactivity feedback transients
- Detrimentally impacts the robustness of the choked flow model
- Metastable fluid conditions (superheated liquid and subcooled vapor) not handled well

Global TRACE Modeling Options

Equation-of-State (continued):

It is generally recommended to use the IAPWS formulation for most applications.

TRACE defaults to the IAPWS formulation.

To invoke the legacy built-in curve fit formulation requires the application of NAMELIST variable USE_IAPWS_ST.

> ⚠ The IAPWS formulation currently behaves very badly when conditions of the fluid reach far into the metastable region (i.e., liquid becomes highly superheated or vapor becomes highly subcooled). In these situations, the violence of water packing events, even after correction, will have an abnormal tendency to drive the fluid into odd states during iteration. Currently, the best guidance is that any input model which fails with water packing and iteration failure messages in close proximity needs to be tried with the legacy curve-fit EOS formulation

Modeling Features Common to Multiple Component Types

While every 1D component type offers its own unique capabilities, many modeling features/options are common across multiple components. The common modeling features/options include:

1. Component Geometry – Modeling Cell/Edge Geometry in 1D Components.
2. Friction - Modeling Pressure Losses in 1D Components.
3. Special Modeling Options for 1D Components.
4. Pipe Wall – Modeling Pipe Walls.
5. Initial Conditions.

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

The geometry data for 1D thermal-hydraulic components are specified by five arrays:

1. Cell Length (DX)
2. Cell Fluid Volume (VOL)
3. Cell-Edge Fluid Flow Area (FA)
4. Vertical-Orientation Information (GRAV/ELEV/DH)
5. Cell-Edge Flow-Channel Hydraulic Diameter (HD)

DX, VOL, ELEV and DH arrays are identified with "cells" while FA, GRAV and HD arrays are identified with the "cell edge"

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Cell Length Array (DX):

- This array defines the cell lengths.
- Each 1D cell should be made as long as possible while justifying the requirement of an average homogeneous fluid condition over the length of each cell.
- Cell lengths should be shorter where the thermal-hydraulic conditions are expected to vary more per unit length.
- As a general rule-of-thumb, the average cell should have an L/D of between 4 and 5.

1D flow equations are constructed by averaging across the width of the flow-channel. Selection of a cell length less than the hydraulic diameter of the flow channel does not normally make sense. Exceptions may occur when it is necessary to limit numerical diffusion. Trade-offs may be necessary when exercising modeling judgment.

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Cell Volume Array (VOL):

- This array defines the cell volumes.
- The fluid volume input data is an independent data array to account for constant and variable geometry configurations.
- Because the fluid inventory and its spatial distribution throughout the system are important for simulating the behavior of many transients, the fluid volume of each cell should be carefully determined.

TRACE computes a cell-average flow area (VOL/DX) that is used to accurately calculate the cell-average pressure and the define the momentum flux at the cell center. For large changes in VOL/DX, TRACE checks to see if loss coefficients have been input so the irreversible flow losses can be properly calculated. Warning messages are issued if loss coefficients have not been entered for large changes in VOL/DX.

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Cell-Edge Fluid Flow Area Array (FA):

- The flow area is defined at each cell edge.
- Generally, define cell-edge boundaries at locations where the fluid flow area can be easily determined.
- Some flow areas are reduced by the presence of structural materials. In these cases, special attention should be paid to the specification of flow areas and hydraulic diameters.
- Unlike RELAP5, for TRACE note that the combination of DX, VOL and FA does not have to be consistent.

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Cell-Edge Fluid Flow Area Array (FA):

TRACE imposes some limitations on the values that can be specified for this array. Three checks are performed on the flow areas:

1. Junction-to-junction

   Checks to see if the flow area on each side of a junction between components actually match.

2. Volume-to-edge-to-volume

   Checks the ratio of the volume average flow areas on each side of a cell edge. If one flow area is 2X the other flow area, TRACE checks to see if loss coefficients have been supplied or the abrupt area change model has been activated. If not a fatal error is given. The purpose is to force the user to model the irreversible losses correctly.

3. Volume-to-volume

   Checks for large expansions immediately adjacent to a contraction. If an instance is located in which an edge or junction flow area is more than 1.1 times the maximum adjacent volume average flow area a fatal error is given. This type of mesh topology can cause robustness or stability problems for TRACE.

