INEEL-EXT-98-00834
Revision 3.0
September 2009

RELAP5-3D© Code Manual Volume II: User's Guide and Input Requirements

The RELAP5-3D© Code Development Team

The INL is a U. S. Department of Energy National Laboratory operated by Battelle Energy Alliance

INEEL-EXT-98-00834
Volume II
Revision 3.0

RELAP5-3D© CODE MANUAL
VOLUME II: USER'S GUIDE AND INPUT REQUIREMENTS

The RELAP5-3D© Code Development Team

Original Manuscript Completed:
February 1999
Revision 3.0 Completed:
September 2009

Idaho National Laboratory
Idaho Falls, Idaho 83415

Prepared Under DOE/NE Idaho Operations Office
Contract No. DE-AC07-05ID14517

The RELAP5-3D© manuals are living documents and are being corrected and updated continuously. A printed version of the manuals is frozen and archived when a code version is released. This version of the manual corresponds to RELAP5-3D© version 3.0, released September 2009.

ABSTRACT

The RELAP5-3D© code has been developed for best estimate transient simulation of light water reactor coolant systems during postulated accidents. The code models the coupled behavior of the reactor coolant system and the core for loss-of-coolant accidents, and operational transients, such as anticipated transient without scram, loss of offsite power, loss of feedwater, and loss of flow. A generic modeling approach is used that permits simulating a variety of thermal hydraulic systems. Control system and secondary system components are included to permit modeling of plant controls, turbines, condensers, and secondary feedwater systems.

RELAP5-3D© code documentation is divided into six volumes: Volume I provides modeling theory and associated numerical schemes; Volume II contains detailed instructions for code application and input data preparation; Volume III provides the results of developmental assessment cases that demonstrate and verify the models used in the code; Volume IV presents a detailed discussion of RELAP5-3D© models and correlations; Volume V contains guidelines that have evolved over the past several years through the use of the RELAP5 code; Volume VI discusses the numerical scheme used in RELAP5-3D©.

CONTENTS

1 Introduction 1-1
1.1 General 1-1
1.2 Areas of Application 1-1
1.3 Modeling Philosophy 1-1
2 Hydrodynamics 2-1
2.1 Basic Flow Model 2-2
2.2 State Relationships 2-14
2.3 Process Models 2-14
2.3.1 Abrupt Area Change 2-15
2.3.2 Choked Flow 2-15
2.3.3 Branching 2-18
2.3.4 Noncondensables 2-25
2.3.5 Water Packing 2-28
2.3.6 Countercurrent Flow Limitation Model 2-28
2.3.7 Mixture Level Tracking Model 2-30
2.3.8 Thermal Stratification Model 2-31
2.3.9 Energy Conservation at an Abrupt Change 2-31
2.3.10 Jet Junction Model 2-31
2.3.11 Variable Volume Model 2-31
2.3.12 References 2-32
2.4 Hydrodynamic Components 2-33
2.4.1 Common Features of Components 2-33
2.4.2 Time-Dependent Volume 2-43
2.4.3 Time-Dependent Junction 2-44
2.4.4 Single-Volume 2-45
2.4.5 Single-Junction 2-45
2.4.6 Pipe 2-45
2.4.7 Branch 2-46
2.4.8 Pump 2-46
2.4.9 Jet Pump 2-59
2.4.10 Valves 2-62
2.4.11 Separator 2-65
2.4.12 Turbine 2-70
2.4.13 Accumulator 2-74
2.4.14 Annulus 2-75
2.4.15 ECC Mixer 2-76

2.4.16 Multi-Dimensional Component 2-76
2.4.17 Pressurizer 2-79
2.4.18 Feedwater Heater 2-80
2.4.19 Compressor 2-80
2.4.20 References 2-83

3 Heat Structures 3-1
3.0.1 Reference 3-1
3.1 Heat Structure Geometry 3-1
3.2 Heat Structure Boundary Conditions 3-4
3.3 Heat Structure Sources 3-8
3.4 Heat Structure Changes at Restart 3-9
3.5 Heat Structure Output and Recommended Uses 3-9
3.6 Reflood Model 3-10

4 Trips and Controls 4-1
4.1 Trips 4-1
4.1.1 Variable Trips 4-2
4.1.2 Logical Trips 4-3
4.1.3 Trip Execution 4-4
4.1.4 Trip Logic Example 1 4-4
4.1.5 Trip Logic Example 2 4-6
4.2 Control Components 4-7
4.2.1 Basic Control Components 4-8
4.2.2 Control System Examples 4-12
4.2.3 Shaft Control Component (SHAFT) 4-14

5 Reactor Kinetics 5-1
5.1 Power Computation Options 5-1
5.1.1 References 5-2
5.2 Reactivity Feedback Options 5-2

6 General Tables and Component Tables 6-1

7 Initial and Boundary Conditions 7-1
7.1 Initial Conditions 7-1
7.1.1 Input Initial Values 7-1
7.1.2 Steady-State Initialization 7-2
7.2 Boundary Conditions 7-4
7.2.1 Mass Sources or Sinks 7-4

7.2.2 Pressure Boundary 7-5

8 Problem Control 8-1

8.1 Problem Types and Options 8-1

8.1.1 References 8-1

8.2 Time Step Control 8-1

8.3 Printed Output 8-5

8.3.1 Input Editing 8-5

8.3.2 Major Edits 8-6

8.3.3 Minor Edits 8-24

8.3.4 Diagnostic Edit 8-24

8.4 Plotted Output 8-29

8.4.1 External Plots 8-29

8.4.2 Internal Plots 8-29

8.5 RELAP5-3D© Control Card Requirements 8-31

8.6 Transient Termination 8-31

8.7 Problem Changes at Restart 8-31

FIGURES

Figure 2.1-1. Possible volume orientation specifications. 2-5
Figure 2.1-2. Horizontal volume schematic showing face numbers for one-dimensional components. 2-6
Figure 2.1-3. Vertical volume schematic showing face numbers for one-dimensional components. 2-7
Figure 2.1-4. Cartesian control volume showing face numbers for multi-dimensional components. 2-9
Figure 2.1-5. Cylindrical control volume showing face numbers for multi-dimensional components. 2-9
Figure 2.1-6. Sketch of possible coordinate orientation for three volumes and two junctions. 2-10
Figure 2.1-7. Sketch of possible vertical volume connections. 2-10
Figure 2.2-1. Pressure-temperature diagram. 2-14
Figure 2.3-1. A 90-degree tee model using a crossflow junction. 2-19
Figure 2.3-3. Typical branching junctions. 2-20
Figure 2.3-2. Tee model using a branch component. 2-20
Figure 2.3-4. Plenum model using a branch. 2-21
Figure 2.3-5. Leak path model using the crossflow junction. 2-22
Figure 2.3-6. High-resistance flow path model. 2-24
Figure 2.4-1. Four-quadrant head curve for Semiscale MOD1 pump (ANC-A-2083). 2-50
Figure 2.4-2. Four-quadrant torque curve for Semiscale MOD1 pump (ANC-A-3449). 2-51
Figure 2.4-3. Homologous head curve. 2-54
Figure 2.4-4. Homologous torque curve. 2-55
Figure 2.4-5. Schematic of mixing junctions. 2-60
Figure 2.4-6. Jet pump model design. 2-62
Figure 2.4-7. Schematic of separator. 2-66
Figure 2.4-8. Physical picture of a separator. 2-67
Figure 2.4-9. Separator volume fraction of liquid fluxed out the liquid outlet. 2-68
Figure 2.4-10. Separator volume fraction of vapor/gas fluxed out the vapor/gas outlet. 2-68
Figure 2.4-11. Schematic of a cylindrical accumulator. 2-75
Figure 2.4-12. Schematic of a spherical accumulator. 2-76
Figure 2.4-13. Schematic of an accumulator showing standpipe/surgeline inlet. 2-77
Figure 2.4-14. Possible accumulator configurations. 2-78
Figure 2.4-15. Schematic of a feedwater heater. 2-80
Figure 2.4-16. Compressor performance map. 2-82
Figure 3.1-1. Mesh point layout. 3-2

Figure 6.0-1. Input data for a power-type general table and graph. 6-2
Figure 8.3-1. Example of major edit..... 8-8
Figure 8.3-2. Example of additional output for pumps, compressors, accumulators, and turbines..... 8-15
Figure 8.3-3. Example of reflood major edit. 8-21
Figure 8.3-4. Example of cladding oxidation and rupture major edit. 8-23
Figure 8.3-5. Example of radiation major edit. 8-24
Figure 8.3-6. Example of minor edit. 8-25
Figure 8.3-7. Example of printout before the diagnostic edit when a failure occurs. 8-27
Figure 8.3-8. Example of printout buried in the diagnostic edit when a failure occurs. 8-28
Figure 8.4-1. Strip input file. 8-29

TABLES

Table 2.1-1. Flow regime letters and numbers. 2-11
Table 2.1-2. Bubbly/slug flow regime numbers for vertical junctions. 2-12
Table 2.3-1. Values of m, c_7, and c_8 for Tien's CCFL correlation form. 2-30
Table 2.4-1. Area change options. 2-41
Table 2.4-2. Pump homologous curve definitions. 2-52
Table 3.2-1. Boundary condition options. 3-7
Table 4.1-1. Logical operations. 4-3
Table 4.1-2. Truth table examples. 4-5
Table 4.1-3. Boolean algebra identities. 4-6
Table 4.2-1. Input data for a sample problem to test pump, generator, and shaft. 4-19
Table 8.2-1. Time step selection options. 8-3
Table 8.3-1. Flow map identifiers. 8-6

EXECUTIVE SUMMARY

The RELAP5 series of codes has been developed at the Idaho National Laboratory (INL) under sponsorship of the U.S. Department of Energy, the U.S. Nuclear Regulatory Commission, members of the International Code Assessment and Applications Program (ICAP), members of the Code Applications and Maintenance Program (CAMP), and members of the International RELAP5 Users Group (IRUG). Specific applications of the code have included simulations of transients in light water reactor (LWR) systems such as loss of coolant, anticipated transients without scram (ATWS), and operational transients such as loss of feedwater, loss of offsite power, station blackout, and turbine trip. RELAP5-3D©, the latest in the series of RELAP5 codes, is a highly generic code that, in addition to calculating the behavior of a reactor coolant system during a transient, can be used for simulation of a wide variety of hydraulic and thermal transients in both nuclear and nonnuclear systems involving mixtures of vapor, liquid, noncondensable gases, and nonvolatile solute.

The mission of the RELAP5-3D© development program was to develop a code version suitable for the analysis of all transients and postulated accidents in LWR systems, including both large- and small-break loss-of-coolant accidents (LOCAs) as well as the full range of operational transients.

The RELAP5-3D© code is based on a nonhomogeneous and nonequilibrium model for the two-phase system that is solved by a fast, partially implicit numerical scheme to permit economical calculation of system transients. The objective of the RELAP5-3D© development effort from the outset was to produce a code that included important first-order effects necessary for accurate prediction of system transients but that was sufficiently simple and cost effective so that parametric or sensitivity studies are possible.

The code includes many generic component models from which general systems can be simulated. The component models include pumps, valves, pipes, heat releasing or absorbing structures, reactor kinetics, electric heaters, jet pumps, turbines, compressors, separators, annuli, pressurizers, feedwater heaters, ECC mixers, accumulators, and control system components. In addition, special process models are included for effects such as form loss, flow at an abrupt area change, branching, choked flow, boron tracking, and noncondensable gas transport.

The system mathematical models are coupled into an efficient code structure. The code includes extensive input checking capability to help the user discover input errors and inconsistencies. Also included are free-format input, restart, renodalization, and variable output edit features. These user conveniences were developed in recognition that generally the major cost associated with the use of a system transient code is in the engineering labor and time involved in accumulating system data and developing system models, while the computer cost associated with generation of the final result is usually small.

The development of the models and code revisions that constitute RELAP5-3D© has spanned more than two decades from the early stages of RELAP5-3D© numerical scheme development (circa 1976) to the present. RELAP5-3D© represents the aggregate accumulation of experience in modeling core behavior

during accidents, two-phase flow process, and LWR systems. The code development has benefited from extensive application and comparison to experimental data in the LOFT, PBF, Semiscale, ACRR, NRU, and other experimental programs.

The RELAP5-3D© version contains several important enhancements over previous versions of the code. The most prominent attribute that distinguishes the RELAP5-3D© code from the previous versions is the fully integrated, multi-dimensional thermal- hydraulic and kinetic modeling capability. This removes any restrictions on the applicability of the code to the full range of postulated reactor accidents. Enhancements include a new matrix solver for 3D problems, new thermodynamic properties for water, and improved time advancement for greater robustness. Enhancements also include all features and models previously available in the ATHENA configuration version of the code, which are as follows: addition of new working fluids (e.g., ammonia, blood, carbon dioxide, glycerol, helium, helium-xenon, hydrogen, lead-bismuth, lithium, lithium-lead, molten salts, nitrogen, potassium, sodium, sodium-potassium, and xenon) and a magneto-hydrodynamic model. The multi-dimensional component in RELAP5-3D© was developed to allow the user to more accurately model the multi-dimensional flow behavior that can be exhibited in any component or region of a LWR system. Typically, this will be the lower plenum, core, upper plenum and downcomer regions of an LWR. However, the model is general, and is not restricted to use in the reactor vessel. The component defines a one, two, or three- dimensional array of volumes and the internal junctions connecting them. The geometry can be either Cartesian (x, y, z) or cylindrical (r, θ, z). An orthogonal, three-dimensional grid is defined by mesh interval input data in each of the three coordinate directions. The multi-dimensional neutron kinetics model in RELAP5-3D© is based on the NESTLE code, which solves the two or four group neutron diffusion equations in either Cartesian or hexagonal geometry using the Nodal Expansion Method (NEM) and the non-linear iteration technique. Three, two, or one-dimensional models may be used. Several different core symmetry options are available including quarter, half, and full core options for Cartesian geometry and 1/6, 1/3, and full core options for hexagonal geometry. Zero flux, non-reentrant current, reflective, and cyclic boundary conditions are available. The steady-state eigenvalue and time dependent neutron flux problems can be solved by the NESTLE code as implemented in RELAP5-3D©. The new Border-Profile Lower Upper (BPLU) matrix solver is used to efficiently solve sparse linear systems of the form AX = B. BPLU is designed to take advantage of pipelines, vector hardware, and shared-memory parallel architecture to run fast. BPLU is most efficient for solving systems that correspond to networks, such as pipes, but is efficient for any system that it can permute into border-banded form. Speed-ups over the previously used sparse matrix solver are achieved in RELAP5-3D© running with BPLU on multi-dimensional problems, for which it was intended. For one-dimensional problems, the BPLU solver runs as fast or faster than the previously used sparse matrix solver.

The RELAP5-3D© code manual consists of six separate volumes. The modeling theory and associated numerical schemes are described in Volume I, to acquaint the user with the modeling base and thus aid in effective use of the code. Volume II contains more detailed instructions for code application and specific instructions for input data preparation.

Volume III provides the results of developmental assessment cases run with RELAP5-3D© to demonstrate and verify the models used in the code. The assessment matrix contains phenomenological problems, separate-effects tests, and integral systems tests.

Volume IV contains a detailed discussion of the models and correlations used in RELAP5-3D©. It provides the user with the underlying assumptions and simplifications used to generate and implement the base equations into the code so that an intelligent assessment of the applicability and accuracy of the resulting calculations can be made. Thus, the user can determine whether RELAP5-3D© is capable of modeling his or her particular application, whether the calculated results will be directly comparable to measurement or whether they must be interpreted in an average sense, and whether the results can be used to make quantitative decisions.

Volume V provides guidelines for users that have evolved over the past several years from applications of the RELAP5-3D© code at the Idaho National Laboratory, at other national laboratories, and by users throughout the world.

Volume VI discusses the numerical scheme in RELAP5-3D©.

ACKNOWLEDGMENTS

Development of a complex computer code such as RELAP5-3D© is the result of team effort and requires the diverse talents of a large number of people. Special acknowledgment is given to those who pioneered in the development of the RELAP5 series of codes. In particular, V. H. Ransom, J. A. Trapp, and R. J. Wagner. A number of other people have made and continue to make significant contributions to the continuing development of the RELAP5-3D© code. Recognition and gratitude is given to the other current members of the RELAP5-3D© team:

N. A. Anderson	H-H. Kuo	R. A. Riemke
P. D. Bayless	N. S. Larson	R. R. Schultz
V. T. Berta	M. A. Lintner	A. S-L. Shieh
K. E. Carlson	G. L. Mesina	R. W. Shumway
P. P. Cebull	C. S. Miller	C. E. Slater
C. B. Davis	G. A. Mortensen	S. M. Sloan
J. E. Fisher	P. E. Murray	J. E. Tolli
C. D. Fletcher	C. E. Nielson	W. L. Weaver
E. C. Johnsen	R. B. Nielson	J. R. Wolf
G. W. Johnsen	S. Paik	

The list of contributors is incomplete, as many others have made significant contributions in the past. Rather than attempt to list them all and risk unknowingly omitting some who have contributed, we acknowledge them as a group and express our appreciation for their contributions to the success of the RELAP5-3D© effort.

Finally, acknowledgment is made of all the code users who have been very helpful in stimulating timely correction of code deficiencies and suggesting improvements.

1 Introduction

The purpose of this volume is to help educate the code user by documenting the modeling experience accumulated from developmental assessment and application of the RELAP5-3D© code. This information includes a blend of the model developers' recommendations with respect to how the model is intended to be applied and the application experience that indicates what has been found to work or not to work. Where possible, approaches known to work are definitely recommended, and approaches known not to work are pointed out as pitfalls to avoid.

1.1 General

The objective of the user's guide is to reduce the uncertainty associated with user simulation of light water reactor (LWR) systems. However, we do not imply that uncertainty can be eliminated or even quantified in all cases, since the range of possible system configurations and transients that could occur is large and constantly evolving. Hence, the effects of nodalization, time step selection, and modeling approach are not completely quantified. As the assessment proceeds, there will be a continual need to update the user guidelines document to reflect the current state of simulation knowledge.

1.2 Areas of Application

RELAP5-3D© is a generic transient analysis code for thermal-hydraulic systems using a fluid that may be a mixture of vapor, liquid, noncondensable gases, and a nonvolatile solute.

The fluid and energy flow paths in fluid components are approximated by either one-dimensional stream tube or multi-dimensional models. The energy flow paths in solid heat conductors are approximated by either one-dimensional or two-dimensional heat conduction models; the two-dimensional model is used for reflood. The code contains system component models applicable to LWRs. In particular, pumps, turbines, generator, valves, separator, and controls are included. The code also contains a jet pump component and an ecc mixer component. A point kinetics model and a multi-dimensional nodal neutron kinetics model are available to simulate neutronics behavior.

The LWR applications for which the code is intended include accidents initiated from small break loss-of-coolant accidents, operational transients such as anticipated transients without SCRAM, loss of feed, loss-of-offsite power, and loss of flow transients. The reactor coolant system (RCS) behavior can be simulated up to and slightly beyond the point of fuel damage.

1.3 Modeling Philosophy

RELAP5-3D© is designed for use in analyzing system component interactions; it offers both non-detailed (one-dimensional) and detailed (multi-dimensional) simulations of fluid flow within components. As such, it contains the ability to model multi-dimensional effects, either for fluid flow, heat transfer, or reactor kinetics. It also allows the modeling of crossflow effects in a pressurized water reactor (PWR) core and the reflood modeling that uses a two-dimensional conduction solution in the vicinity of a

quench front. To further enhance the overall system modeling capability, a control system model is included. This model provides a way to perform basic mathematical operations, such as addition, multiplication, integration, and control components such as proportional-integral, lag, and lead-lag controllers, for use with the basic fluid, thermal, and component variables calculated by the remainder of the code. This capability can be used to construct models of system controls or components that can be described by algebraic and differential equations. The code numerical solution includes the evaluation and numerical time advancement of the control system coupled to the fluid and thermal system.

The hydrodynamic model and the associated numerical scheme are based on the use of fluid control volumes and junctions to represent the spatial character of the flow. For the one-dimensional model, the control volumes can be viewed as stream tubes having inlet and outlet junctions. The control volume has a direction associated with it that is positive from the inlet to the outlet. Velocities are located at the junctions and are associated with mass and energy flow between control volumes. Control volumes are connected in series, using junctions to represent a flow path. All internal flow paths, such as recirculation flows, must be explicitly modeled in this way since only single liquid and vapor/gas velocities are represented at a junction. (In other words, a countercurrent liquid-liquid flow cannot be represented by a single-junction.) For flows in pipes, there is little confusion with respect to nodalization. However, in a steam generator having a separator and recirculation flow paths, some experience is needed to select a nodalization that will give correct results under all conditions of interest. Nodalization of branches or tees also requires more guidance. For the multi-dimensional model, use of control volumes and junctions are also used based on Cartesian or cylindrical coordinates.

Heat conduction flow paths are usually modeled in a one-dimensional sense, using a finite difference mesh to calculate temperatures and heat flux vectors. The heat conductors can be connected to hydrodynamic volumes to simulate a heat flow path normal to the fluid flow path. The heat conductor or heat structure is thermally connected to the hydrodynamic volume through a heat flux that is calculated using heat transfer correlations. Electrical or nuclear heating of the heat structure can also be modeled as either a surface heat flux or as a volumetric heat source. The heat structures are used to simulate pipe walls, heater elements, nuclear fuel pins, and heat exchanger surfaces.

A special, two-dimensional, heat conduction solution method with an automatic fine mesh rezoning is used for low-pressure reflood. Both axial and radial conduction are modeled, and the axial mesh spacing is refined as needed to resolve the axial thermal gradient. The hydrodynamic volume associated with the heat structure is not rezoned, and a spatial boiling curve is constructed and used to establish the convection heat transfer boundary condition. At present, this capability is specialized to the LWR core reflood process, but the plan is to generalize this model to higher pressure situations so that it can be used to track a quench front anywhere in the system.

The point reactor kinetics model is advanced in a serial and implicit manner after the heat conduction-transfer and hydrodynamic advancements but before the control system advancement. The kinetics model consists of a system of ordinary differential equations integrated using a modified Runge-Kutta technique. The integration time step is regulated by a truncation error control and may be less than the hydrodynamic time step; however, the thermal and fluid boundary conditions are held fixed over

each hydrodynamic time interval. The reactivity feedback effects of fuel temperature, moderator temperature, moderator density, and boron concentration in the moderator are evaluated, using averages over the hydrodynamic control volumes and associated heat structures that represent the core. The averages are weighted averages established a priori such that they represent the effect on total core power. Certain nonlinear or multidimensional effects caused by spatial variations of the feedback parameters cannot be accounted for with such a model. Thus, the user must judge whether or not the model is a reasonable approximation of the physical situation being modeled. A multi-dimensional nodal neutron kinetics model is also available.

The control system model provides a way for simulating any lumped process, such as controls or instrumentation, in which the process can be defined in terms of system variables through logical, algebraic, differentiating, or integrating operations. These models do not have a spatial variable and are integrated with respect to time. The control system is coupled to the thermal and hydrodynamic components serially and implicitly. The control system advancement occurs after the heat conduction transfer, hydrodynamic, and reactor kinetics advancements and uses the same time step as the hydrodynamics so that new time thermal and hydrodynamic information is used in the control model advancement. However, the control variables are fed back to the thermal and hydrodynamic model in the succeeding time step, i.e., they are explicitly coupled.

A system code such as RELAP5-3D© contains numerous approximations to the behavior of a real, continuous system. These approximations are necessitated by the finite storage capability of computers, by the need to obtain a calculated result in a reasonable amount of computer time, and in many cases because of limited knowledge about the physical behavior of the components and processes modeled. For example, knowledge is limited for components such as pumps and separators, processes such as two-phase flow, and heat transfer. Examples of approximations required because of limited computer resources are limited spatial nodalization for hydrodynamics, heat transfer, and kinetics; and density from thermodynamic property tables. In general, the accuracy effect of each of these factors is of the same order; thus, improving one approximation without a corresponding increase in the others will not necessarily lead to a corresponding increase in physical accuracy. At the present time, very little quantitative information is available regarding the relative accuracies and their interactions. What is known has been established through applications and comparison of simulation results to experimental data. Progress is being made in this area as the code is used; but there is, and will be for some time, a need to continue the effort to quantify the system simulation capabilities.

2 Hydrodynamics

The hydrodynamics simulation is based on a one-dimensional model and a multi-dimensional model of the transient flow for a vapor-liquid-noncondensable gas mixture. The numerical solution scheme used results in a system representation using control volumes connected by junctions. A physical system consisting of flow paths, volumes, areas, etc., is simulated by constructing a network of control volumes connected by junctions. The transformation of the physical system to a system of volumes and junctions is an inexact process, and there is no substitute for experience. General guidelines have evolved though application work using RELAP5-3D©. The purpose here is to summarize these guidelines.

In selecting a nodalization for hydrodynamics, the following general rules should be followed:

1. The length of volumes should be such that all have similar material Courant limits, i.e., flow length divided by velocity about the same. (Expected velocities during the transient must be considered.)

2. The volumes should have $\frac{L}{D} \geq 1$ for typical engineering problems. However, special cases may require smaller volumes to resolve sharp gradients, such as the emptying of the pressurizer or the choking at the end of a straight pipe. For two-phase applications, lengths less than the physical length scale (such as the length of a Taylor bubble) may not provide any improvement in the accuracy of the results. Similarly, a finer nodalization may not improve the representation of mixing in tees. For single-phase applications, a finer nodalization should not cause concerns except for those relative to the code's assumption of fully developed flow. This is discussed further in Volume V, Section 2.2.2.2.

3. The total system cannot exceed the computer resources. RELAP5-3D© dynamically allocates memory based on the requirements of each problem, and most models require memory based on factors such as the number of volumes, junctions, number of heat structures and the number of meshes, and the number and length of various user-input tables. The number of hydrodynamic volumes is a reasonable measure of problem size, and typical LWR systems with over 600 volumes have been run on workstations with 32 Mbytes of memory. The memory should be sufficiently large to avoid paging during transient advancement.

4. If possible, a nodalization sensitivity study should be made in order to estimate the uncertainty owing to nodalization. Volume V provides guidance and examples of appropriate nodalizations for reactor systems.

5. Avoid nodalizations where a sharp density gradient coincides with a junction (a liquid interface, for example) at steady-state or during most of the transient. This type of situation can result in time-step reduction and increased computer cost.

6. Eliminate minor flow paths that do not play a role in system behavior or are insignificant compared to the accuracy of the system representation. This can not usually be done until

some preliminary trial calculations have been made that include all the flow paths. Care must be used here because in certain situations flow through minor flow paths can have a significant effect on system behavior. An example is the effect of hot-to-cold-leg leakage on core level depression in a PWR under small break loss-of-coolant accident conditions.

7. Establish the flow and pressure boundaries of the system beyond which modeling is not required and specify appropriate boundary conditions at these locations.

2.1 Basic Flow Model

The RELAP5-3D© flow model is a nonhomogeneous, nonequilibrium two-phase flow model. See Section 3 of Volume I for a detailed description of the model and the governing equations. Options exist for homogeneous, equilibrium, or frictionless models if desired. These options are included to facilitate comparisons with other homogeneous and/or equilibrium codes. Generally, the code will not run faster if these options are selected.

The RELAP5-3D© flow model includes a one-dimensional, stream-tube formulation option in which the bulk flow properties are assumed to be uniform over the fluid passage cross-section. The control volumes are finite increments of the flow passage and may have a junction at the inlet or outlet (normal junctions) or at the side of a volume (crossflow junctions). The stream-wise variation of the fluid passage is specified through the volume cross-sectional area, the junction areas, and through use of the smooth or abrupt area change options at the junctions. The smooth or abrupt area change option affects the way in which the flow is modeled, both through the calculation of loss factors at the junction and through the method used to calculate the volume average velocity. (Volume average velocity enters into momentum flux, boiling heat transfer, and wall friction calculations.) The abrupt area change model should be used to model the effect of sudden area changes such as reducers, orifices, or any obstruction in which the flow area variation with length is great enough to cause turbulence and flow separation. Only flow passages having a low wall angle (< 10 degrees, including angle) should be considered smooth. An exception to this rule is the case where the user specifies the kinetic loss factor at a junction and uses the smooth option. This type of modeling should only be attempted for cases where the actual flow area change is modest (less than a factor of two).

The RELAP5-3D© flow model also includes a multi-dimensional formulation option. It is based on the control volume approach, and it allows for Cartesian and cylindrical coordinates.

The hydrodynamic boundaries of a system are modeled using time-dependent volumes and junctions. For example, a reservoir condition would normally be modeled as a constant pressure source of mass and energy (a sink in the case of an outflow boundary). The reservoir is connected to the system through a normal junction, and the inflow velocity is determined from the momentum equation solution. For this type of boundary, some caution is required, since the energy boundary condition is in terms of the thermal energy rather than total energy. Thus, as the velocity increases, the total energy inflow increases owing to the increase in kinetic energy. This effect can be minimized for simulation of a reservoir by making the

cross-sectional area of the time-dependent volume very large compared to the inlet junction area. This policy should be followed for outflow boundaries as well, or else flow reversals may occur.

A second way of specifying a flow boundary is using the time-dependent junction in addition to a time-dependent volume. This type of boundary condition is analogous to a positive displacement pump where the inflow rate is independent of the system pressure. In this case, the cross-sectional area of the time-dependent volume is not used because the velocity is fixed and the time-dependent volume is only used to specify the properties of the inflow. Thus, the total energy of the inflow is specified. When only time-dependent junctions are used as boundary conditions, the system pressure entirely depends on the system mass, and, in the case of all liquid systems, a very stiff system results. An additional fact that should be considered when using a time-dependent junction as a boundary is that pump work is required for system inflow if the system pressure is greater than the time-dependent volume pressure. In particular, any energy dissipation associated with a real pumping process is not simulated. The flow work done against the system pressure is approximated by work terms in the thermal energy equation.

In RELAP5-3D©, any volume that does not have a connecting junction at an inlet or outlet is treated as a closed end. Thus, no special boundary conditions are required to simulate a closed end.

The fluid properties at an outflow boundary are not used unless flow reversal occurs. In this respect, some caution is necessary and is best illustrated by an example. In the modeling of a subatmospheric pressure containment, saturated vapor/gas is often specified for the containment volume condition. This will result in the outflow volume containing pure vapor/gas at low pressure and temperature. If in the course of calculation a flow reversal occurs, even a very minute one (possibly caused by numerical noise), a cascading result occurs. The low-pressure or low-temperature vapor/gas can rush into a volume at higher pressure and rapidly condense. The rapid condensation leads to depressurization of the volume and increased flow. Such a result can be avoided by using air or superheated vapor/gas in the containment volume.

A general guide to modeling hydrodynamic boundary conditions is to simulate the actual process as closely as possible. This guideline should be followed unless initial calculations result in unphysical results because of unanticipated numerical idiosyncrasies.

Only the algebraic sign is needed in the one-dimensional hydrodynamic components to indicate the direction of vector quantities, i.e., the volume and junction velocities. Both the volumes and the junctions have coordinate directions that are specified through input. Each hydrodynamic volume has three coordinate directions, named x, y, and z, and each coordinate direction has an associated inlet and outlet face. The coordinate direction is positive from the inlet to the outlet. The normal, one-dimensional flow is along the x-coordinate. Normal volume connections are to the inlet and outlet faces associated with the x-coordinate. Crossflow connections are to the inlet and outlet faces associated with coordinates orthogonal to the x-coordinate, that is, the y- and z-coordinates.

Which faces of a volume are the inlet or outlet faces depend upon the specifications of the volume orientation. For a positive vertical elevation change, the inlet is at the lowest elevation, whereas for a

negative vertical elevation change, the inlet is at the highest elevation of the volume. For a horizontal volume, whether the inlet is at the left or right depends upon the azimuthal angle. (A zero value implies an orientation with the inlet at the left.) This orientation of a horizontal volume is not important as far as hydrodynamic calculations are concerned but is important if one tries to construct a three-dimensional picture of the flow path. Several possible volume orientations, depending upon the input values for the azimuthal and inclination angles, are illustrated in **Figure 2.1-1**. In the figure, the letter "i" is the inlet and the letter "o" is the outlet.

The junction coordinate direction is established through input of the junction connection code (e.g., Words W1 and W2 of Cards CCC0101 through CCC0109, Section A-7.4 of Appendix A for a single-junction component). The junction connection codes designate a *from* and a *to* component, and the velocity is positive in the direction from the *from* component to the *to* component. The connection codes can be entered in an old or an expanded format. The expanded format is recommended, but the old format is still valid.

The connection code for one-dimensional components has the format CCCXX000F, where CCC is the component number, XX is the volume number, and F is the face number, where zero indicates the old format and nonzero indicates the expanded format. The old format (F = 0) can only specify connections to the faces associated with normal flow, that is, flow along the x-coordinate. In the old format, XX is not a volume number but, instead, XX = 00 specifies the inlet face of the component, and XX = 01 specifies the outlet face of the component. The volume number is only implied. For components specifying single-volumes (currently only a pipe specifies multiple volumes), normal flow (as opposed to crossflow) to either the inlet or outlet face can be specified. For a pipe, however, the old format allows specification of normal flow only to the inlet of the first pipe volume or to the outlet of the last pipe volume. Crossflow meaning connections to faces associated with y- or z-faces cannot be specified with the old format.

The expanded connection code assumes that a volume has six faces, i.e., an inlet and outlet for each of three coordinate directions (see **Figure 2.1-2** and **Figure 2.1-3**). The expanded connection code indicates the volume being connected and through which face it is being connected. In the new format (F nonzero), F is the face number and XX is the volume number. For components specifying single-volumes, XX is 01; but for pipes, XX can vary from 01 for the first pipe volume to the last pipe volume number. The quantity F is 1 and 2 for the inlet and outlet faces, respectively, for the volume's normal or x-coordinate direction. The quantity F is 3 and 4 to indicate inlet and outlet faces, respectively, for the volume's y-coordinate direction. The quantity F is 5 and 6 to indicate inlet and outlet faces, respectively, for the volume's z-coordinate direction. Entering F as 1 or 2 specifies normal connections to a volume; entering F as 3 through 6 specifies a crossflow connection to a volume. In **Figure 2.1-2** and **Figure 2.1-3**, the world (inertial) coordinates are indicated by x_o, y_o, and z_o, whereas the local coordinates for the volume are indicated by x, y, and z. **Figure 2.1-2** is for a horizontal volume. **Figure 2.1-3** is for a vertical volume, and is obtained from the horizontal volume (**Figure 2.1-2**) by a 90° counter-clockwise rotation about the local y-axis.

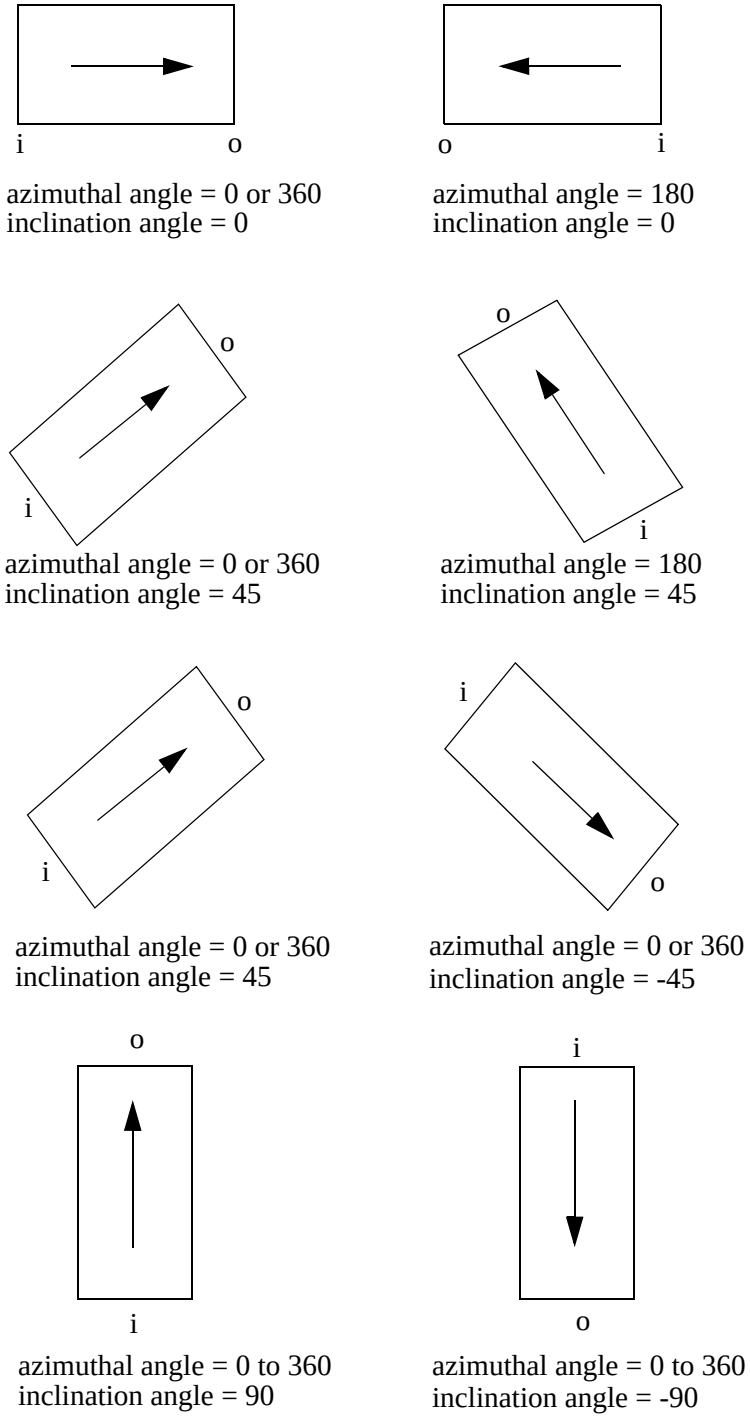


Figure 2.1-1 Possible volume orientation specifications.