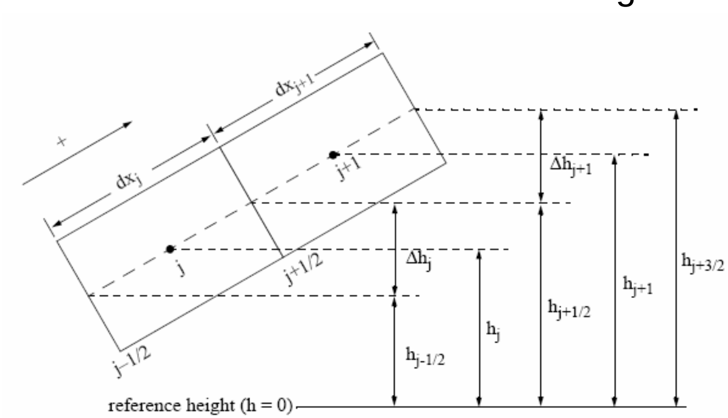
Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Vertical-Orientation Information (GRAV/ELEV/DH):

There are three methods for establishing the elevation and/or vertical orientation of components and cell volumes: GRAV, ELEV or DH. When ELEV or DH is used, TRACE internally converts them to a GRAV. Each method has advantages and disadvantages. Which method chosen is specified using the IELEV NAMELIST option.

1. Gravity Vector (GRAV) – IELEV = 0:
   - Defined as the ratio of the change in elevation to the length of flow path between cells



$$GRAV_{j+\frac{1}{2}} = \frac{h_{j+1} - h_j}{\frac{dx_j}{2} + \frac{dx_{j+1}}{2}}$$

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Vertical-Orientation Information (GRAV/ELEV/DH):

Gravity Vector (GRAV) Rules:

1. Gravity terms are specified at cell edges
2. The numerical sign of a GRAV term is defined by the positive direction of travel across the cell face. Positive direction of travel is from the lowest numbered cell to the highest numbered cell in a component.
3. At a cell-edge interface along the direction of travel, the sign of the GRAV term is positive if the cell center ahead is higher than the cell center behind. Its sign is negative if the cell center ahead is lower than the cell center behind. If the cell centers are at the same elevation a zero is assigned to the GRAV term.
4. At junctions between two components, the magnitude of the GRAV terms specified for each adjoining component must be identical.

Tip – When using SNAP for model development, you may find the use of the GRAV terms to be more convenient. When working with the ELEV array, small perturbations in elevation differences introduced by numerical round-off can cause TRACE to detect situations which an equivalent GRAV term is slightly greater than one and generates numerous warning messages as a result.

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Vertical-Orientation Information (GRAV/ELEV/DH):

2. Cell Centered Elevation (ELEV) - IELEV = 1:
    1. The cell-centered elevation is simply the difference between the center of a cell and a reference elevation.
    2. The reference elevation can be arbitrary.
    3. Conceptually, this is the most straightforward approach and is generally recommended if developing a new model outside of SNAP.
    4. TRACE does not attempt to go through flow loops to check for loop closure (SNAP contains this feature), but it does include warning checks on the ELEV terms to determine if the absolute value of the equivalent GRAV terms (calculated internally by TRACE) are greater than 1.0

For a more detailed discussion of using the ELEV term for elevation changes refer to the TRACE Users Guide, Volume 2

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Vertical-Orientation Information (GRAV/ELEV/DH):

3. Change in Cell Elevation (DH) - IELEV = 2:

   1. Change in elevation from the cell inlet face to the cell outlet face.
   2. Feature implemented in TRACE to better support the conversion of models from RELAP5.
   3. Cells that increase in height have positive DH values, while cells that decrease in height have negative DH values.

For a more detailed discussion of using the DH term for elevation changes refer to the TRACE Users Guide, Volume 2

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Cell-Edge Flow-Channel Hydraulic Diameter (HD)

Hydraulic diameters are primarily used for the evaluation of pressure losses resulting from wall friction.

- If the cross section of the fluid volume is circular, the hydraulic diameter is the pipe diameter.
- If the cross section is not circular, the hydraulic diameter is evaluated using the standard formula:

  HD = 4 * FA / WP

  where FA is the flow area and WP is the wetted perimeter
- Hydraulic diameters are cell-edge values because wall friction is currently evaluated over the momentum cell (i.e., from one cell-center to the next cell-center)

Modeling Features Common to Multiple Component Types

Component Geometry – Modeling Cell/Edge Geometry in 1D Components:

Cell-Edge Flow-Channel Hydraulic Diameter (HD)

- When modeling an abrupt area change the recommended approach for the hydraulic diameter is:
  - ➢ Set HD to the smaller value
  - ➢ Turn off the wall friction calculation for this cell-edge entirely
  - ➢ Supply an appropriate K-factor

One source of common error is calculating the correct hydraulic diameter when flow paths are being "lumped" together such as combining multiple intact loops or combining all the steam generator tubes into one flow path. Keep in mind the wetted perimeter is not the wetted perimeter of an equivalent diameter circle after the flow areas have been lumped together, but rather is the sum of each individual flow path's wetted perimeter. If all of the flow paths being lumped together have the same hydraulic diameter, the hydraulic diameter is simply the diameter of a single flow path.

Modeling Features Common to Multiple Component Types

Friction – Modeling Pressure Losses in 1D Components:

Friction-factor correlation option (NFF)

Additive Loss Coefficient Array (FRIC/FRICR/KFAC/RKFAC)

Wall-friction Design Factor Multipliers (WFMFL/WFMFV)

Modeling Features Common to Multiple Component Types

Friction – Modeling Pressure Losses in 1D Components:

Friction-factor correlation option (NFF)

- NFF is the friction-factor correlation option flag (found in the Properties Window under “Friction”).
- Several options are available:
  - 1 applies a homogeneous-flow friction factor for wall and structure drag.
  - -1 is the same as a 1, but adds an internal form-loss computation for abrupt changes in flow area between mesh cells.
  - -100 applies the form-loss computation only.
  - 0 constant friction factor based on FRIC input.
- Recommend using friction-factor correlations (NFF) 1 or -1 everywhere except at an interface where flow choking is anticipated.
- Where flow choking is anticipated, use friction-factor correlation (NFF) 0. This avoids the interface from becoming friction limited as the onset of flow choking is approached.

Modeling Features Common to Multiple Component Types

Friction – Modeling Pressure Losses in 1D Components:

Additive Loss Coefficient Array (FRIC/FRICR/KFAC/RKFAC)

- TRACE models all flow area changes as smooth area changes and evaluates only the Bernoulli-equation reversible pressure loss or gain associated with such an area change.
- Additive loss coefficients are input to account for all irreversible form losses in a modeled system (example: change in area, change in flow direction or a flow orifice).
- Specified in either of two forms:
  - FRIC [FRIC = K-factor*(D/L)]
  - K-factors
- It is highly recommended that K-factors be input (namelist variable IKFAC = 1). K-factors are more commonly used throughout the industry.
- TRACE internally converts input K-factors to FRIC
- Different forward and reverse K-factors or FRICs can be specified by setting namelist variable NFRC1 = 2.

Modeling Features Common to Multiple Component Types

Friction – Modeling Pressure Losses in 1D Components:

Wall-friction Design Factor Multipliers (WFMFL/WFMFV)

- User can supply multipliers to the wall friction calculation for the liquid and vapor phases when Namelist variable WMFL and/or WMFV is set to 1, respectively.
- These input values simply act as multipliers on the friction factor.
- The multiplicative factors must fall within the range from 0.9 to 1.1.

Modeling Features Common to Multiple Component Types

Special Modeling Options for 1D Components:

Cell-edge choked-flow model option (ICFLG)

Countercurrent flow limiting option (LCCFL)

1D level tracking (ILEV)

Modeling Features Common to Multiple Component Types

Special Modeling Options for 1D Components:

Cell-edge choked-flow model option (ICFLG)

- The critical flow model in TRACE is implemented as a modeling option that can be turned on or off at individual cell edges in 1D components.
- The model is turned on globally with Namelist variable ICFLOW:
  - ICFLOW = 0 turns off the choked flow model
  - ICFLOW = 1 (default) applies default choking model multipliers only at component junctions connected to a BREAK.
  - ICFLOW = 2 allows enabling/disabling choking model at all cell faces in the model
- When ICFLOW is 2 then three options are available at each cell face:
  - 0 = no choked flow calculation
  - 1 = choke flow calculation using default multipliers
  - 2 to 5 = choke flow calculation using Namelist variable defined multipliers.

Modeling Features Common to Multiple Component Types

Special Modeling Options for 1D Components:

Countercurrent flow limiting option (LCCFL)

- Countercurrent flow can occur at any location in a reactor system.
- The flow limitation is on the liquid downflow rate for a given vapor upflow rate.
- The CCFL calculation is performed at cell faces flagged by the user.
- CCFL model is implemented by:
    - Inputting the CCFL data set cards (CCFL Models in the Navigator Window in SNAP)
    - Activating the model at a cell edge (Component Geometry in the Properties Window in SNAP) within a hydrodynamic component.