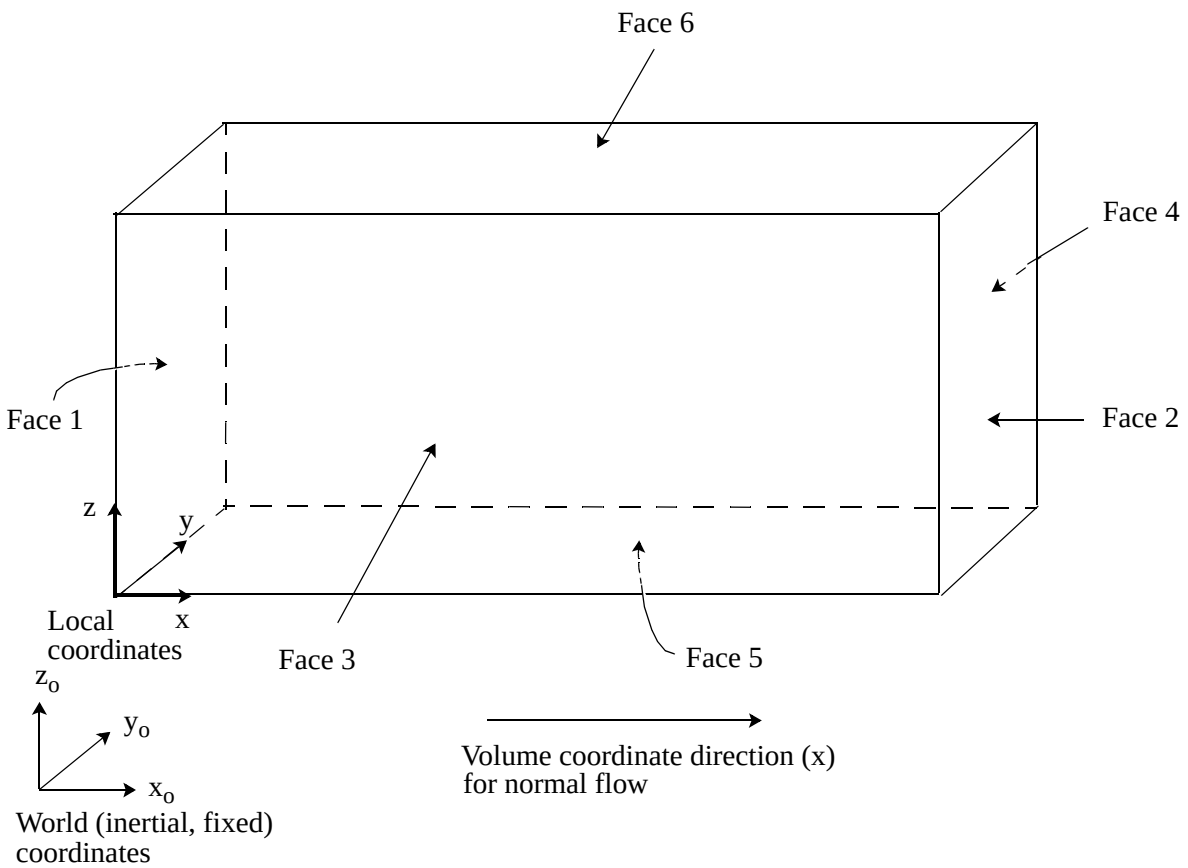


Figure 2.1-2 Horizontal volume schematic showing face numbers for one-dimensional components.

Average volume velocities are computed along each coordinate direction that is active. The x-coordinate is assumed active, and a warning message is issued during input processing if no junctions attach to normal faces. A y- or z-coordinate is active only if a junction attaches to one of the associated faces. The average volume velocity for each coordinate direction involves only junction velocities at the faces associated with that coordinate direction. Thus, a crossflow entering a y-face does not contribute to the computation of the volume velocity in the x-direction. But that crossflow does contribute to the average velocity in the y-direction.

Users of previous versions of RELAP5-3D© will note that the crossflow discussed above is different from older versions. The crossflow capability has been improved, but unfortunately the differing meanings for the term *crossflow* may lead to misunderstanding. The previous use of crossflow implied the following: Flow entered a face orthogonal to the normal flow; crossflows never contributed to any average volume velocity; a limited form of the momentum equation was used; and face numbers 3 through 6 and/or junction flags could specify a crossflow connection. The limited momentum equation ignored momentum flux, wall friction, and gravity terms. Now, crossflow means only that the connection is to a face other than one of the normal faces. Note especially that crossflow does not imply a modified form of the momentum equation. The same momentum equation options are available to both normal flows and crossflows. The

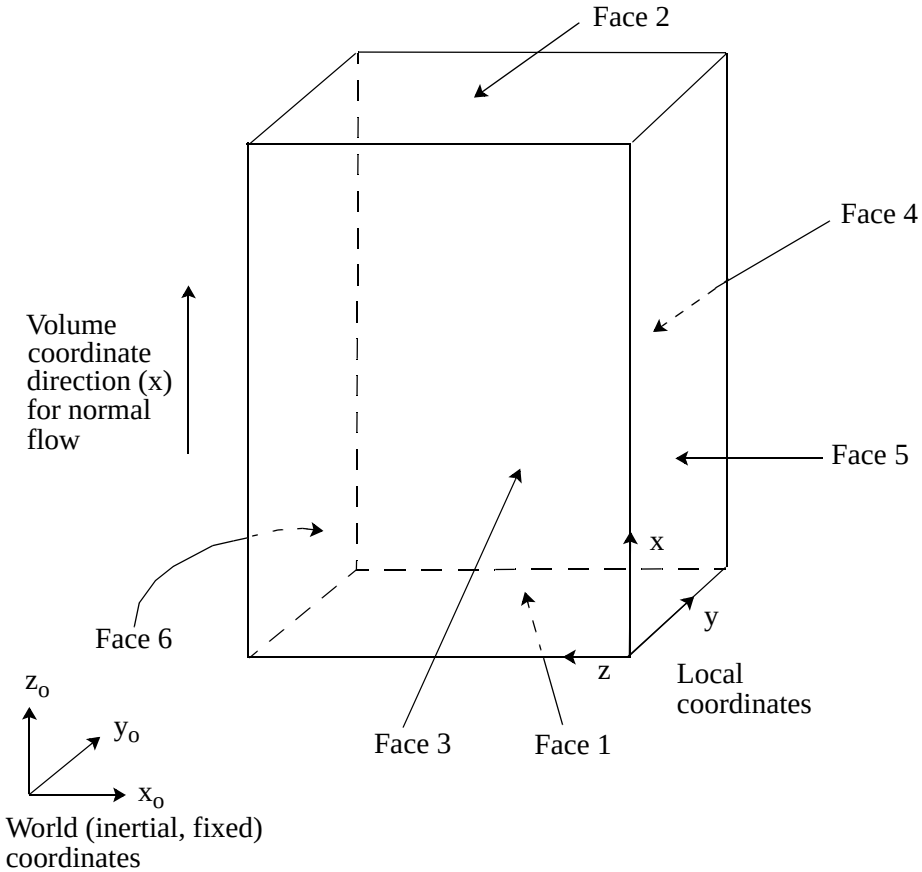


Figure 2.1-3 Vertical volume schematic showing face numbers for one-dimensional components.

standard one-dimensional momentum equations can be applied to both normal and crossflows. Optionally, and only through the use of the momentum flux junctions flags, the momentum flux contribution in the *from* or *to* volume can be ignored for normal and crossflow connections.

There is no difference in the application of the conservation equations to the normal and crossflow types of connections. The only difference is that the term normal is applied to the flow that would occur in a strictly one-dimensional volume; crossflow is an approximation to multidimensional effects consisting of applying the one-dimensional momentum equation to each of the coordinate directions in use. To give some perspective to the approximation, the three-dimensional momentum equation contains nine terms for momentum flux; the momentum in each of the three directions being convected by velocities in the three directions. In the crossflow model, only three momentum flux terms are used--the momentum in each direction convected by velocity in the same direction.

The code input provides junction flags to ignore momentum flux effects in either the *from* volume, the *to* volume, both volumes, or to include momentum effects in both volumes (the default). Intuitively, including momentum effects is more accurate modeling, and momentum effects should be included in junctions attached to the normal faces. In previous versions of the code, a restricted form of the momentum equation was used that omitted momentum flux, wall friction, and gravity terms. One reason was that the

geometric information necessary for computing these terms was not available and average volume velocity terms in the crossflow directions were not computed. The earliest motive for the crossflow model was to treat recirculation flows in the reactor core, and these restrictions were acceptable since velocities were low and there were no elevation changes. The crossflow model was subsequently used for tees since the crossflow model, even with the restrictions, was a better model than previous approaches for tees. The current recommendation is to include the momentum flux terms for crossflows but remove them if computational difficulties involving crossflow junctions are encountered. The crossflow model is currently under developmental assessment. A more definite recommendation is not to have multiple junctions with differing momentum flux options attached to the same coordinate direction. Even though the momentum flux is ignored in one junction, its velocity contributes to the average velocity in that coordinate direction and thus other junctions using momentum flux terms use that average volume velocity.

The current crossflow model requires input information for the y- and z-coordinates similar to that entered for the x-coordinate. Default data for the y and z-coordinates are obtained from the x-coordinate data by assuming the volume is a section of a right circular pipe. Optional input data may be entered when this assumption is not valid.

In major edits and similar input edits, the junction connection code is edited in the new format. Note that the new logic allows branching and merging flow (i.e., multiple junctions at a face) at any volume, including interior pipe volumes. The primary reason for this change is to permit crossflow to all volumes in a pipe. Now it is possible to use pipe volumes to represent axial levels in a vessel and to use multiple pipe components to represent radial or azimuthal dependence. Single-junctions can crosslink any of the pipe volumes at the same axial level.

A simpler method to crosslink volumes is to use the multiple junction component. This component describes one or more junctions, with the limitation that all volumes connected by the junctions must be part of the same hydrodynamic system. Although this component can be considered a collection of single-junctions, its common use is to crosslink adjacent volumes of parallel pipes. Because the junctions linking pipe volumes tend to be similar, N junctions crosslinking N volumes per pipe can be entered with the amount of input comparable to one junction.

The connection code for multi-dimensional components assumes that a volume has six faces, i.e., an inlet and an outlet for each of three coordinate directions (see **Figure 2.1-4** and **Figure 2.1-5**). The multi-dimensional connection code indicates the volume being connected and through which face it is being connected. The quantity F is 1 and 2 for inlet and outlet faces, respectively, for the first coordinate direction (x in Cartesian geometry, r in cylindrical geometry). The quantity F is 3 and 4 to indicate inlet and outlet faces, respectively, for the second coordinate direction (y in Cartesian geometry, θ in cylindrical geometry). The quantity F is 5 and 6 to indicate inlet and outlet directions (z in both Cartesian and cylindrical geometries). In **Figure 2.1-4** and **Figure 2.1-5**, the world (inertial) coordinates are indicated by x_o, y_o, z_o or r_o, θ_o, z_o, whereas the local coordinates for the volume are indicated by x, y, z or r, θ, z.

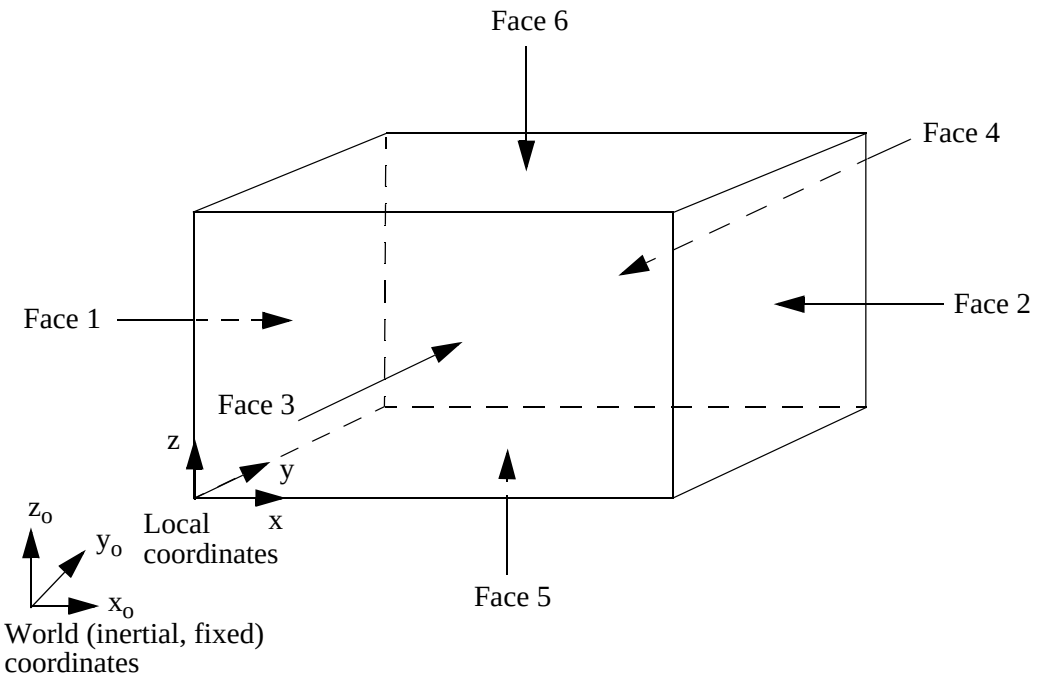


Figure 2.1-4 Cartesian control volume showing face numbers for multi-dimensional components.

.

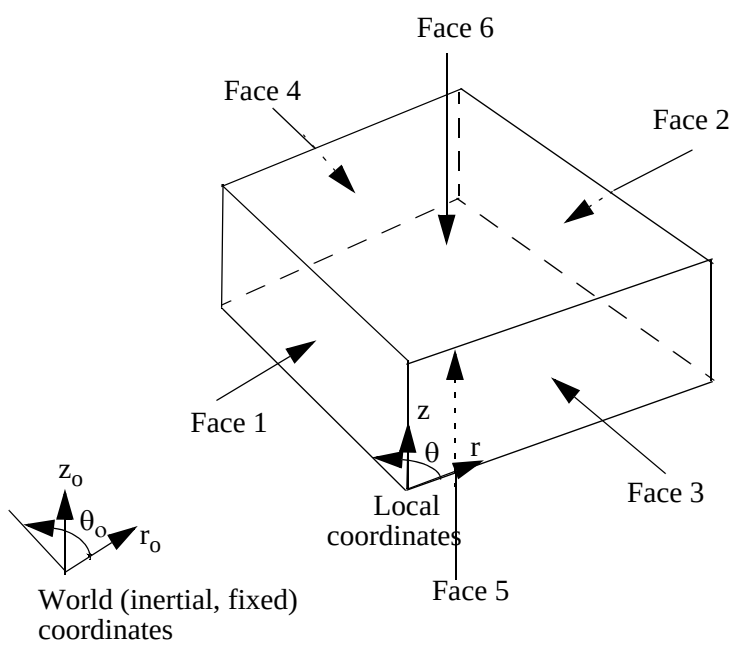


Figure 2.1-5 Cylindrical control volume showing face numbers for multi-dimensional components.

A sketch showing a series of three horizontal one-dimensional volumes connected by two junctions is shown in **Figure 2.1-6** to illustrate some of the possible coordinate orientations that result from

combinations of the connection codes and the volume orientation data. In **Figure 2.1-7**, two possible combinations are illustrated for the connection of two vertical volumes. **Figure 2.1-7**a shows the two volumes unconnected; **Figure 2.1-7**b shows the result when the outlet of Volume 1 is joined to the inlet of Volume 2; and **Figure 2.1-7**c shows the result when the inlet of Volume 1 is connected to the inlet of Volume 2. In particular, note that the geometry can be modified from a straight passage to a manometer configuration by simply reversing the inlet/outlet designator in the junction connection code.

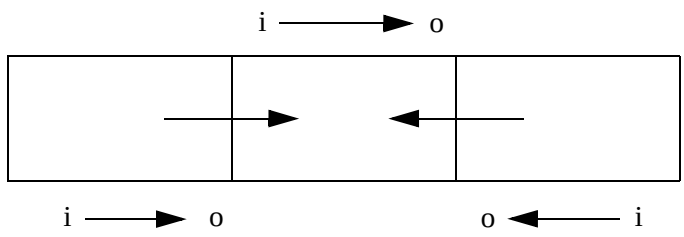


Figure 2.1-6 Sketch of possible coordinate orientation for three volumes and two junctions.

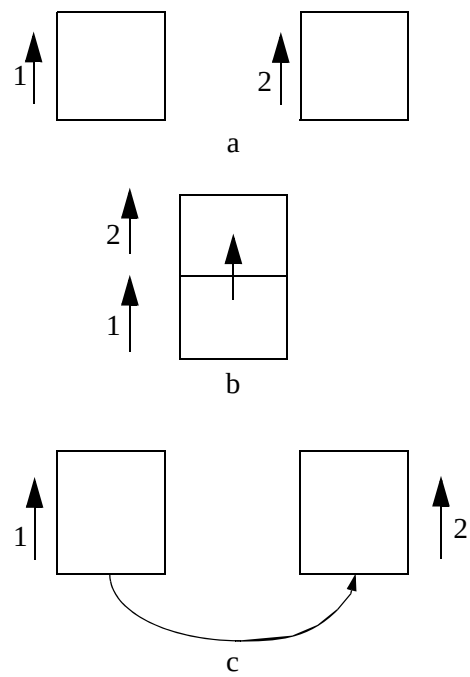


Figure 2.1-7 Sketch of possible vertical volume connections.

When systems of volumes or components are connected in a closed loop, the summation of the volume elevations must close when they are summed according to the junction connection codes and sequence, or an unbalanced gravitational force will result. RELAP5-3D© has an input processing feature that finds all loops or closed systems (which are defined by the input) and checks for elevation closure around each loop. The error criterion is 10^{-4} m. If closure is not obtained, the fail flag is set, and no

transient or steady-state calculations will be made. The elevation checker will print out that elevation closure does not occur at a particular junction that formed a closed loop during input processing. The junction at which closure of the loop occurs depends on the numbering of the components. Input elevation inaccuracies in the fifth significant figure after the decimal point (i.e., on the order of 10^{-5} m) may or may not accumulate to give a difference of 10^{-4} m, depending on the numbering of the components.

The elevation checking with crossflows differs from earlier versions of RELAP5-3D©. The elevation checking starts from the center of a volume with the initial volume and its elevation obtained from input data or defaulted. Using *to* and *from* junction information and elevation change information from the connected volumes, the elevation to the common face of the volumes is computed; then, the elevation of the center of the connected volume is computed. This computing of the elevations by tracking the junctions continues until all junctions have been used. Whenever a volume is reentered, the newly obtained elevation is compared to the previously computed elevation, and an error occurs if they do not match. With the previous crossflow model, the elevation from the center to a face was zero for a crossflow connection. This meant that the same elevation would be obtained regardless of which face the crossflow connection used. The face number is now important, both for elevation checking and in computing elevation effects, momentum flux effects, and friction. We recommend that decks prepared for previous versions of the code have all crossflow connections reviewed for use with the newer crossflow model.