Modeling Features Common to Multiple Component Types

Special Modeling Options for 1D Components:

1D level tracking (ILEV)

- Tracks the two-phase level in a system
- Only vertical oriented components can have the level tracking model turned on.
- The model is implemented through Namelist variable NOLT1D
  - NOLT1D = 1 (default) disable level tracking for all 1D components
  - NOLT1D = 0 allows level tracking to occur, but requires the input of array ILEV for all 1D cells
  - NOLT1D = -1 forces level tracking to be used everywhere without requiring additional component input
- ILEV is input for each 1D component if NOLT1D = 0
  - ILEV = 1 indicates that the two-phase level exists in the current cell
  - ILEV = 0 indicates that the two-phase level does not exist in the current cell
  - ILEV = -1 indicates the level tracking calculation will be turned off for this cell

Modeling Features Common to Multiple Component Types

Pipe Wall – Modeling Pipe Walls :

The heat transfer calculation in TRACE is based on conduction through solid structures and convection at structure surfaces to the hydraulic channel contacting fluid. One-dimensional heat transfer may be evaluated across the cylindrical wall of several 1D hydraulic components.

Required input for 1D component wall heat transfer are:

- Number of radial heat transfer nodes
- Radius of the wall inner surface
- Wall thickness
- Wall outer-surface liquid and gas heat transfer coefficients and temperatures
- Wall material identifier
- Volumetric heat source/sink
- Wall temperature distribution

> The input for the 1D wall heat transfer is applied to all cells within the component regardless of any area changes that may occur within the hydrodynamic component. The user must be cautious of not doubling up on pipe wall heat structures by inadvertently adding a HTSTR component to model the same metal structure that was modeled by the component's internal heat structure capability.

Modeling Features Common to Multiple Component Types

Initial Conditions :

Each 1D hydraulic component requires initial conditions to be input for each cell. These initial conditions are:

- Void fraction
- Liquid velocity
- Vapor velocity
- Liquid temperature
- Vapor temperature
- Pressure
- Non-condensable pressure
- Wall temperature (if wall structure modeled)

Typically, rough estimates of the expected steady state conditions are entered for the initial conditions

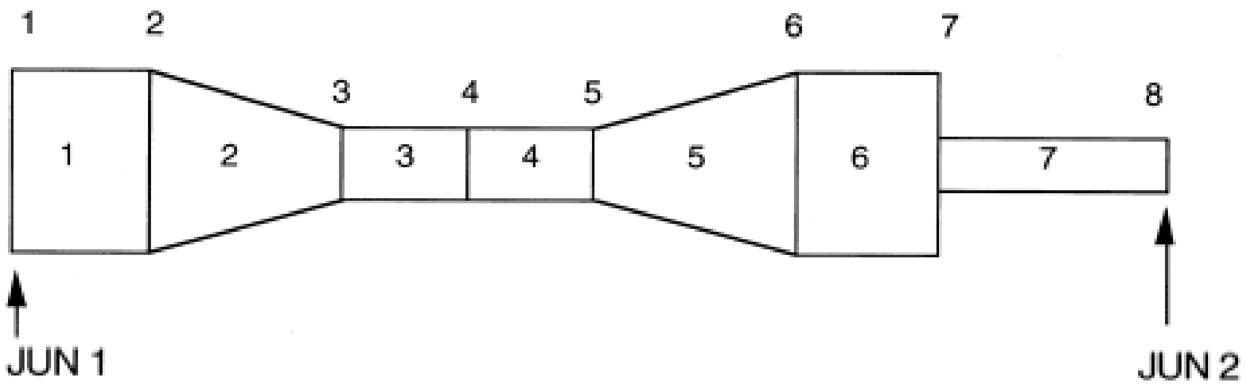
The TRACE model is then executed, allowing the code to calculate an accurate set of steady conditions from which to begin transient calculations.

PIPE Component

The PIPE component is a basic building block in TRACE. It models coolant flow in a 1D tube, channel, duct, or pipe.

- It can be used with BREAK and/or FILL component boundary conditions to model 1D flow in a pipe
- It can be used as a connecting pipe between other components to model a system
- It has the capability to model flow area changes, wall heat structures and heat transfer between the wall inner and outer surfaces.