The junctions are printed out in the major edits in the hydrodynamic junction information sections (Section 8.3.2.10 and Section 8.3.2.11). The *from* and *to* volumes are listed for each junction. In addition, the flow regimes for the volumes (floreg) and the junctions (florgj) are also listed using three letters. It is also possible to list the flow regime for the volumes and the junctions in the minor edits and plots, where a number is used. **Table 2.1-1** shows the three-letter code and number used for each flow regime.

Table 2.1-1 Flow regime letters and numbers.

Flow regime	Three-letter code (major edits)	Number (minor edits/plots)
High mixing bubbly	CTB	1
High mixing bubbly/mist transition	CTT	2
High mixing mist	CTM	3
Bubbly	BBY	4
Slug	SLG	5
Annular mist	ANM	6
Mist pre-CHF	MPR	7
Inverted annular	IAN	8

Table 2.1-1 Flow regime letters and numbers. (Continued)

Flow regime	Three-letter code (major edits)	Number (minor edits/plots)
Inverted slug	ISL	9
Mist	MST	10
Mist post-CHF	MPO	11
Horizontal stratified	HST	12
Vertical stratified	VST	13
Mixture level tracking	LEV	14
Jet junction	JET	15
ECC mixer wavy	MWY	16
ECC mixer wavy/annular mist	MWA	17
ECC mixer annular mist	MAM	18
ECC mixer mist	MMS	19
ECC mixer wavy/slug transition	MWS	20
ECC mixer wavy-plug-slug transition	MWP	21
ECC mixer plug	MPL	22
ECC mixer plug-slug transition	MPS	23
ECC mixer slug	MSL	24
ECC mixer plug-bubbly transition	MPB	25
ECC mixer bubbly	MBB	26

Table 2.1-2 Bubbly/slug flow regime numbers for vertical junctions.

Geometry and flow conditions	Correlations used	Numbers (minor edits/plots)
Rod bundles (b = 1) or narrow rectangular channels (b = 2)	EPRI (b = 1) or Griffith (b = 2)	2
High up/down flows in small pipes	EPRI	3
Low up/down countercurrent flows in small pipes	Zuber-Findlay slug	4
Transition regions between 3 and 4	EPRI & Zuber-Findlay slug	5

Table 2.1-2 Bubbly/slug flow regime numbers for vertical junctions. (Continued)

Geometry and flow conditions	Correlations used	Numbers (minor edits/plots)
High up/down flows in intermediate pipes	EPRI	9
Low up/down countercurrent flows in intermediate pipes	Churn-turbulent bubbly	10
Transition regions between 10 and 12	Churn-turbulent bubbly & Kataoka-Ishii	11
Low up/down countercurrent flows in intermediate pipes	Kataoka-Ishii	12
Transition between regions 9 and 10-11-12	EPRI & Churn-turbulent bubbly/Kataoka-Ishii	13
Large pipes	Churn-turbulent bubbly	14
Transition regions between 14 and 16	Churn-turbulent bubbly & Kataoka-Ishii	15
Large pipes	Kataoka-Ishii	16

In the bubbly and slug flow regimes for vertical junctions, it is possible to list an additional flow regime number (iregj) in the minor edits and plots that is associated with a particular geometry/flow and correlation that is used in the interphase drag. If not in bubbly or slug flow and not a vertical junction, the number will be 0. **Table 2.1-2** shows the number used for each regime. In the transition regions 11 and 15, a fraction (between 0 and 1) is added to the number that indicates how far the junction conditions are between churn-turbulent bubbly and Kataoka-Ishii, based on the dimensionless vapor/gas superficial velocity (j_g^+).

The interphase friction model for bundles (i.e., core and steam generator) can be activated with a volume control flag (b=1). The model is based on a correlation from EPRI, as discussed in Volume I of this manual. The interphase friction model for narrow rectangular channels can also be activated with a volume control flag (b=2). The model is based on a correlation by Griffith, as discussed in Volume I of this manual. When in bubbly or slug flow for volume control flag b=1 or b=2, for vertical junctions, the bubbly/slug flow regime number is 2, as indicated in **Table 2.1-2**; otherwise it is 0.

The user should be aware that all plant or experimental facility geometries that are not circular should have an input junction hydraulic diameter to specify the necessary information required for the code calculated interphase friction. For bundles and steam generators, the junction hydraulic diameter should match the volume hydraulic diameter (including grid spacers, which should use the volume hydraulic diameter at the junction). In addition for grid spacers, the volume flow area should be used at the junction and the user-input loss should be multiplied by ratio of squared areas of the volume and the grid spacer.

For area changes, the donor diameter for the normal flow direction is recommended. For orifices, the actual diameter is recommended.

2.2 State Relationships

The default thermodynamic property table for light water vapor and liquid light water has upper and lower limits on pressure and temperature. A pressure-temperature diagram is shown in **Figure 2.2-1**.

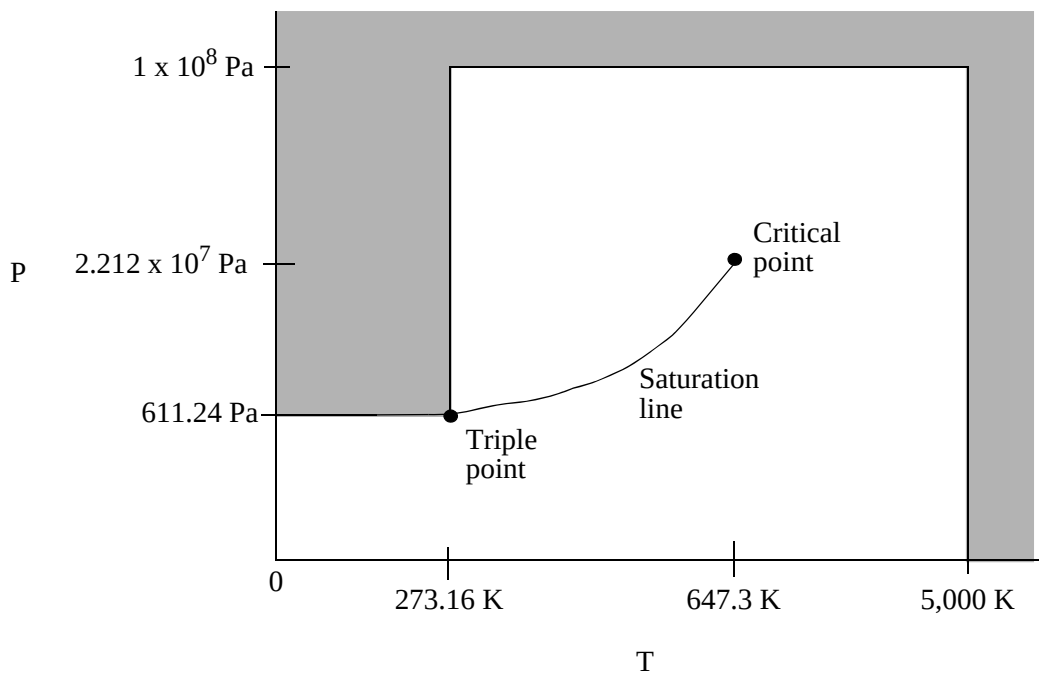


Figure 2.2-1 Pressure-temperature diagram.

If the calculation predicts a pressure and temperature in the shaded region, a thermodynamic property error will result, and the code will cut the time step. If the calculation continues to predict pressure and temperature in the shaded region down to the minimum time step, the calculation will be terminated.

2.3 Process Models

In RELAP5-3D©, process models are used for simulation of processes that involve large spatial gradients or which are sufficiently complex that empirical models are required. The flow processes for an abrupt area change, a choked flow, a branch, reflood, noncondensables, water packer, CCFL, mixture level tracking, and thermal stratification are all simulated using specialized modeling. These particular processes are not peculiar to a component and will be discussed as a group. Some components, such as pumps and separators, also involve special process models; these models will be discussed with the component models. The use of the process models is specified through input, and proper application is the responsibility of the user. Under certain circumstances, we recommend that the user not mix process

models; e.g., we recommend the user not use the choking model at a junction connected to either the side of a volume where the abrupt area change is activated for the junction and more than one junction is connected. The purpose of this section is to advise the user regarding proper application of the process models.

2.3.1 Abrupt Area Change

The abrupt area change option should generally be used in the following situations:

1. Sharp edged area changes.

2. Manifolds and plena connecting parallel flow passages.

3. At break locations.

For the abrupt area model, the junction area (upon which the velocities are based) is the minimum area of the two connecting volumes. The abrupt area change model is discussed in more detail in Section 2.4.1.

In addition to the computed form loss from the abrupt area change model, users have the option of input form loss factors to achieve the desired pressure drop. See Section 2.3.3.3 for discussion for modeling of minor flow paths. It is recommended that the abrupt area change model not be used for minor flow paths where the area change ratio is less than 0.1. For this situation, the smooth area change option and an appropriate loss coefficient is recommended.

The pressure drop calculated by using form losses from the abrupt area change model is a function of junction velocity.

2.3.2 Choked Flow

The choked flow option is specified in the junction flags on the junction geometry card. In general, the choked flow model should be used at all exit junctions of a system. We recommend that the choked flow model be usually used at the choke plane and that the user not model anything past this plane. (Therefore, just use a time-dependent volume downstream of the choke plane.) Internal choking is allowed but may not be desirable under certain conditions. Some applications of RELAP5-3D© require that volumes downstream of the choke plane be modeled with non-time-dependent volumes. For this case, the user should monitor the mass error in the downstream volumes to ensure that the total mass error is not governed by these volumes. This is done by examining the hydrodynamic time step control information in the major edits (see Section 8.3.2); one of the columns labeled LRGST.MASS ERR gives the number of times a volume had the largest mass error. If the mass error in these volumes is large (i.e., the number of times a volume had the largest mass error is high), the user should consider adjusting the size of the volumes. This would involve reducing the size of these volumes, since the mass error is given by the density error times the volume.

The recommended input junction flags when the standard Ransom-Trapp choking model is on (c = 0) are abrupt (a = 1 or 2) and nonhomogeneous (h = 0). (1) With regard to the abrupt area options (a = 1 or 2), these are discussed in Section 2.4.1. The full abrupt area change model (a = 1, code calculated losses) is recommended for sudden (i.e., sharp, blunt) area changes, while the partial abrupt area change model (a = 2, no code calculated losses, user input losses are to be used) is recommended for rounded or beveled area changes. The extra interphase drag term (see Volume IV) in the abrupt area model (a = 1 or 2) helps ensure more homogeneous flow that would be expected through a sudden area change. The smooth area change option (a = 0) is recommended only for when there is no area changes or there are smooth area changes (i.e., venturi). (2) With regard to the nonhomogeneous (h = 0) option, it is generally recommended that h = 0 be used. There may be rare situations where the combined interphase drag is too low, resulting in too much slip and too low mass flow. For this situation, the homogeneous option (h = 1 or 2) is recommended. (3) The user should monitor the calculated results for nonphysical choking. If this occurs, the user should turn choking off (c = 1) at junctions where this occurs.

Guidelines for the standard Ransom-Trapp choking model discharge coefficients (subcooled, two-phase, and superheated) are as follows: For a break nozzle/venturi geometry, discharge coefficients of nearly 1.0 should be used. For an orifice geometry, the discharge coefficients depend on the break configuration and may be somewhat less than 1.0.

The throat $\frac{dA}{dx}$ used in the standard Ransom-Trapp subcooled choking, which is denoted by $\left(\frac{dA}{dx}\right)_t$ in Volume I of this manual, is calculated differently for the normal junction abrupt area option and the normal junction smooth area option.

For the recommended abrupt area change option, the following formula is used:

$$\left(\frac{dA}{dx}\right)_{t,\,abrupt} = \frac{A_K - A_t}{10.0 D_K} \tag{2.3-1}$$

where

A_K = the upstream volume flow area in the coordinate direction of the junction

A_t = the throat or junction area (minimum physical area)

D_K = the upstream volume diameter in the coordinate direction of the junction.

It is recommended the user input the actual physical values for A_K, A_t, and D_K. This formula is empirical, and the data base is limited. It was developed primarily to obtain the proper subcooled discharge at the break for the LOFT-Wyle Blowdown Test WSB03R,[2.3-1] which is one of the developmental assessment

separate-effects test problems. In addition, it has been used successfully in many Semiscale test comparisons for the break flow.[2.3-2]

If the user selects the smooth area change option, the code uses the following formula:

$$\left(\frac{dA}{dx}\right)_{t,\,smooth} = \frac{A_K - A_t}{0.5\Delta x_K} \tag{2.3-2}$$

where

A_K	=	the upstream volume flow area in the coordinate direction of the junction
A_t	=	the throat or junction area (minimum physical area)
Δx_K	=	is the upstream volume length in the coordinate direction of the junction.

The smooth area option is intended to be used for smoothly varying geometries. The length 0.5 Δx_K would be the actual length of the upstream volume (A_K) to the throat (A_t). Since the smooth area change option is not recommended, this formula has had little assessment.

Sometimes, it is observed that the choking junction with the standard Ransom-Trapp choking model oscillates in time between the inlet and outlet junctions of a control volume. This may induce flow oscillations and should be avoided. The situation most often occurs in modeling a break nozzle. The choking plane is normally located in the neighborhood of the throat. The break can be adequately modeled by putting the break junction at the throat and including only the upstream portion of the nozzle. If the entire nozzle is modeled, the choked flow option should be applied only to the junction at the throat.

The internal choking option with the standard Ransom-Trapp choking model must be removed when supersonic flows are anticipated or when its application causes unphysical flow oscillations. Typical cases are propagation of shock waves downstream from a choked junction. Sometimes, it is necessary to remove the choking option at junctions near a known internal choked junction in order to avoid oscillations.

The recommended input junction flags when the optional modified Henry-Fauske choking model is on (c = 2) are abrupt (a = 1 or a = 2) and nonhomogeneous (h = 0). (1) With regard to the abrupt options (a = 1 or a = 2), these are discussed in Section 2.4.1. The full abrupt area change model (a = 1, code calculated losses) is recommended for sudden (i.e., sharp, blunt) area change, while the partial abrupt area change model (a = 2, no code calculated losses, user input losses are to be used) is recommended for rounded or beveled area changes. The smooth area change option (a = 0) is recommended only for when there is no area change or there are smooth area changes (i.e., venturi). (2) With regard to the nonhomogeneous (h = 0) option, it is generally recommended that h = 0 be used. Thus, when the flow is unchoked, slip is allowed between the vapor/gas and liquid. (3) The used should monitor the calculated

results for nonphysical choking. If this occurs, the user should turn choking off (c = 1) at junctions where this occurs.

Guidelines for the optional modified Henry-Fauske choking model discharge coefficient are as follows: For a break nozzle/ventruri geometry, a discharge coefficient of nearly 1.0 should be used. For an orifice geometry, the discharge coefficient depends on the break configuration and may be somewhat less than 1.0.

Guidelines for the optional modified Henry-Fauske choking model thermal nonequilibrium constant are as follows: The default value of 0.14 is recommended by Henry and Fauske for their original choking model for nozzles, orifices, and short tubes, A value of 0.0 causes the model to emulate the homogeneous equilibrium (equal phase temperatures) critical flow model and would be appropriate for long pipes or nozzles. A value of 1000.0 causes the model to emulate the homogeneous frozen (unequal phasic temperatures) critical flow model and would be appropriate for very thin orifice plates.

2.3.3 Branching

A fundamental and vital model needed for simulation of fluid networks is the branched flow path. Two types of branches are common, the tee and the plenum. The tee involves a modest change in flow area from branch to branch and a large change in flow direction, while the plenum may involve a very large change in flow area from branch to branch and little or no change in flow direction. In PWR simulations, a tee model would be used at pressurizer surge line connections, hot leg vessel connections, and cold leg connections to the vessel inlet annulus. A plenum model would be used for modeling upper and lower reactor vessel plenums, steam generator models, and low-angle wyes.

Two special modeling options are available for modeling branched flow paths. These are a crossflow junction model and a flow stratification model, in which the smaller pipe at a tee or plenum may be specified as connected to the top, center, or bottom of a larger connecting pipe. When stratified flow is predicted to exist at such a branch, vapor/gas pullthrough and/or liquid entrainment models are used to predict the void fraction of the branched flow. The use of these models for simulating tees, plenums, and leak paths are discussed in greater detail below.

2.3.3.1 Tees. The simplest tee is the 90-degree tee, in which all branches have the same or comparable diameters. The recommended nodalization for this flow process is illustrated in **Figure 2.3-1**. The small volume at the intersection of the side branch with the main flow path should have a length equal to the pipe diameters. Generally, this length will be shorter than most other hydraulic volumes and will have a relatively small material Courant limit. The code, however, has a time step scheme that permits violation of the material Courant for an isolated volume for the semi-implicit scheme. Thus, this modeling practice may not result in a time step restriction. User experience has shown that if the code runs too slowly and is Courant-limited in the small volume, it is possible to increase the length of the volume to allow faster running without adversely affecting the results.

The Junction J3 is specified as a half normal junction and half crossflow junction. The half of Junction J3 associated with Volume V4 is a normal junction, whereas the half associated with Volume V2

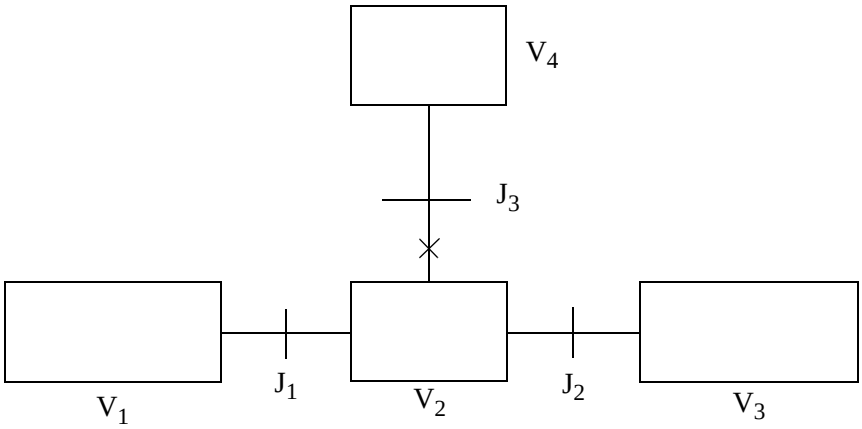


Figure 2.3-1 A 90-degree tee model using a crossflow junction.

is a crossflow junction. The junction specification is made using the junction flag *jefvcahs*, which (for a single-junction) is Word W6(I) of Cards CCC0101 through CCC0109. As noted in previous crossflow junction discussions (see Section 2.1), the same momentum equation options are used available in both normal and crossflow junctions. Both flow types allow ignoring of momentum flux and wall friction terms through the use of volume and junction flags. User experience shows that temperature oscillations may develop in Volume V2. It may be necessary to increase the length of Volume V2 to remove the oscillations. In general, a user loss coefficient will be needed at Junction J3. This coefficient should be determined to obtain the proper pressure drop.

A tee can also be modeled using the branch component, as illustrated in **Figure 2.3-2**. This approach has the advantage that fewer volumes are used. Disadvantages are that the calculated result may be altered, depending on whether Junction J_2 is connected to Volume V_1 or V_2, and that the flow division has less resolution at the tee in the presence of sharp density gradients. In cases where the Volumes V_1 and V_3 are nearly parallel, the model illustrated in **Figure 2.3-2** may be a more accurate representation of the physical process (such as for a wye).

2.3.3.2 Branch. The branch model approximates the flow process that occurs at merging or dividing flows, such as at wyes and plenums. This model does not include momentum transfer caused by mixing and thus is not suited for high-velocity merging flows. A special component, the JETMIXER, is provided for modeling the mixing of high-velocity, parallel streams. Application of this model is discussed in Section 2.4.9.

A branch component consists of one system volume and zero to nine junctions. The limit of nine junctions is due to a card numbering constraint. Junctions from other components, such as single-junctions, pumps, other branches, or even time-dependent junction components, may be connected to the branch component. The results are identical whether junctions are attached to the branch volume as part of the branch component or as part of other components. Use of junctions connected to the branch but defined in

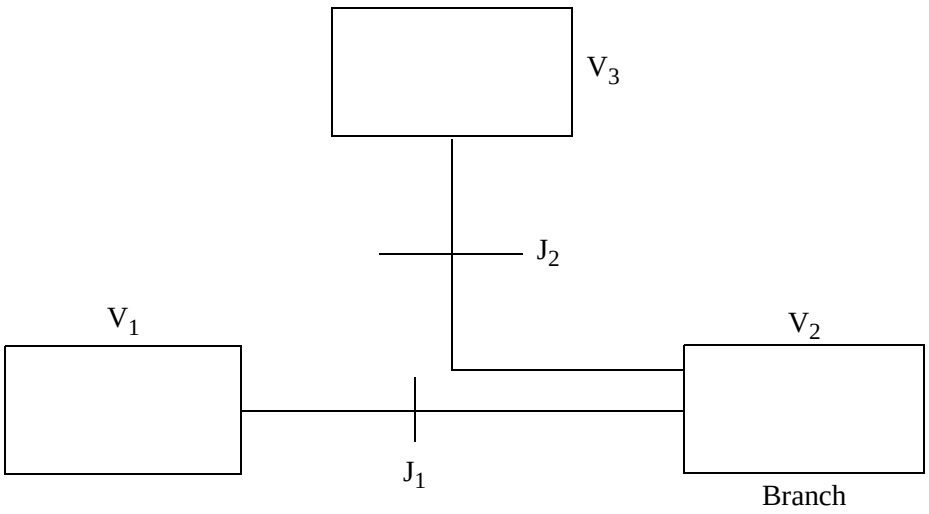


Figure 2.3-2 Tee model using a branch component.

other components is required in the case of pump and valve components. Any of these may also be used to attach more than the maximum of nine junctions that can be described in the branch component input.

A typical one-dimensional branch is illustrated in **Figure 2.3-3**. The figure is only one example and implies merging flow. Additional junctions could be attached to both ends, and any of the volume and junction coordinate directions could be changed. The actual flows may be in any direction; thus, flow out of Volume V_3 through Junction J_1 and into Volume V_3 through Junction J_2 is permitted.

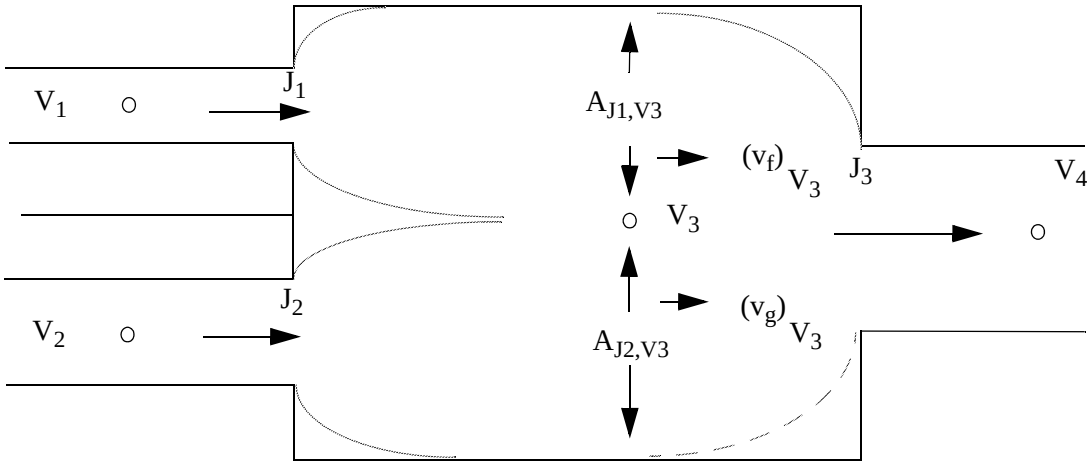


Figure 2.3-3 Typical branching junctions.

The volume velocities are calculated by a method that averages the phasic mass flows over the volume cell inlets and outlets. The volume velocities of Volume V_3 are used to evaluate the momentum flux terms for all junctions connected to Volume V_3. The losses associated with these junctions are calculated using a stream tube formulation based on the assumption that the fraction of volume flow area associated with a junction stream tube is the same as the volumetric flow fraction for the junction within the respective volume. Also, using the junction flow area, the adjacent volume flow areas, and the branch volume stream tube flow area, the stream tube formulation of the momentum equation is applied at each junction. However, if the smooth area change is specified, large changes in flow can lead to nonphysical results. Therefore, it is normally recommended that the abrupt area change option (a = 1 or a = 2) be used at branches.

Plenums are modeled using the branch component. Typical LWR applications of a plenum are the upper and lower reactor vessel regions, steam generator plenums, and steam domes. The use of a branch to model a plenum having four parallel connections is illustrated in **Figure 2.3-4**. The flows in such a configuration can be either inflows or outflows. The junctions connecting the separate flow paths to the plenum are ordinary junctions with the abrupt area change option recommended. It is possible to use crossflow junctions at a branch for some or all of the connections.

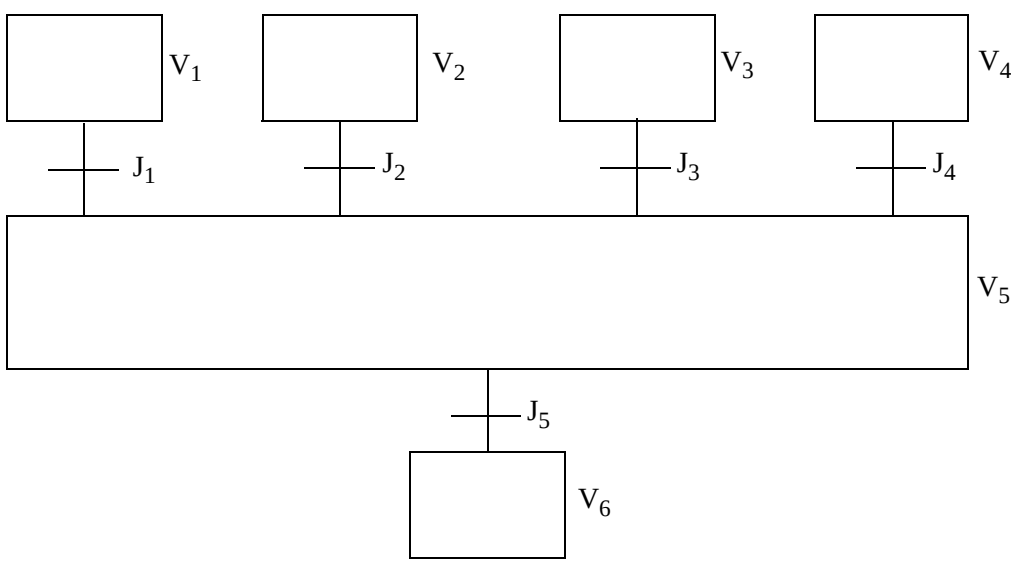


Figure 2.3-4 Plenum model using a branch.

A wye is modeled, as illustrated in **Figure 2.3-3**, using the branch component. The flow can either merge or divide. Either the smooth or the abrupt area change option may be used. If the smooth area change is specified, large changes in flow can lead to nonphysical results. Therefore, it is normally recommended that the abrupt area change option be used at wyes.

2.3.3.3 Leak Paths. An application, that may or may not involve branching but which is frequently a source of problems, is the modeling of small leak paths. These may be high-resistance paths or may involve extreme variations in flow area. The approximation of the momentum flux terms for such flow paths is highly uncertain and can lead to large forces, resulting in numerical oscillations. Modeling of small leak paths was one of the primary motivations for developing the crossflow connections. As needed, the momentum flux and wall friction can be omitted, and the flow resistance could instead be computed from a user-specified kinetic loss factor.

It is recommended that the momentum flux be omitted in the cross direction (perpendicular to the main upward flow direction) if either of the following two modeling approaches are used:

- 1-D components (i.e., pipes, etc.) with cross flow junctions
- MULTID component, 1-D option (1-D momentum equations per Card CCC0001, Word 7 = 1 or 3).

In applying the crossflow junction to leak path models, the actual area of the leak path is used as the junction area. A kinetic loss factor is input, based on the fluid junction area velocity for the forward and reverse loss factors. The forward and reverse loss factors should be equal unless there is a physical reason why they should be different. In particular, a very large forward and small reverse loss factor should not be used to simulate a check valve. This approach can cause code failure. A typical leak path model between vertical volumes is illustrated in **Figure 2.3-5**.

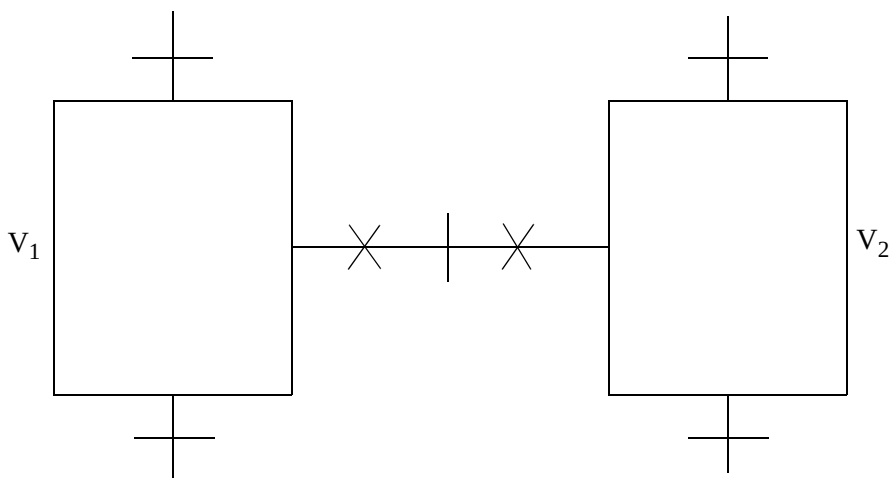


Figure 2.3-5 Leak path model using the crossflow junction.

Minor flow paths having extreme area variations or flow splits, in which the minor flow is a small fraction of the main flow (< 0.1), can also be modeled using the standard junction by the following special procedures. The smooth area change option is used for the junction (the *jefvcahs* flag with a = 0), and the junction area is allowed to default (the minimum area of the adjoining volume areas). It may be necessary

for the user to input a more reasonable flow area if the default area is too large. With this specification, it is necessary to enter user-input form loss coefficients normalized to the default area in order to give the proper flow rate and pressure drop relationship. The loss factor to be input can be estimated using the following equation:

$$K = 2\Delta P A^2 \frac{\rho}{\dot{m}^2} \tag{2.3-3}$$

where

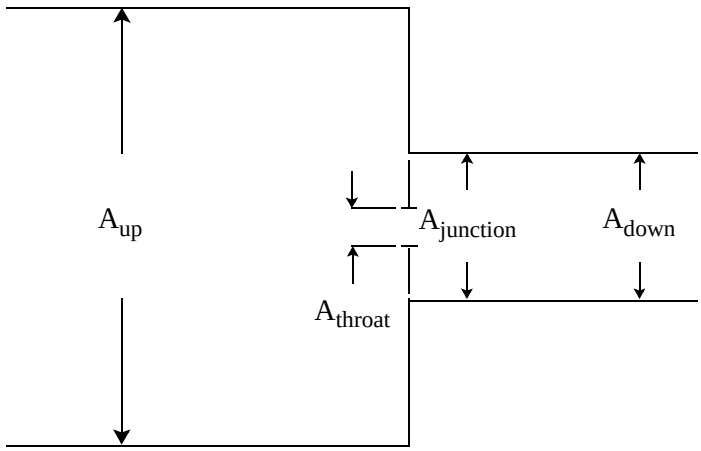
K	=	loss factor
ΔP	=	nominal pressure drop (Pa)
A	=	junction area (m^2)
ρ	=	fluid density (kg/m^3)
$\dot{m}$	=	nominal mass flow rate (kg/s).

The value computed for K in this way may be very large because the default area is much larger than the actual flow area. Also, critical flow would not be detected with this approach. Both the forward and reverse loss coefficients should be equal unless there is a reason why they are physically different. In this case, Equation (2.3-3) should be used to calculate the effective loss factor for both the forward and reverse flow conditions (i.e., assume ΔP and $\dot{m}$ also correspond to the reverse flow case). The geometric relationship between the actual situation and the model is illustrated schematically in **Figure 2.3-6**.

In the case of minor flow paths that connect at branches having large main flows, a similar approach can be used. In this case, let the junction area default to the minimum of the adjoining volumes (presumably the area of the minor flow path) and use the smooth option (*jefvcahs* with a = 0). The determination of the loss factor may require some experimentation because of the possible large momentum flux effect, which is ignored in the derivation of Equation (2.3-3). If one of the volumes is quite large compared to the other, a modified Bernoulli equation can be used in which the overall loss factor defined by Equation (2.3-3) can be replaced by K + 1. [In other words, the user-input loss factor is computed by substituting K + 1 for K in Equation (2.3-3).]

All of the development herein assumes that known pressure drop flow relations exist for the single-phase case and that compressibility effects are small. If such is not the case, then the effective loss factor values must be determined experimentally by running the code for a series of cases. Some experimentation may be required, since the actual momentum flux calculation is complicated by several factors and may differ slightly from the simple Bernoulli form.

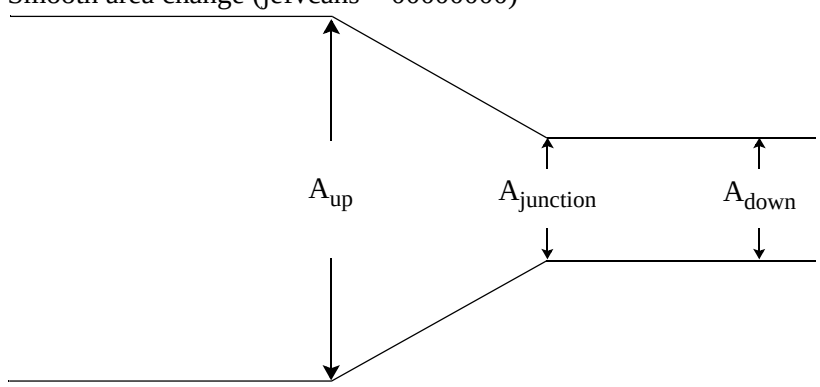
Abrupt area change (jefvcahs = 00000100)



Throat ratio = $\frac{A_{throat}}{A_{junction}}$

Physical situation with a loss factor K_{actual}

Smooth area change (jefvcahs = 00000000)



$$K_{effective} = K_{actual} \left(\frac{A_{junction}}{A_{throat}}\right)^2$$

Equivalent model with effective loss factor for the same pressure drop-flow relation

Figure 2.3-6 High-resistance flow path model.

Another problem relative to a minor leak path can occur when an incorrect flow rate through an orifice for a given ΔP and loss coefficient K is calculated. As noted previously, this problem can be avoided if the user inputs reasonable values for the flow area and the loss coefficient K, rather than allowing the flow area to default and using a very large K.

2.3.4 Noncondensables

The noncondensable model has the ability to be applied at every hydrodynamic volume in a system model. While in operation, the model affects interface mass and heat transfer, wall heat transfer, and the output of several variables that may cause discontinuities in plotted output. The purpose of this discussion is to clarify the operation of the noncondensable model and its affect on the calculated results and to give guidance for its use in system calculations.

In order to properly understand the operation of the noncondensable model, the fundamental assumptions used in the model need to be discussed. First, the vapor/noncondensable mixture is assumed to be in thermal equilibrium. Second, the total pressure is the sum of the partial pressures of the vapor and the noncondensable. Third, the vapor/gas specific internal energy is the mass weighted sum of the vapor specific internal energy and the noncondensable specific internal energy. Fourth, the liquid phase nonequilibrium properties are calculated in the same manner as for the case without noncondensables, i.e., these liquid properties are based on the total pressure and the liquid specific internal energy. Fifth, the saturation properties of the liquid and vapor/gas used in the interface temperature derivatives are assumed to be a function of the partial pressure of the vapor/gas. Sixth, the velocity of the noncondensable gas is assumed to be equal to the velocity of the vapor.

One of the effects of these assumptions is to force the phasic temperatures and the saturation temperature based on the partial pressure of vapor to the same value. This causes a reduced driving potential for the interface mass and heat transfer models. The interface heat transfer coefficients are reduced in the presence of noncondensables. Consequently, low interfacial heat transfer regimes, such as the vertical stratification flow regime, may give heat transfer coefficients that are too low for stable calculations (as evidenced by oscillatory behavior). When this occurs, the vertical stratification model should be turned off on a volume basis. The highest probability for this occurrence is under very low flow conditions.

A second problem may occur when noncondensables first appear in a system volume. At times, again depending on the convection of noncondensable into the volume, the noncondensable iteration may fail or thermodynamic property errors occur at the minimum time step. This problem can usually be overcome by reducing the size of the minimum time step. If this procedure fails, the convection rate or the concentration of the noncondensable convection must be changed. This may be accomplished by modifying the boundary conditions or by renodalizing the problem areas with acceptable thermodynamic conditions.

Last, the output from the code may contain discontinuities as noncondensables appear or disappear. The variables that will have these discontinuities are the partial pressure of vapor, phasic temperatures, saturation temperature, vapor/gas specific internal energy, and noncondensable quality. The partial pressure of vapor is set to the system pressure if noncondensables are not present, to 1.0 Pa if the volume state is pure noncondensable, or to the calculated value otherwise. When only pure noncondensable is present in a volume, all temperatures are set to equal values and are a function of the gas energy. As a fluid is injected into the volume, the temperatures, the partial pressure of vapor, and possibly the vapor specific internal energy, will abruptly change to new values based on the calculated thermodynamic conditions.

Additionally, the liquid and saturation temperature may appear at the fluid triple point value if the partial pressure of the vapor is calculated to be lower than the minimum thermodynamic property table value.

As an example, a checkout problem used for development[2.3-3] consisted of 322 K liquid water being injected into 436 K helium. The liquid temperature and saturation temperature both changed from 436 to 273 K in one time step as the volume changed from a pure noncondensable state to a vapor/noncondensable mixture state. As more water was injected, the liquid temperature transitioned to the correct value.