PIPE Component



- Geometry is specified by providing a volume and length for each cell and a flow area and hydraulic diameter for each cell face.
- The junction interface variables, JUN 1 and JUN 2, provide reference numbers for connecting the PIPE to other components.

Reference numbers for JUN 1 and JUN 2 are automatically assigned when using the SNAP connect tool to connect two components together. If a junction number is not assigned, i.e. 0 is input for the junction number, the component is "dead-ended".

PIPE Component

Setting the number of cells to zero, turns the PIPE component into a Single Junction Component (SJC).

- Namelist variable USESJC needs to be set to greater than 0.
  - USESJC = 0 (default) - No SJC
  - USESJC = 1 - PIPE, VALVE and PUMP can be used as a SJC by setting the number of cells to zero
  - USESJC = 2 - PIPE VALVE and PUMP can have side junctions (requires additional input) with the side junction connecting at 90^{o}.
  - USESJC = 3 – Same as 2 except it requires additional side junction angle input.
- The SJC was developed to allow for the conversion of RELAP5 input models into TRACE input models.
- In general, a SJC is appropriate when there is essentially no fluid volume associated with the flow path. For example, a small break in the side of a PIPE component can be modeled with a SJC.

PIPE Component

Special Model Options for the PIPE Component:

0. Normal pipe – no special model options
1. Accumulator (non-spherical) option – calculates water level, volumetric flow, and liquid volume discharge. Implementation of interface sharpener.
2. Same as 1. but with level tracking (gas phase is not allowed to cross the exit cell edge as long as there is liquid level.
3. Spherical Accumulator – The number of cells must be set to one.
4. CANDU horizontal pressure tube fuel bundle – Not yet active.
5. Model falling film condensation heat transfer in vertical tube bundles.
6. Model condensation phenomena is a suppression pool – Not yet active.
7. This PIPE is connected to HTSTR components that have the fine mesh model turned on – Requires additional input
8. Model wall condensation phenomena as would be appropriate for a drywell
9. Pressurizer model – Requires additional input

PIPE Component

PIPE Component Side Junctions:

The PIPE component can have side junctions when Namelist variable USESJC is set to 2 or 3. The PIPE component in conjunction with side junctions can be used to model a tee or plenum.

- Each cell can have any number of side junctions attached to it
- Requires additional input such as junction connection numbers, side junction angles, activation of the offtake model.
- The side junction angle is defined as the angle from the low-numbered cell end of the main pipe path to the side arm component.

FILL Component

Fill Component

- Used to impose boundary conditions at any 1D hydraulic component junction
- Does not perform any hydrodynamic or heat transfer calculations
- Imposes a coolant velocity or mass flow boundary condition at the junction with its adjacent component
- Pressure, void fraction, fluid temperatures, non-condensable-gas partial pressure, and solute concentration that are specified for the FILL define the properties of the fluid convected into the adjacent component if an inflow condition occurs.
- Inflow to the adjacent component corresponds to a positive velocity at the FILL component’s JUN1 junction.
- A FILL cannot be connected directly to a BREAK or PLENUM component
- A FILL can be connected directly to a VESSEL component or a side junction in a 1D component

FILL Component

There are 11 different options specifying how a FILL component behaves. They can be grouped into four categories according to how the boundary conditions are varied:

1. Constant Velocity/Mass Flow/Generalized State (IFTY = 1/2/3)
2. Velocity/Mass Flow/Generalized State Table (IFTY = 4/5/6)
3. Trip-controlled Velocity/Mass Flow/Generalized State Table (IFTY = 7/8/9
4. Generalized State via Control System with or without slip (IFTY = 10/11)

HTSTR Component

- Models heat conduction within solid structures and specifies interfaces with fluid components, other solid structures through radiation, or specific boundary conditions
- Generic model building block that may be made as specific as needed for representing physical structures
- Lumped parameter or up to 2D conduction in Cartesian, cylindrical or spherical geometry
- Models powered or passive solid structures
- Fine mesh rezoning based on temperature differences between adjacent mesh cells for simulating reflood
- Model for Zirconium metal-water reaction and fuel cladding interaction gap heat transfer.