Selecting noncondensable input consists of specifying type and mass fraction of species on Cards 110 and 115 and by selecting options 4, 5, 6, or 8 on the volume initial condition cards. The mass fractions on Card 115 are the default values. For all volumes except accumulator volumes (which only has nitrogen), the noncondensable species mass fractions can also be entered in the hydrodynamic data. Option 4, which consists of pressure, temperature, and static quality at equilibrium conditions (100% relative humidity), is the easiest to use. A restriction on the temperature is that it has to be less than the saturation temperature as a function of pressure and less than the critical temperature. The static quality used in option 4 (equilibrium) is given by $\frac{M_g}{M_g + M_f}$, where $M_g = M_s + M_n$. Little experience has been obtained in using option 5 (equilibrium), and it has not been checked out. Option 6 (nonequilibrium or equilibrium) is generally used to renode system models from Pygmalion input decks.[2.3-4] See Volume I of this manual for the equations and variables used. If option 6 is selected, but noncondensable quality is 0.0, the coding uses option 0. Option 8 (nonequilibrium or equilibrium) allows the phasic temperatures to be input instead of the phasic specific internal energies used in option 6. If option 8 is selected, but noncondensable quality is 0.0, the coding used is similar to option 0 except that the input phasic temperatures are used instead of the phasic specific internal energies.

The capability for initializing and performing transients with pure noncondensables (0% relative humidity) has been implemented. Input options 4, 6, and 8 have provisions for initializing a system volume to a pure noncondensable state. This is accomplished in option 4 by using the equilibrium quality variable as a flag. By setting this quantity to 0, the logic specifies an ideal noncondensable gas equation of state. The variable is reset to 1.0 for transient calculations. To use option 6 for initializing a pure noncondensable, both void fraction and noncondensable quality must be set to 1.0 and the vapor/gas specific internal energy must be set to a value that gives the desired gas temperature. To use option 8 for initializing a pure noncondensable, both void fraction and noncondensable quality must be set to 1.0 and the vapor/gas temperature must be set to a value that gives the desired gas specific internal energy.

The code will not allow a noncondensable to exist with pure liquid and no vapor. The code will add a little bit of vapor (keeps vapor quality $X_s = \frac{M_s}{M_s + M_n} \geq 10^{-8}$) when pure liquid and noncondensable are present. Thus, we recommend that users input some vapor when noncondensables and liquid are present.

Option 4 (equilibrium) is normally used because it is much easier to use. Only a saturated noncondensable state (100% relative humidity) or a dry non-condensable state (0% relative humidity) are obtained by this option. Improvement of input conveniences for initial noncondensable states is under consideration. The users employ the following method (for the 100% relative humidity case) with this option. For a given total pressure (P), temperature (T), and void fraction (α_g), the static quality (X) in equilibrium must be determined. The static quality is given by $X = \frac{\alpha_g \rho_g}{\alpha_g \rho_g + \alpha_f \rho_f}$, where $\alpha_f = 1 - \alpha_g$. The mixture vapor/gas density (ρ_g) is given by $\rho_g = \rho_n + \rho_s$, there ρ_n is the noncondensable density and ρ_s is the vapor density. The vapor density is calculated from the thermodynamic tables as $\rho_s = \rho_s^s(T)$, where ρ_s^s is the vapor density at saturation conditions for T. The vapor partial pressure (P_s) is obtained from the thermodynamic tables as $P_s = P^s(T)$, where P^s is the saturation pressure function. The noncondensable gas partial pressure (P_n) is obtained from $P_n = P - P_s$. The noncondensable gas density (ρ_n) is obtained from $\rho_n = \frac{P_n}{R_n T}$, where R_n is the noncondensable gas constant (See Volume I).

Option 4, with static quality = 1.0, is recommended for containment volumes.

Option 6 (nonequilibrium or equilibrium) is used to set the relative humidity to less than or equal to 100%. The users employ the following method for option 6 for a nonequilibrium vapor-gas-liquid mixture with a given total pressure (P), vapor/gas temperature (T_g), liquid temperature (T_f), void fraction (α_g), and noncondensable gas quality $\left(X_n = \frac{M_n}{M_n + M_s}\right)$: First calculate the vapor quality ($X_s = 1 - X_n$). The partial pressure of vapor (P_s) is approximated from the relation $P_s = X_s \left(\frac{M_s + M_n}{M_s + \frac{MW_s}{MW_n} M_n} \right) P$, which is exact only if both vapor and noncondensable gas are ideal gases and obey Dalton's mixture law. The variable MW_s is the molecular weight of vapor and MW_n is the molecular weight of the noncondensable gas. The thermodynamic property tables are used to obtain the vapor specific internal energy (U_s) from the known values of P_s and T_g. The noncondensable gas specific internal energy (U_n) is calculated from equations in Volume I of the code manual. The mixture vapor/gas specific internal energy (U_g) is then calculated from $U_g = X_n U_n + X_s U_s$. Finally, the liquid specific internal energy (U_f) is determined using the thermodynamic property tables from the known values of P and T_f.

The users employ the following method for option 6 for a nonequilibrium vapor-gas mixture (no liquid) with a given total pressure (P), vapor/gas temperature (T_g), and a fractional relative humidity (ϕ): The noncondensable quality X_n is first determined. The humidity ratio (or specific humidity) (ω) is defined

by $\omega = \frac{M_s}{M_n}$. Assuming both vapor and noncondensable gas are ideal gases, the humidity ratio can be expressed as $\omega = \frac{MW_s}{MW_n} \cdot \frac{P_s}{P_n}$, where $P_n = P - P_s$, MW_s is the vapor molecular weight, and MW_n is the noncondensable molecular weight (note: for steam and air, $\frac{MW_s}{MW_n} = 0.622$). The fractional relative humidity is given by $\phi = \frac{P_s}{P_g}$, where P_g is obtained from the thermodynamic property tables as $P_g = P^s(T_g)$ and P_s is the partial pressure of vapor. Using $P_s = \phi P_g$, then ω can be determined. Using $X_n = \frac{M_n}{M_n + M_s} = \frac{1}{\omega + 1}$, then X_n can be determined. The vapor quality X_s is given by $X_s = 1 - X_n$. The thermodynamic property tables are then used to get the vapor specific internal energy (U_s) from the known values of P_s and T_g. The noncondensable gas specific internal energy (U_n) is calculated from equations in Volume I of the code manual. The mixture vapor/gas specific internal energy (U_g) is the calculated from $U_g = X_n U_n + X_s U_s$. The liquid specific internal energy (U_f) is determined using the thermodynamic tables from $U_f = U_f^s(P_s)$ where U_f^s is the liquid specific internal energy at saturation conditions for pressure P_s. The void fraction (α_g) is input a 1.0.

2.3.5 Water Packing

The volume control flag p is used to activate the water packing mitigation scheme. The scheme is invoked if the detection criteria are met.

The number of partial time step repeats is shown in the hydrodynamic volume statistics block (time step control information) in the major edit. Both the number of repeats since the last major edit and for the whole calculation are shown.

2.3.6 Countercurrent Flow Limitation Model

The countercurrent flow limitation (CCFL) model (discussed in detail in Section 3 of Volume I) is controlled by the junction control flag. The CCFL flag (f) can be used with a single-junction, pipe, annulus, branch, valve, pump, and multiple junction. It cannot be used with a time-dependent junction, separator, jet mixer, ECC mixer, turbine, or accumulator. Setting f = 1 will activate the CCFL model if all other conditions are met, and setting f = 0 will not activate the model. The other conditions are as follows:

1. The orientation of both the connecting volumes cannot be horizontal (i.e., the elevation angle must be greater than or equal to 45 degrees).

2. Both vapor/gas and liquid phases must be present.

3. Countercurrent flow must exist, with liquid flowing down and vapor/gas flowing up.

As with the choking model, we recommend that if a junction is designated CCFL (f = 1), then an adjacent junction should not be designated CCFL (f = 0). It is anticipated that this flag will find use in activating the CCFL model in such internal structures as the upper core tie plate, downcomer annulus, steam generator tube support plates, and entrance to the tube sheet in the steam generator inlet plenum.

Junction data cards can be used to input four quantities (junction hydraulic diameter, correlation form, vapor/gas intercept, and slope). For these CCFL junction data cards, all four quantities must be entered (must have five quantities for pipe and multiple junction). If no card is entered but the CCFL flag f is set to 1, then default values of the four quantities will be used. Presently, the default values are

$$D_j = 2\left(\frac{A_j}{\pi}\right)^{1/2}$$

$$\beta = 0$$

$$c = 1$$

$$m = 1.$$

This corresponds to a Wallis CCFL correlation with a vapor/gas intercept of 1 and a slope of 1, which, according to Wallis,[2.3-5] is the case for turbulent flow (m = 1) and when end effects are minimized (c = 1).

The input was made general so that the user can input CCFL correlations for the particular geometry of interest. Wallis,[2.3-5] Bankoff et al.,[2.3-6] and Tien et al.[2.3-7] discuss numerous examples, and these, along with other references, should be consulted in order to justify the use of a particular correlation for a given geometry. Wallis suggests m = 1 for a turbulent flow, c = 0.725 for tubes with sharp-edged flanges, and c = 0.88 to 1.0 for tubes when end effects are minimized. Bankoff suggests $\beta = \tanh(\gamma k_c D_j)$, where the critical wave number $k_c = \frac{2\pi}{t_p}$ corresponds to the maximum wavelength that can be sustained on a interface of length t_p (the plate thickness), and γ is the perforation ratio (fraction of plate area occupied by holes). Bankoff suggests m = 1 and c of the form

$$c = 1.07 + 4.33 \times 10^{-3} D^* \qquad D^* < 200 \tag{2.3-4}$$

$$= 2 \qquad D^* \geq 200$$

where D^* is a Bond number defined as

$$D^* = n\pi D\left[\frac{g(\rho_f - \rho_g)}{\sigma}\right]^{1/2}, \tag{2.3-5}$$

and n is the number of holes. Tien uses the Kutateladze form ($\beta = 1$), but the form of c allows the Wallis form also to be invoked for small diameters. He suggests c of the form

$$c = c_7 \, [\tanh c_8 \, (D^*)^{1/4}],$$

where D^* is a Bond number defined differently from Equation (2.3-5) as

$$D^* = D\left[\frac{g(\rho_f - \rho_g)}{\sigma}\right]^{\frac{1}{2}} . \qquad (2.3\text{-}6)$$

The values of m, c_7, and c_8 Tien found for four different conditions are provided in **Table 2.3-1**.

Table 2.3-1 Values of m, c_7, and c_8 for Tien's CCFL correlation form.

Tests	m	c_7	c_8
Nozzle air supply with tapered inlet	0.8	2.1	0.9
Nozzle air supply with sharp edge inlet	0.8	2.1	0.8
Indirect air supply with tapered inlet and sharp edge output	0.65	1.79	0.9
Indirect air supply with sharp edge inlet and tapered output	0.65	1.79	0.8

With regard to guidelines for plant-specific geometry (i.e., tie plates, support plates, etc.), flooding data obtained in measurements from the plant geometry should be used to generate an appropriate CCFL model that can be input with CCFL junction data cards.

Wallis,[2.3-5] Bankoff,[2.3-6] and Tien[2.3-7] discuss the effects of viscosity, surface tension, and subcooling on the correlations. At the present time, these effects have not been directly incorporated into the form of the CCFL correlation used in RELAP5-3D©. We anticipate that these, particularly the subcooling effects, will be addressed in future modifications to the code.

2.3.7 Mixture Level Tracking Model

The volume control flag 1 in t1pvbfe is used to activate the level tracking model as described in Volume I. If more than one junction is connected to the top of the volume or if more than one junction is connected to the bottom of the volume, the mixture level model is not used and is turned off.

If the volume control flag is set, the major edit will print out variables associated with the mixture level in the hydrodynamic volume. The variables are voidla, the void fraction above the level; voidlb, the

void fraction below the level; vollev, the location of the level within the volume; vlev, the velocity of the level movement; and height, the height of the two-phase level above the bottom of the level stack. For each system and each level stack in each system, the parameters LEVVDA (void fraction above level), LEVVDB (void fraction below level), LEVVEL (velocity of level movement), LEVHGT (height of two-phase level above bottom of stack), and LEVVNM (volume number of volume containing two-phase level) can also be written to the restart-plot file if a 2080XXXX card is used and can be used in minor edit requests.

2.3.8 Thermal Stratification Model

The volume control flag t in t1pvbfe is used to turn on the subcell resolution scheme in the model. The model is invoked if the detection criteria are satisfied. The model is intended for one-dimensional components only.

The thermal stratification model should be used to improve the accuracy of calculations where there is a warm liquid layer appearing above a cold liquid in a vertical stack of cells. A complete description of the model is presented in Section 3 of Volume I.

2.3.9 Energy Conservation at an Abrupt Change

The junction control flag e in jefvcahs is used to activate the modification to the energy flux term described in Volume I. This model is recommended for break junctions that connect to containment volumes that are modeled using regular volumes (not time-dependent volumes).

2.3.10 Jet Junction Model

A single-junction (sngljun) component may be flagged as a jet junction. A j value of 1 on the junction flags (jefvcahs) labels the junction as a jet junction. Jet junctions are used where subcooled liquid is injected into the bottom of a stratified pool. The flag activates logic to increase the condensation rate on the surface of the pool. Condensation is only enhanced in the volume above the jet when the volume is in vertical stratification or a level exists from the mixture level tracking model. Normally the letters VST appear in the major edits and 13 appears for the flow regime number (floreg) in the minor edits and on the restart-plot file when vertical stratification exists in a volume and the letters LEV appear on the major edits and 14 appears for the flow regime number (floreg) in the minor edits and on the restart-plot file when a level exists in a volume from the level model. However, when the liquid interfacial coefficient has been altered by the jet junction model and the volume is either in vertical stratification or a level exists from the mixture level tracking model, the letters JET appear in the major edits and 15 appears for the flow regime number (floreg) in the minor edits and on the restart-plot files.

2.3.11 Variable Volume Model

A single-volume (snglvol) component has an option to use the variable volume model. The model has several input options and capabilities. The volume of the computational volume of a single volume

component may be computed from several input options: (1) the volume of the computational volume of a single volume component may be computed from the output of a control variable where the output of the control variable is interpreted as the normalized volume of the computational volume; (2) the volume of the computational volume of a single volume component may be computed from the output of a control variable where the output of the control variable is interpreted as a normalized stem position and is used as the input to a general table of normalized volume versus normalized stem position; (3) the volume of the computational volume of a single volume component may be computed from the output of a general table of normalized volume versus time; and (4) the volume of the computational volume of a single volume component may be computed from a general table where the table is interpreted as normalized volume versus time after a trip is activated. The model has several other features. It checks the volume of a computational volume at each time step and stops the code whenever the volume of a computational volume is less than 1.0×10^{-6} m^3. If the volume of a computational volume is less that 1.0×10^{-6} m^3, the code terminates the current advancement; backs up to the end of the previous advancement; writes a major edit, a minor edit, and a restart-plot record; and stops. The backup is needed because the volume of the computational volumes is computed at the beginning of the time step advancement for each system and some systems may have been successfully advanced before the current system terminates the code execution. Finally, the user input is checked for consistency during the initialization phase of the computation, and the code is stopped if the user selects inconsistent input for the variable volume model.

2.3.12 References

2.3-1. V. H. Ransom et al., *RELAP5/MOD2 Code Manual, Volume 3: Development Assessment Problems*, EGG-TFM-7952, Idaho National Engineering Laboratory, December 1987, pp. 14-17.

2.3-2. *Ibid.*, pp. 61-63.

2.3-3. K. E. Carlson, *Improvements to the RELAP5/MOD3 Noncondensable Model*, EGG-EAST-8879, Idaho National Engineering Laboratory, January 1990.

2.3-4. J. E. Fisher, *Pygmalion Users Manual*, INEEL-CON-01-01145, Idaho National Engineering and Environmental Laboratory, September 2001.

2.3-5. G. B. Wallis, *One-dimensional Two-phase Flow*, New York: McGraw-Hill, 1969, pp. 336-341.

2.3-6. S. G. Bankoff, R. R. Tankin, M. C. Yuen, and C. L. Hsieh, “Countercurrent Flow of Air/Water and Steam/Water Through a Horizontal Perforated Plate,” *International Journal of Heat and Mass Transfer, 24*, 1981, pp. 1381-1385.

2.3-7. C. L. Tien, K. S. Chung, and C. P. Lin, *Flooding in Two-Phase Countercurrent Flows*, EPRI NP-1283, December 1979.

2.4 Hydrodynamic Components

The basic two-fluid model is applied uniformly to all volumes and junctions. Thus, the programming design of the hydrodynamic calculation is primarily organized on volumes and junctions. Components are organized collections of volumes and junctions and, to a lesser extent, the program is organized on components. Components are designed for either input convenience or to specify additional specialized processing. A pipe component is an example of a component designed for input convenience, since by taking advantage of typical features of a pipe, several volumes and junctions can be described with little more data than for one volume. Pump and valve components are examples of components requiring additional processing. A pump component includes data defining pump head and torque characteristics for single-phase and two-phase conditions as a function of pump angular velocity. A pump component requires additional processing to advance the differential equation defining pump angular velocity. A valve component requires additional data defining its characteristics and additional processing to calculate the junction flow area as a function of valve position.

Components are numbered with a three-digit number, 001 - 999. Components need not be in strictly consecutive order so that changes to a model of a hydrodynamic system requiring addition or deletion of components are easily made. Volumes and junctions within a one-dimensional component are numbered by appending a six digit number to the component number, CCCXX0000. The CCC is the component number, and XX is numbered consecutively starting at 01 for the volumes and junctions in the one-dimensional components. Volumes and junctions within a multi-dimensional component are numbered by appending a six digit number to the component number, CCCXYYZZ0. The CCC is the component number, X is the position number in the first coordinate direction (x in Cartesian geometry, r in cylindrical geometry), YY is the position number in the second coordinate direction (y in Cartesian geometry, θ in cylindrical geometry), and ZZ is the position number in the third coordinate direction (z in both Cartesian and cylindrical geometries).

2.4.1 Common Features of Components

Each volume's flow area, length, and volume must be supplied as input. As noted above, each one-dimensional volume has a x-coordinate direction along which fluid flows in a positive or negative direction, and may have y- and z-coordinate directions if crossflow connections are made to the volume or if the multi-dimensional component is used. The x-volume flow area is the volume cross-sectional area perpendicular to the x-coordinate direction. The x-volume length is the length along the x-coordinate direction and similarly for the y- and z-coordinate directions. The hydrodynamic numerical techniques require that the volume be equal to the volume flow area times the length for each coordinate direction. This requirement is easily satisfied for constant area volumes, but poses difficulties for irregular shaped volumes. Since it is very important that such a systems code as RELAP5-3D© conserves mass and energy, with momentum being an important but lesser consideration, we recommend that an accurate volume be used; that the volume flow area be the cross-sectional area averaged over the actual length of the volume; and the volume length be the quotient of the volume and the flow area. The component input routines permit the volume, flow area, and length of each volume to be entered as three nonzero positive numbers or two nonzero positive numbers and a zero. If three nonzero quantities are entered, the volume must equal

the flow area times length within a relative error of 0.000001. If one quantity is zero, that quantity is computed from the other two. The user need not be concerned with y- or z-coordinate data unless crossflow connections are made (and even then only if the default data for those coordinates are not satisfactory) or unless the multi-dimensional component is used.