Material Properties Data

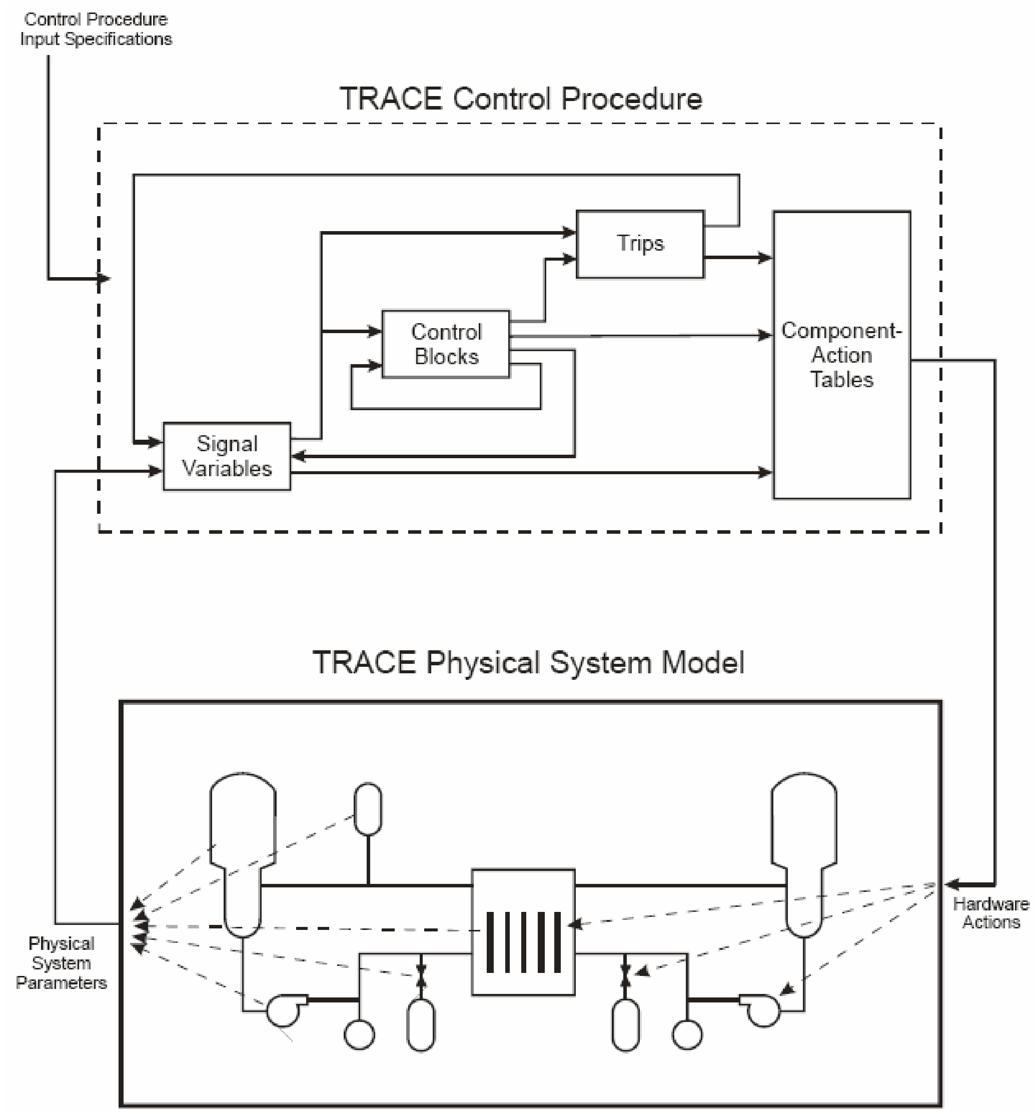
- TRACE provides the user with the option of using built-in material properties or inputting user defined properties.
- The input consists of temperature-dependent properties for thermal conductivity, specific heat, density, and spectral emissivity for use in heat transfer calculations.
- An extensive library of 12 temperature-dependent material properties is incorporated into TRACE. The materials and their indexes are:
  - 1 – mixed-oxide fuel; 2 - Zircaloy;
  - 3 – fuel-clad gap gases; 4 – boron-nitride insulation;
  - 5 – Constantan/Nicrome heater coil; 6 – stainless steel, type 304;
  - 7 – stainless steel, type 316; 8 – stainless steel, type 347;
  - 9 – carbon steel, type A508; 10 – Inconel, type 718;
  - 11 - Zircaloy dioxide; 12 – Inconel, type 600
- To implement user defined properties, the user must provide the following:
  - a unique material identifier (MATB > 50),
  - the number of data-point sets provided for each material (PTBLN),
  - the data tables and/or functional fits that specify the material properties as a function of temperature.

The built-in/user defined materials are accessed in the HTSTR component input under Radial Geometry in SNAP. The User defined material properties input is located in the Navigator Window under Thermal.