The volume azimuthal (horizontal) angle specifies the orientation of the volume in the horizontal plane. The code numerics have no requirement for this quantity; this quantity is entered so a graphics package can be used to show isometric views of the system as an aid in model checking. Such a graphics package is available with RELAP5-3D©. The azimuthal (horizontal) angle is checked to verify that its absolute value is less than or equal to 360 degrees.

The volume inclination (vertical) angle specifies the vertical orientation of the volume. This quantity is used in the flow regime determination, is used in the interphase drag calculation, and is also used in the graphics package. The inclination (vertical) angle must be within the range +90 to -90 degrees. The angle 0 degrees means the x-coordinate direction is in the horizontal plane; a positive angle means that the coordinate direction is directed upward; a negative angle means it is directed downward. Slanted vertical orientation, such as an angle equal to 45 degrees, is permitted. Note that as the inclination (vertical) angle changes from zero, the y-coordinate is always in the horizontal plane, and that the x- and z-coordinates, and their associated faces move out of their original horizontal and vertical planes, respectively.

The horizontal flow regime map is used when the absolute value of the inclination (vertical) angle ϕ is less than or equal to 30 degrees. Horizontal flow calculations include a horizontal stratified flow capability and a horizontal stratified entrainment/pullthrough model. The vertical flow regime map is used when the absolute value of the inclination (vertical) angle ϕ is greater than or equal to 60 degrees. Between 30 and 60 degrees, interpolation is used.

The coordinate direction implies the position of the inlet and outlet ends of the volume. The terms inlet and outlet are convenient mnemonics relative to the coordinate direction but do not necessarily have any relation to the fluid flow. The direction of fluid flow is indicated by the sign of the velocity relative to the coordinate direction. For input convenience or ease in output interpretation, the coordinate direction should be an easily remembered direction, such as the normal flow direction as opposed to the flow in an accident situation.

As noted in the discussion of **Figure 2.1-7**, and described further below, a junction connects a specified end of one volume to the specified end of another volume. This, in turn, establishes relative positioning of the volumes. Because of gravity heads, the relative position is important to any volume with a nonzero vertical component of a volume coordinate direction. If the coordinate direction in a volume with a vertical component is reversed but no other changes are made, the inlet and outlet ends of the volumes are also reversed. The physical problem is changed since the relative vertical positions of the volumes are changed. If appropriate changes are made to junctions connecting the reversed volume, such that the physical problem remains unchanged, the only change in the problem results would be a reversal in the sign of the vector quantities associated with the volume. Furthermore, given a stack of vertically

oriented volumes, the proper gravity head is computed whether the direction coordinates are all upward, all downward, or any random distribution. This assumes that junction connections are such that a vertical stack is specified. As shown in **Figure 2.1-7**, a change in junction specification can change the relative position of two volumes from two, vertically stacked volumes to a U-tube configuration.

Input for a volume also includes the elevation change in a volume. The elevation change (Δz) is related to the volume length (Δx) and the calculated elevation angle (ϕ_{elev}) by

$$\Delta z = \Delta x \sin \phi_{elev} \quad . \tag{2.4-1}$$

Note that the elevation change associated with the x-coordinate has the same sign as the inclination (vertical) angle. To allow for irregularly shaped and curved volumes, the input elevation change is used for gravity head calculations. Input checks are limited to the following: the magnitude of the elevation change must be equal to or less than the volume length; for the x-direction, the elevation change must be zero if the inclination (vertical) angle is zero; and the elevation change must be nonzero and have the same sign as the inclination (vertical) angle if the inclination (vertical) angle is nonzero. The volume input does not need the elevation height of a volume relative to an arbitrary base. The elevation change data performs the same function in determining gravity heads.

If the hydrodynamic system has one or more loops, the user must ensure that the sum of the elevation changes of the volumes in each loop is zero. A loop is any hydrodynamic flow path starting at a volume, passing through one or more other volumes, and returning to the starting volume. If the net elevation change in a loop is not zero, an incorrect gravity head exists; this is comparable to having an undesired pump in the loop. This error is checked by the program. If closure is not within the error criterion of 0.0001 m, an input error will result. Input elevation inaccuracies in the fifth significant figure after the decimal point (i.e., on the order of 10^{-5}m) may or may not accumulate to give a difference of 10^{-4}m, depending on the numbering of the components.

The calculated elevation angle ϕ_{elev} is used in the additional stratified force term when a volume is not absolutely horizontal. This is discussed in Volume I, Section 3.1.1.4 of the manuals.

In modeling a straight pipe using a RELAP5-3D© volume, the inclination (vertical) angle ϕ is the same as the calculated elevation angle ϕ_{elev}. In modeling a curved pipe using a RELAP5-3D© volume, the inclination (vertical) angle ϕ will not be the same as the calculated elevation angle ϕ_{elev}.

Wall friction effects are computed from pipe roughness and hydraulic diameter data entered for each volume. If the input x-direction hydraulic diameter ($D_{h,x}$) is zero, it is computed from the x-direction volume flow area ($A_{v,x}$) assuming the cross-sectional area is circular,

$$D_{h,x} = \sqrt{\frac{4A_{v,x}}{\pi}} \quad . \tag{2.4-2}$$

If the input y-direction hydraulic diameter ($D_{h,y}$) is zero, it is computed from the y-direction volume flow area ($A_{v,y}$) and the x-direction volume flow area ($A_{v,x}$), using the formula

$$D_{h,y} = \frac{4A_{v,y}}{(\pi A_{v,x})^{0.5}} \quad . \tag{2.4-3}$$

This formula was obtained by using

$$D_{h,y} = \frac{4A_{v,y}}{\text{y-direction wetted-perimeter}} \quad . \tag{2.4-4}$$

The y-direction wetted-perimeter is obtained as follows:

The volume V is given by

$$V = A_{v,x}\Delta x = A_{v,y}\Delta y \quad . \tag{2.4-5}$$

Thus, the y-direction volume area is given by

$$A_{v,y} = \frac{V}{\Delta y} = \frac{A_{v,x}\Delta x}{\Delta y} \quad . \tag{2.4-6}$$

Assuming the x-direction cross-sectional area is circular, gives

$$\Delta y = D_{h,x} \tag{2.4-7}$$

and

$$A_{v,x} = \frac{\pi D_{h,x}^2}{4} \quad . \tag{2.4-8}$$

Substituting Equations (2.4-7) and (2.4-8) into Equation (2.4-6) yields

$$A_{v,y} = \frac{\pi}{4} D_{h,x} \Delta x \quad . \tag{2.4-9}$$

Defining the variable $\Delta y'$ as

$$\Delta y' = \frac{\pi}{4} D_{h,x} \quad , \tag{2.4-10}$$

Equation (2.4-9) can be written as

$$A_{v,y} = \Delta y' \bullet \Delta x \quad . \tag{2.4-11}$$

Using Equation (2.4-11), the y-direction wetted-perimeter can be written as

$$\text{y-direction wetted-perimeter} = 2\Delta x + 2\Delta y' \quad . \tag{2.4-12}$$

Substituting in Equation (2.4-10) yields

$$\text{y-direction wetted-perimeter} = 2\Delta x + 2\left(\frac{\pi}{4} D_{h,x}\right) \quad . \tag{2.4-13}$$

Substituting Equation (2.4-13) into the y-direction hydraulic diameter formula yields

$$D_{h,y} = \frac{4A_{v,y}}{2\Delta x + \frac{\pi}{2} D_{h,x}} \quad . \tag{2.4-14}$$

The $2\Delta x$ term can be neglected since a larger hydraulic diameter is desired to minimize the wall friction in the y-direction. This is because the junction K loss is usually more important than the wall friction in the y-direction half-volume of the cross flow volume. It should be noted that the K loss from handbooks includes the wall friction. It should also be noted that the user can turn off the wall friction in the y-direction using the y-direction volume friction (f) flag. If the x-direction cross-sectional area is not circular, the user should input one's own y-direction hydraulic diameter.

Neglecting $2\Delta x$ is Equation (2.4-14) yields

$$D_{h,y} = \frac{4A_{v,y}}{\frac{\pi}{2}D_{h,x}} \quad . \tag{2.4-15}$$

Substituting Equation (2.4-2) into Equation (2.4-15) yields the formula in Equation (2.4-3).

If the input z-direction hydraulic diameter ($D_{h,z}$) is zero, it is computed from the z-direction volume flow area ($A_{v,z}$) and the x-direction volume flow area ($A_{v,x}$), using the formula

$$D_{h,z} = \frac{4A_{v,z}}{(\pi A_{v,x})^{0.5}} \quad . \tag{2.4-16}$$

The derivation of this formula is similar to the derivation of the formula for $D_{h,y}$.

A check is made that the pipe roughness is less than half the hydraulic diameter for all three directions.

Most volumes allow seven control flags (tlpvbfe): the t flag is the thermal stratification flag; the l flag is the mixture level tracking model flag; the p flag is the water packer flag; the v flag is the vertical stratification flag; the b flag is the bundle interphase friction flag, recommended for cores and steam generators; the f flag determines whether wall friction from the volume is to be included or neglected; the e flag controls whether a nonequilibrium (two phases permitted to have unequal temperature) or an equilibrium (two phases forced to have equal temperatures) calculation is used. Generally we recommend that wall friction be computed in the x-direction and usually not in the y-direction or z-direction, and that the nonequilibrium equation of state be used.

The thermal stratification (t), mixture level tracking model (l), water packer (p), vertical stratification (v), bundle interphase friction (b), and equilibrium (e) flags can be entered only for the x- coordinate direction. The friction (f) flag can be entered for the x-, y-, and z-coordinate directions.

System volumes require initial thermodynamic state conditions, and time-dependent volumes require state conditions as a function of time or a time-advanced quantity. Seven options, numbered 0 through 6, are available to specify state conditions. Options 0 through 3 specify liquid/vapor-only conditions and do not allow a noncondensable gas. Option 0 requires pressure, liquid specific internal energy, vapor specific internal energy, and void fraction. Options 1 and 2 always specify saturation conditions. Option 1 requires saturation temperature and static quality in the equilibrium condition. Option 2 requires saturation pressure and static quality in the equilibrium condition. Two phases are present if the quality is neither 0 nor 1. Option 3 always specifies single-phase conditions and requires pressure and temperature. The next three options can specify the presence of a noncondensable gas. Option 4 requires pressure, temperature, and static quality in the equilibrium condition. Equilibrium conditions are assumed, and the vapor/gas consists of noncondensable gas and vapor at 100% humidity. Option 5 requires temperature, static quality, and

noncondensable quality in the equilibrium condition. Option 6 requires pressure, liquid specific internal energy, vapor/gas specific internal energy, void fraction, and noncondensable quality. Option 8 requires pressure, liquid temperature, vapor/gas temperature, void fraction, and noncondensable quality.

For options 0 through 6 and 8, the boron concentration is assumed to be zero. If 10 is added to the above option numbers, a boron concentration is required. See Section 3 of Volume I for a complete description of the boron transport model. Boron is assumed to be only present in and to be convected by liquid. If a volume with liquid and boron has the liquid removed by convection, the boron is also removed. If the liquid is evaporated, the boron remains. This is analogous to boron precipitating out as liquid is evaporated. Infinite solubility of boron in liquid is assumed and boron remains in solution regardless of its concentration until all of the liquid disappears. Boron instantly redissolves the instant the quality becomes less than 1. Boron concentrations are computed using only a boron continuity equation for each volume. Boron is assumed to have no momentum, no internal energy, and to have no effect on the equation of state.

Junctions connect two volumes by specifying a connection code for each volume. The connection code specifies both the volume and a specific face of the volume. Except for a pipe component, current components have only one volume, and a component reference is essentially a volume reference. The connection codes for each component type are described in the beginning of the input description for that component, as well as in Section 2.1.

The junction coordinate direction is from one volume face to another volume face and the input description use the words FROM and TO to identify the connections. If a junction is reversed, the sign of the vector quantities associated with the junction are reversed. To maintain the same physical problem, no further changes are needed in other components. The initial velocities in reversed system junctions or time-dependent velocities in time-dependent junctions should also be reversed.

Two quantities, the junction flow area and the junction area ratio (throat ratio), are defined from the user-supplied junction area (physical junction area at the throat). These are printed in the major edit as JUN.AREA and THROAT RATIO. Junctions can connect two volumes with possibly different volume flow areas, and the junction can also have a different flow area. Two options (smooth area and abrupt area) are provided for calculating area change effects as the fluid flows through the upstream volume flow area, the junction flow area, and the downstream volume flow area.

The smooth area change option uses only the stream tube form of the momentum equations that includes spatial acceleration and wall friction terms. This option should be used when there are no area changes or when the area changes are smooth, such as in a venturi. The user-supplied junction area may be smaller than, larger than, or between the adjacent volume flow areas (i.e., the junction area is not restricted). The junction flow area is set to the user-supplied junction area and the junction area ratio is set to 1.0 (except for valves, where the junction area ratio can be less than 1.0).

The abrupt area change option consists of two suboptions. The full abrupt area change option (a = 1) provides for area apportioning at branches (see Volume I, Section 3.5.1.1), restricted junction area (discussed below), extra interphase drag (see Volume IV, Section 7.1.1.2), and additional losses (see

Volume I, Section 3.4.3 and Volume IV, Section 7.1) resulting from sudden expansions, sudden contractions, sharp-edge orifices, and vena-contracta effects. The partial abrupt area change option (a = 2) includes everything in the a = 1 option except for the losses. The term “restricted junction area” means the user-supplied junction area (for a = 1 or a = 2) must be equal to or less than the minimum of the adjacent volume flow areas for an abrupt area change. The junction flow area is then set to the minimum of the adjacent volume flow areas, and the junction area ratio is set to the ratio of the user-supplied junction area and the minimum of the adjacent volume flow areas. The junction area ratio may be less than 1.0 for orifices and valves. When the user-supplied junction area equals the minimum of the adjacent volume flow areas, the junction area ratio is 1.0, and the junction is a contraction/expansion. Program logic checks flow direction, and an expansion with flow in one direction is treated as a contraction when flow reverses, and vice versa. If the user-supplied junction area is less than the minimum of the adjacent volume flow areas, an orifice is indicated, and the junction area ratio is less than one. Specifying the abrupt area change options (a = 1 or a = 2) when there is no area change gives the same result as specifying the smooth area change option (a = 0), but slightly more computer time is required.

Valve junctions using any area change options vary the junction area ratio as the valve opens and closes.

Junction velocities correspond to the junction flow area. Thus, the flow rate of a phase is the product of the appropriate junction volume fraction (donored quantity), the junction density (donored quantity), the junction velocity, and the junction flow area. For orifices and valves, the actual physical velocity is higher at the minimum area, which is the junction flow area multiplied by the junction area ratio. The junction area ratio is used to compute the velocity at the minimum area, when needed, such as in the choked flow model.

For user convenience, if the user-supplied junction area is zero, it is set to the minimum of the adjacent volume flow areas regardless of the area change option. This is the proper default value for most junctions, and only smooth area changes (where there is an area change) and orifices need nonzero user-supplied junction areas.

Junction user-inputted loss coefficients can be entered when additional losses above or instead of the wall friction and abrupt area change losses are needed. These losses could arise from pipe bends, irregularly shaped volumes, entrance or exit losses, or internal obstructions. Since the full abrupt area change model (a = 1) is only for sudden (i.e., sharp, blunt) area changes and sharp-edge orifices, the code-calculated loss is in general too high and not applicable for rounded or beveled area changes. The partial abrupt area change model (a = 2) with user input loss coefficients, should be used for rounded or beveled area changes. Six coefficients are entered. Three are for forward (positive) flow, where one is for the Reynolds number independent part and two are for the Reynolds number dependent part. The other three are for reverse (negative) flow, where one is for the Reynolds number independent part and two are for the Reynolds number dependent part. The coefficients are applied to the junction dynamic pressure. Zero coefficients mean no additional losses are computed.

Table 2.4-1 shows the junction flow area, abrupt area change model loss, and user-input loss for area change options a = 0, 1, and 2. The user input loss is optional in all three cases. For area change option a = 1, the abrupt area change model loss and the optional user-input loss are additive.

Table 2.4-1 Area change options.

Area change option (a)	Junction flow area	Abrupt area change model loss	User-input loss
0	User-supplied junction area	No	Optional
1	Minimum of adjacent volume flow areas	Yes	Optional
2	Minimum of adjacent volume flow areas	No	Optional

If junction user-supplied loss coefficients are entered and the abrupt area change option (a = 1 or a = 2) is selected, the user should note that the junction area is set to the minimum of the adjacent volume flow areas rather than the user-supplied junction area. Thus, the user-inputted loss coefficient needs to be adjusted for the difference between the minimum of the adjacent volume flow areas and the user-supplied junction area. For example, if the loss coefficient associated with the user-supplied junction area is K_j, then the adjusted loss coefficient $K_j^{'}$ would be computed as

$$K_j^{'} = K_j \frac{A_v^2}{A_j^2} \tag{2.4-17}$$

where A_j is the user-supplied junction area and A_v is the minimum of the adjacent volume flow areas.

It is important to note that RELAP5-3D© computes interfacial drag at junctions rather than within volumes. This has important implications with respect to modeling reactor core bundles and steam generator bundles. In these instances, the user should invoke the bundle interfacial drag model by specifying b = 1 on the volume control flags (see Appendix A). In addition, in modeling grid spacers as junctions within the core or steam generator bundle, the user should specify the junction area and hydraulic diameter as equal to that for the bundle, rather than those characteristics of the grid spacer. The reason for this is that the bundle interfacial drag model was formulated on the basis of bundle geometry. In order to achieve the correct pressure drop at each grid spacer junction, the user should input a loss coefficient that is adjusted for the difference between specifying bundle geometry rather than grid geometry. For example, if

the loss coefficient associated with the grid spacer is K_j, then the adjusted loss coefficient K_j' would be computed as

$$K_j' = K_j \frac{A_b^2}{A_g^2} \tag{2.4-18}$$

where A_g is the flow area of the grid spacer and A_b is the flow area of the bundle. In modeling the upper core tie plate, the user should specify the junction area and hydraulic diameter characteristics of the tie plate. The reason is the CCFL model was formulated based on actual geometry.

Eight control flags are associated with junctions (jefvcahs). The j flag is the jet junction flag, recommended at junctions where subcooled liquid is injected into the bottom of a pool. The e flag is the energy correction flag, recommended at break junctions into a containment. The f flag is the CCFL model flag, recommended for tie plates, downcomer annulus, etc. The v flag is the stratification entrainment/pullthrough model flag, recommended at break junctions connected to horizontal and vertical volumes.

The c flag controls applications of the choking model. The recommended input junction flags when the choking model is on (c = 0) are abrupt (a = 1 or 2) and nonhomogeneous (h = 0). (1) With regard to the abrupt area options (a = 1 or 2), these are discussed in Section 2.4.1. The full abrupt area change model (a = 1, code calculated losses) is recommended for sudden (i.e., sharp, blunt) area changes, while the partial abrupt area change model (a = 2, no code calculated losses, user input losses are to be used) is recommended for rounded or beveled area changes. The extra interphase drag term (see Volume IV) in the abrupt area model (a = 1 or 2), helps ensure more homogeneous flow that would be expected through a sudden area change. The smooth area change option (a = 0) is recommended only for when there is no area changes or there are smooth area changes (i.e., venturi). (2) With regard to the nonhomogeneous (h = 0) option, it is generally recommended that h = 0 be used. There may be rare situations where the combined interphase drag is too low, resulting in too much slip and too low mass flow. For this situation, the homogeneous option (h = 1 or 2) is recommended. (3) The user should monitor the calculated results for nonphysical choking. If this occurs, the user should turn choking off (c = 1) at junctions where this occurs.