Control System: Signal Variables and Control Blocks

Control System

The control system in TRACE is general and flexible. It consists of signal variables, control blocks and trips. These components can be combined into control-procedure models that may be used to control water levels, system pressures, pump speeds, etc. A conceptual model of the TRACE control system and how it interacts with the model of the physical system is shown.



Control System: Signal Variables and Control Blocks

Signal Variables

Signal variables are the only means by which information is communicated to the control system from the rest of an input model. Signal variables are identified by positive numbers.

Signal inputs can originate from parameters that are:

- General Signals - global to an entire model (for example, the problem time),
- Volume and Edge Signals - part of the hydrodynamic component database (for example, a pressure, temperature, density, mass flow or velocity),
- Component Signals – part of the component specific database (e.g, pump head),
- Heat Signals - part of the heat structure database (e.g., a wall temperature),
- Power Signals - part of the kinetics database (e.g., the core power),
- Control Signals - part of the control system itself (e.g., trip output signals)

SNAP categorizes the signal variables into general signals, volume signals, edge signals, component signals, heat signals, power signals, control signals

Control System: Signal Variables and Control Blocks

Signal Variables

Signal variables are single-valued parameters by nature. However, TRACE has been designed to allow the signal variable to have up to six different functional forms which define what the output signal represents:

- Exact form – the exact value of the desired parameter,
- Cell Diff form – the difference in some parameter between two different cells in a component,
- TimeDiff form – the difference in a parameter since the last time step,
- Min and Max forms – the minimum or maximum over some range of cells in a single component,
- VolAvg form – the volume average of a parameter over some range of cells in a single component

Control System: Signal Variables and Control Blocks

Signal Variables

Examples of the different functional forms for pressure in PIPE 10, Cells 1, 2:

```
*n: Pressure in Cell 1, PIPE 10
*
*d:     SV ID     Pressure     Comp ID     Cell No.     Cell No.
```

Function Form	SNAP View	ASCII Form					
Exact	Pressure in Cell 1 P 2 mode=exact value signal=Pipe 10	*	idsv 2	isvn 21	ilcn 10	icnl 1	icn2 0
CellDiff	DP Between Cells 1 and 2 P 3 mode=cell diff signal=Pipe 10	*	idsv 3	isvn -21	ilcn 10	icnl 1	icn2 2
TimeDiff	Press diff since last timestep P 4 mode=time diff signal=Pipe 10	*	idsv 4	isvn -21	ilcn 10	icnl 1	icn2 0
Min	Min press between Cells 1 and 2 P 5 mode=min value signal=Pipe 10	*	idsv 5	isvn 21	ilcn 10	icnl -1	icn2 -2
Max	Max press between Cells 1 and 2 P 6 mode=max value signal=Pipe 10	*	idsv 6	isvn 21	ilcn 10	icnl 1	icn2 2
VolAvg	Avg press between Cells 1 and 2 P 7 mode=volume avg signal=Pipe 10	*	idsv 7	isvn 21	ilcn 10	icnl 1	icn2 -2

Control System: Signal Variables and Control Blocks

Control Blocks

A control block is a mathematical function that operates on zero or more inputs defined by signal variables or other control blocks. Control blocks are identified by negative numbers.

The output from a control block may be used as:

- Input to another control block or signal variable
- A trip parameter
- The independent variable for component-action tables
- A source of data for component actions….

Control System: Signal Variables and Control Blocks

Control Blocks (continued)

Component Actions:

Parameters that can be controlled	Component
Pressure and fluid-state boundary condition	BREAK
Velocity or mass-flow and fluid-state boundary condition	FILL
Reactor-core programmed reactivity or neutronic power	POWER
Reactor-core axial-power shape	POWER
Energy deposition directly in the coolant	PIPE,TEE,TURB,FLPOWER
Energy generation in the hydro-component wall	PIPE,TEE,PUMP,VALVE
PUMP-impeller rotational speed	PUMP
Turbine power demand	TURB
VALVE flow-area fraction or relative stem position	VALVE

Control System: Signal Variables and Control Blocks

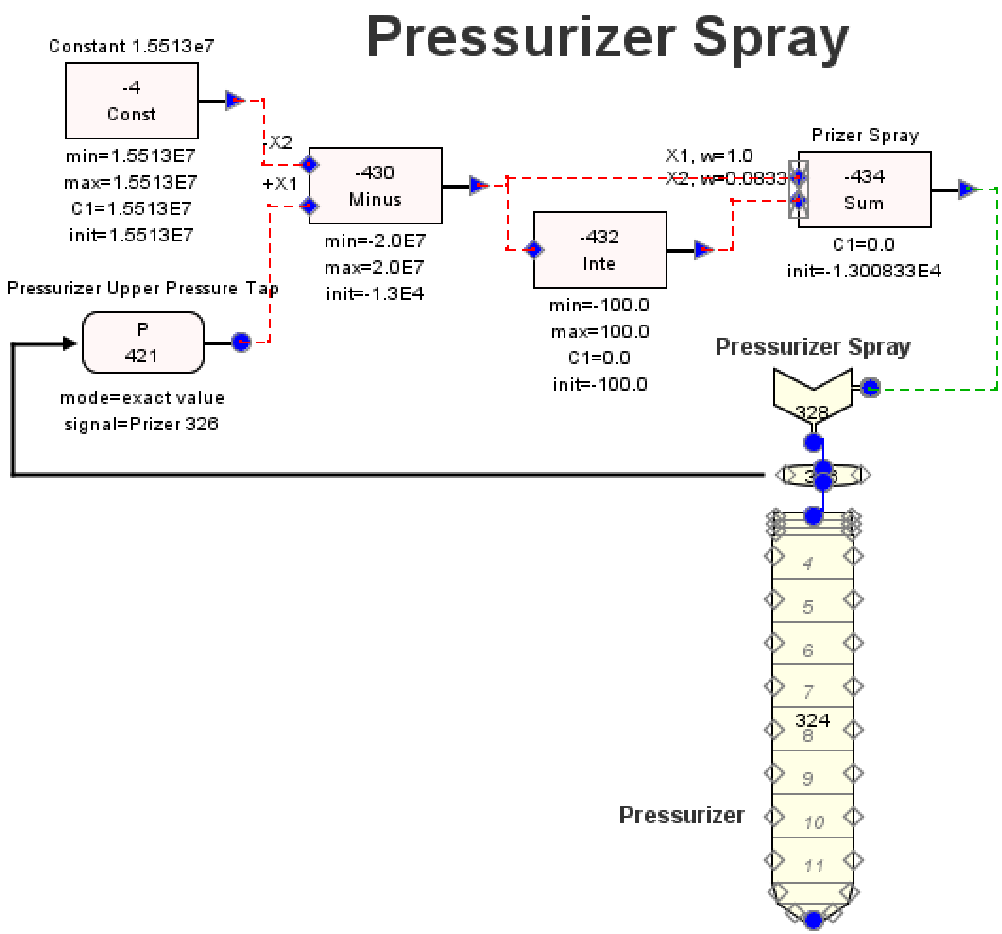
Control Blocks

Control Block Categories in SNAP

Category	Control Block Action
Arithmetic	constant, add, subtract, multiply,…..
Calculus	derivative, integrate, natural logarithm,…..
Controller	input switch, PID controller, pressure controller,…..
Logical	less than, greater than, logical AND,…..
Manipulation	absolute value, maximum of two, trip time,…..
Time	first order lag, time, ramp,…..
Trigonometric	sin, cos, arcsine,…..

Control System: Signal Variables and Control Blocks

Control System Example:



Time Step Data Input

The time step data controls the calculation and the output edits. A time step data set contains 9 inputs. Any number of time step sets can be entered.

- End Time: end time for this time segment.
- Minimum Size: the minimum time step size allowed for this time segment.
- Maximum Size: the maximum time step size allowed for this time segment.
- Heat vs Power Diff: suggested value is 10.0
- Max Conv. Power Diff: default value is 1.0e20
- Long Edit Interval: long-printout-edit time interval for this time segment.
- Graphics Interval: graphic-edit time interval for this time segment.
- Restart Interval: restart-edit time interval for this time segment
- Short Edit Interval: short-printout-edit time interval for this edit

End Time	Minimum Size	Maximum Size	Heat vs Fluid Size	Max Conv. Power Diff	Long Edit Interval	Graphics Interval	Restart Interval	Short Edit Interval
100.0	1.0E-6	0.1	10.0	1.0E20	50.0	1.0	100.0	10.0
500.0	1.0E-6	0.3	10.0	1.0E20	100.0	2.0	500.0	20.0
1500.0	1.0E-6	0.5	10.0	1.0E20	500.0	4.0	1500.0	100.0

Questions?

Any questions before moving on to the workshop exercise?