The a flag is for the area change option, and has already been discussed. The h flag controls the type of momentum treatment; two-velocity, or one-velocity models. The two-velocity model is recommended except as indicated above. The s flag controls whether the momentum flux is to be used.

It is recommended that the momentum flux be omitted in the cross direction (perpendicular to the main upward flow direction) if either of the following two modeling approaches are used:

- 1D components (i.e., pipes, etc.) with cross flow junctions

- MULTID component, 1D option (1D momentum equations per Card CCC0001, Word 7 = 1 or 3).

System junctions require initial velocities, and time-dependent junctions require velocities as a function of time. Two options are available to specify the velocities. One option requires the velocities; the other requires mass flow rates from which the velocities are computed. If the flow is single-phase, the velocity of the missing phase is set to that of the flowing phase. This matches the transient calculation that computes equal phasic velocities when one phase is missing. The velocity conditions also require an interface velocity. This input is for future capability involving moving volume interfaces. For now, the interface velocity must be set to zero.

2.4.2 Time-Dependent Volume

A time-dependent volume must be used wherever fluid can enter or leave the system being simulated. The geometry data required are similar to system volumes, but during input processing the volume's length, elevation change, and volume are set to zero. With the staggered mesh, the pressure boundary would be applied in the center of the time-dependent volume. Setting these quantities to zero moves the boundary to the edge of the system volume.

The state conditions as a function of time or some time-advanced quantity are entered as a table, with time or the time-advanced quantity as the independent or search variable. The table must be ordered in increasing values of the search variable, and each succeeding value of the search variable must be equal to or greater than the preceding value. Linear interpolation is used if the search argument lies between search variable entries. End point values are used if the search argument lies outside the search variable entries. If constant state values are desired, only one set of data consisting of any search value and the associated constant data needs to be entered. The program recognizes when only one set of data is entered, and computer time is saved since the equation of state is evaluated only once rather than every time advancement. Step changes can be accommodated by entering two adjacent sets of data with the same time or an extremely small time difference.

The default search argument is time. If no trip number is entered, or if the trip number is zero, the current advancement time is used as the search argument. When a nonzero trip number is entered, a unit step function based on the time the trip was last set is applied. If the trip is false, the search argument is -1.0. When the trip is true, the search argument is the current advancement time minus the last time the trip was set. Thus, the search argument is always -1.0 when the trip is false and can range between zero and the problem time when the trip is true. A time-dependent volume could have some constant condition when the trip is false. In order to ensure proper operation for the constant condition, both a card with a negative search argument and a card with a zero search argument need to be entered (see example in Volume V). When the trip is true, it could follow a prescribed function of time where the time origin is the time of the trip not the start of the transient.

Through an input option, nearly any time-advanced quantity can be specified as the search argument. The allowed quantities are listed in the input description. The search variables in the table are assumed to

have the same units as the search argument, and the table lookup, interpolation, and treatment of out of range arguments are identical to those described for the default time argument. However, handling of trips is different. If the trip number is zero, the current value of the specified time-advanced variable is used. If the trip number is nonzero, the time delay cannot be applied as for the default time case, since the search argument may not be time. Thus, if the trip is false, the search argument is -1.0×10^{308}; if the trip is true, the current value of the specified variable is the search argument.

When time is the search argument, the current value is the value at the end of the time step; for any other variable, the current value is the value at the beginning of the time step. Time is the default search argument, but time can also be specified as the search argument through the input option of naming a time-advanced variable. These two uses of time as the search argument are different if a trip is used, since the default method can apply a time delay and the other cannot.

2.4.3 Time-Dependent Junction

Time-dependent junctions can be used whenever the phasic velocities or phasic mass flow rates are known as a function of time or other time-advanced quantity. Time-dependent junctions can connect any two system volumes, or a system volume and a time-dependent volume. Phasic mass flow rates are converted to phasic velocities using the upstream phasic densities and upstream phasic volume fractions. Examples of their use would be to model a constant displacement pump in a fill system, a pump or a valve (or both), by using an associated control system or measured experimental data. Time-dependent junctions are also used frequently in test problems to check code operation.

The phasic velocities or phasic mass flow rates in the time-dependent junction as a function of time or the time-advanced quantity are entered as a table, with time or the time-advanced quantity as the independent or search variable. The requirements, interpolation, and trip logic are identical to that for time-dependent volumes.

The capability of using time-advanced quantities as search arguments can be used to model pressure-dependent liquid injection systems. If the injection flow is a function of the pressure at the injection point, the volume pressure at that point is used as the search argument. A trip is defined to be true when the injection system is actuated. Entry of table data with a negative pressure and zero flows causes the flow to be zero when the trip is false. In order to ensure proper operation for zero flow, both a card with negative pressure and a card with a zero pressure need to be entered (see example in Volume V). The remaining table entries define the injection flow as a function of positive pressures. The source of injection liquid is a time-dependent volume. The pressure of the liquid supplied by the time-dependent volume could also be a function of the pressure at the injection volume to represent the work of pumping the liquid into the system. If the injection flow is a function of a pressure difference, the pressure difference can be defined by a control system variable, and that control variable is then defined as the search argument.

Some uses of time-dependent junctions can cause modeling difficulties. When using a time-dependent junction to specify flow from a time-dependent volume into the system, the incoming phasic densities, phasic volume fractions, phasic velocities, phasic mass flows, and phasic specific internal

energies can be specified. But when using a time-dependent junction to specify flow out of a system, the phasic densities, phasic volume fractions, and phasic specific internal energies of the fluid leaving the system are not known in advance. Thus, use of time-dependent junctions to control outflow is not recommended. The following is one example of a modeling problem. The user anticipates that a volume will contain only vapor/gas and accordingly sets a time-dependent junction to a nonzero vapor/gas flow and zero liquid flow. If the user anticipated incorrectly and liquid condenses or is carried into the volume, the liquid will accumulate unrealistically since it cannot leave the system.

In a simple pipe modeling application, a time-dependent volume and junction can be used to specify the inlet flow. Likewise, a time-dependent volume and time-dependent junction can model the feedwater flow into a reactor steam generator. Controlling the fluid flow out of the pipe or controlling the liquid-vapor/gas flow out of the steam generator through a time-dependent junction is not recommended. If a system junction (flows computed by the simulation rather than specified as boundary conditions) connected to a time-dependent volume is not sufficient, perhaps a servo valve can provide the required simulation.

2.4.4 Single-Volume

A single-volume component is simply one system volume. A single-volume can also be described as a pipe component containing only one volume. This single-volume component uses fewer input cards and fewer data items than does a pipe component. However, if the single-volume might be divided into several volumes for nodalization studies, we suggest the pipe component, since such changes are quite easy for pipes.

2.4.5 Single-Junction

A single-junction component is simply one system junction. It is used to connect other components such as two pipes. Initial junction conditions can be phasic velocities or phasic mass flow rates.

2.4.6 Pipe

A pipe component is a series of volumes and interior junctions, the number of junctions being one less than the number of volumes, and the junctions connect the outlet of one volume to the inlet of the next volume. Pipe components can be used for those portions of the system without branches. Pipe components offer input conveniences, since most characteristics of the volumes and junctions in a pipe are similar or change infrequently along the pipe, and input data requirements can be reduced accordingly. Because of the sequential connection of the volumes, junctions are generated automatically rather than being individually described. Although the input is designed to assume considerable similarity in volume and junction characteristics, any of the volume and junction features (such as flow area, orientation, pipe roughness, or control flags) can be changed at each volume or junction.

2.4.7 Branch

Branch components are provided to model interconnected piping networks. The branching model is based on one-dimensional fluid flow, which is adequate for most cases of branching and merging flow. Such situations include wyes, parallel flow paths from upper and lower plenums, and any branch from a vessel of large cross-section. For branching situations where phase separation effects caused by momentum or gravity are important, an approximate mapping technique can be used to map the two-dimensional situation into the one-dimensional space of the fluid model.

A branch component consists of one system volume and zero to nine junctions. The limit of nine junctions is due to a card numbering constraint. Junctions from other components, such as single-junction, pump, other branch, or even time-dependent junction components, may be connected to the branch component. The results are identical whether junctions are attached to the branch volume as part of the branch component or are in other components. Use of junctions connected to the branch, but defined in other components, is required in the case of pump and valve components and may also be used to attach more than the maximum of nine junctions that can be described in the branch component input.

2.4.8 Pump

The pump component model can be separated into models for hydrodynamics, pump-fluid interaction, and pump driving torque. The pump component input provides information for the hydrodynamic and pump-fluid interaction models and may optionally include input for an electric motor to drive the pump. A pump may also be connected to a shaft that is a specialized component within the control system. A shaft component is used when the pump is driven by a turbine or by an electric motor with a control system to regulate speed.

2.4.8.1 Pump Model Description. The hydrodynamic model of a pump component consists of one volume and two associated junctions. The coordinate directions of the junctions are aligned with the coordinate direction of the volume. One junction is connected to the inlet and is called the suction junction; the other junction is connected to the outlet and is called the discharge junction. The pump head, torque, and angular velocity are computed using volume densities and velocities. The head developed by the pump is divided equally and treated like a body force in the momentum equations for each junction. With the exception of the head term, the hydrodynamic model for the pump volume and junctions is identical to that for normal volumes and junctions.

2.4.8.1.1 Pump Performance Modeling--Interaction of the pump and the fluid is described by empirically developed curves relating pump head and torque to the volumetric flow and pump angular velocity. Pump characteristic curves, frequently referred to as four-quadrant curves, present the information in terms of actual head (H), torque (τ), volumetric flow (Q), and angular velocity (ω). These data are generally available from pump manufacturers. For use in RELAP5-3D©, the four-quadrant curves must be converted to a more condensed form, called homologous curves, which use dimensionless quantities. The dimensionless quantities involve the head ratio, torque ratio, volumetric flow ratio, and angular velocity ratio, where the ratios are actual values divided by rated values. The rated values are also

required pump component input and correspond to the design point or point of maximum efficiency for the pump.

The homologous curves are entered in tabular form, and the dependent variable is obtained as a function of the independent variable by a table search and linear interpolation scheme. There is a separate set of curves for head and torque, and each set is composed of eight curves. Not all the regimes need be described by the input, but a problem is terminated if an empty table is referenced. Both head and torque data must be entered for the regimes that are described with input.

The homologous curves for pump head and torque are for single-phase operation. These same tables are used by the two-phase pump model, but additional data must be input to model two-phase degradation effects.

Pump head data are always used in the momentum equations. Torque data may or may not be used in computing pump rotational velocity, depending on the pump motor model selected and if it is energized or not. However, both head and torque are used to determine pump energy dissipation, and consistent data must therefore be entered. The pump homologous data should be checked by computing pump efficiency from the homologous data. No such checking is currently included in RELAP5-3D©, nor is the operating efficiency edited on major edits.

The pump energy dissipation (see Volume I, Section 3.5.4) is given by

$$DISS = \tau\omega - gH(\alpha_f\rho_f v_f + \alpha_g\rho_g v_g)A \tag{2.4-19}$$

and the pump efficiency (η) is given by

$$\eta = \frac{gH(\alpha_f\rho_f v_f + \alpha_g\rho_g v_g)A}{\tau\omega} . \tag{2.4-20}$$

The pump energy dissipation is normally positive, and the pump efficiency is normally less than 1. If the efficiency exceeds 1 (i.e., the pump energy dissipation is negative), the code will take the thermal energy from the fluid and convert it to work. To avoid this, the user must input the rated torque carefully.

The sign conventions for various pump quantities are as follows: a pump operating in the normal pump regime has a positive angular velocity; the volumetric flow is positive if it is in the same direction as the volume coordinate direction; the head is positive if it accelerates the flow in the volume coordinate direction; and the torque is that exerted by the fluid on the pump and is negative if it tends to decelerate the pump. In normal pump regimes and in steady-state, this torque is negative and is balanced by the positive torque from the pump motor.

2.4.8.1.2 Pump Data Homologous Representation--The use of pump performance data in terms of nondimensional homologous parameters is often confusing. The purpose of this discussion is to briefly outline rules for a procedure to properly use the homologous data.

The homologous parameters for pumps are obtained from dimensional analysis that can only provide the conditions for similarity. Three independent parameters are obtained from application of Buckingham's Pi theorem.[2.4-1] They are

$$\pi_1 = \frac{Q}{\nu D} \tag{2.4-21}$$

$$\pi_2 = \frac{\omega Q^{\frac{1}{2}}}{(gH)^{\frac{3}{4}}} \tag{2.4-22}$$

$$\pi_3 = \frac{Q}{\omega D^3} \quad . \tag{2.4-23}$$

A fourth parameter that is commonly used can be obtained by a combination of π_2 and π_3 to yield

$$\pi_4 = \frac{gH}{\omega^2 D^2} \quad . \tag{2.4-24}$$

The first parameter, π_1, is analogous to a Reynolds number and is the only parameter involving the fluid kinematic viscosity, ν. Experience with pump design and scaling has shown that viscous effects caused by skin friction are small, especially for high Reynolds number flows, and, in practice, the requirement to maintain π_1 constant is not used. The use of π_2, π_3, and π_4 to correlate pump performance has proven quite useful. The parameter π_2 is called the specific speed and is often used as a single parameter to characterize the type of pump impeller best suited for a particular application. In practice, the acceleration of gravity, g, is omitted, and the specific speed is simply defined as

$$\omega_s = \frac{\omega Q^{\frac{1}{2}}}{H^{\frac{3}{4}}} \tag{2.4-25}$$

where the pump speed ω is in rpm, the capacity Q is in gpm, and the head H is in ft. In this form, ω_s is not dimensionless but has a history of usage that still persists.

Two performance parameters that are used for pump modeling are the specific nondimensional capacity,

$$Q_s = \frac{Q}{\omega D^3} \tag{2.4-26}$$

and the specific head (dimensional due to omission of the gravitational acceleration constant),

$$H_s = \frac{H}{\omega^2 D^2} \quad . \tag{2.4-27}$$

The D that appears in Equations (2.4-26) and (2.4-27) is a characteristic dimension of the pump and is assumed to be the impeller diameter. When scaling pump performance using homologous parameters, the implication is that all pump dimensions are geometrically similar (i.e., changing D implies a proportional change in impeller width, leakage paths, and in all linear dimensions of the pump).

When the pump torque performance is included, one additional dimensionless parameter is obtained from dimensional analysis and is

$$\pi_5 = \frac{\tau}{\rho \omega^2 D^5} \tag{2.4-28}$$

where π_5 is the nondimensional specific torque.

Generally, constant density is assumed, so the dimensional specific torque used in constructing the homologous representation is reduced to

$$\pi_s = \frac{\tau}{\omega^2 D^5} \quad . \tag{2.4-29}$$

Homologous states are states for which specific capacity, head, and torque are all constant. Thus, at any state it is possible, within the limitations of similarity theory, to predict the performance for other combinations of speed, head, and flow that have the same homologous state. It is also possible to scale pump performance with reasonable accuracy to account for changes in physical pump size through the diameter D by keeping the homologous parameters fixed.

Pump performance data are usually displayed by plotting head and torque as functions of speed and volumetric flow. **Figure 2.4-1** is a four-quadrant pump curve for the Semiscale MOD1 pump, and has speed and flow as independent variables with lines of constant head.

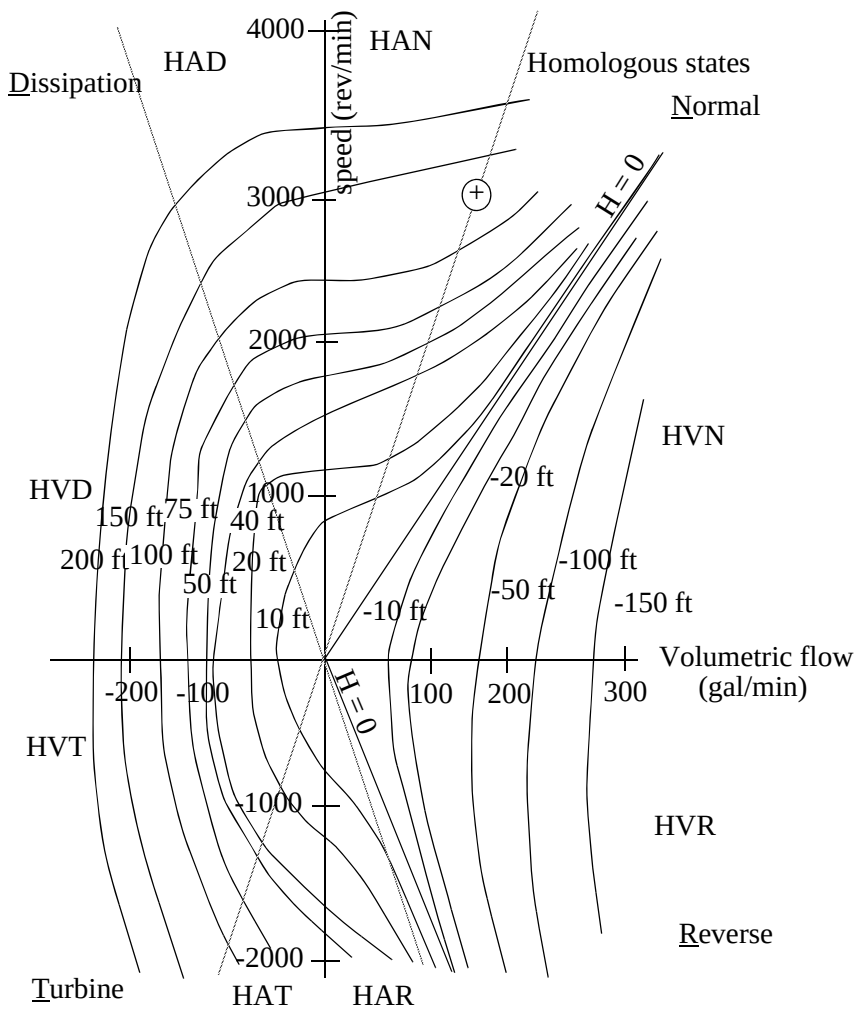


Figure 2.4-1 Four-quadrant head curve for Semiscale MOD1 pump (ANC-A-2083).

Figure 2.4-2 is a comparable four-quadrant plot of the Semiscale MOD1 pump torque data. All possible operating states of the pump are represented on such plots. These data for a particular pump can be approximately collapsed into a single curve by nondimensionalizing specific head and capacity parameters for corresponding homologous operating points using the design point values for head, capacity, and speed.

All points on **Figure 2.4-1** having the same specific capacity are straight lines passing through the origin (lines of constant Q/ω). The impeller diameter is omitted from the homologous parameters since it is constant for a particular pump. The design operating point is indicated by the cross. The line of constant Q/ω, passing through the design point and its reflection about the ordinate divides each quadrant into two

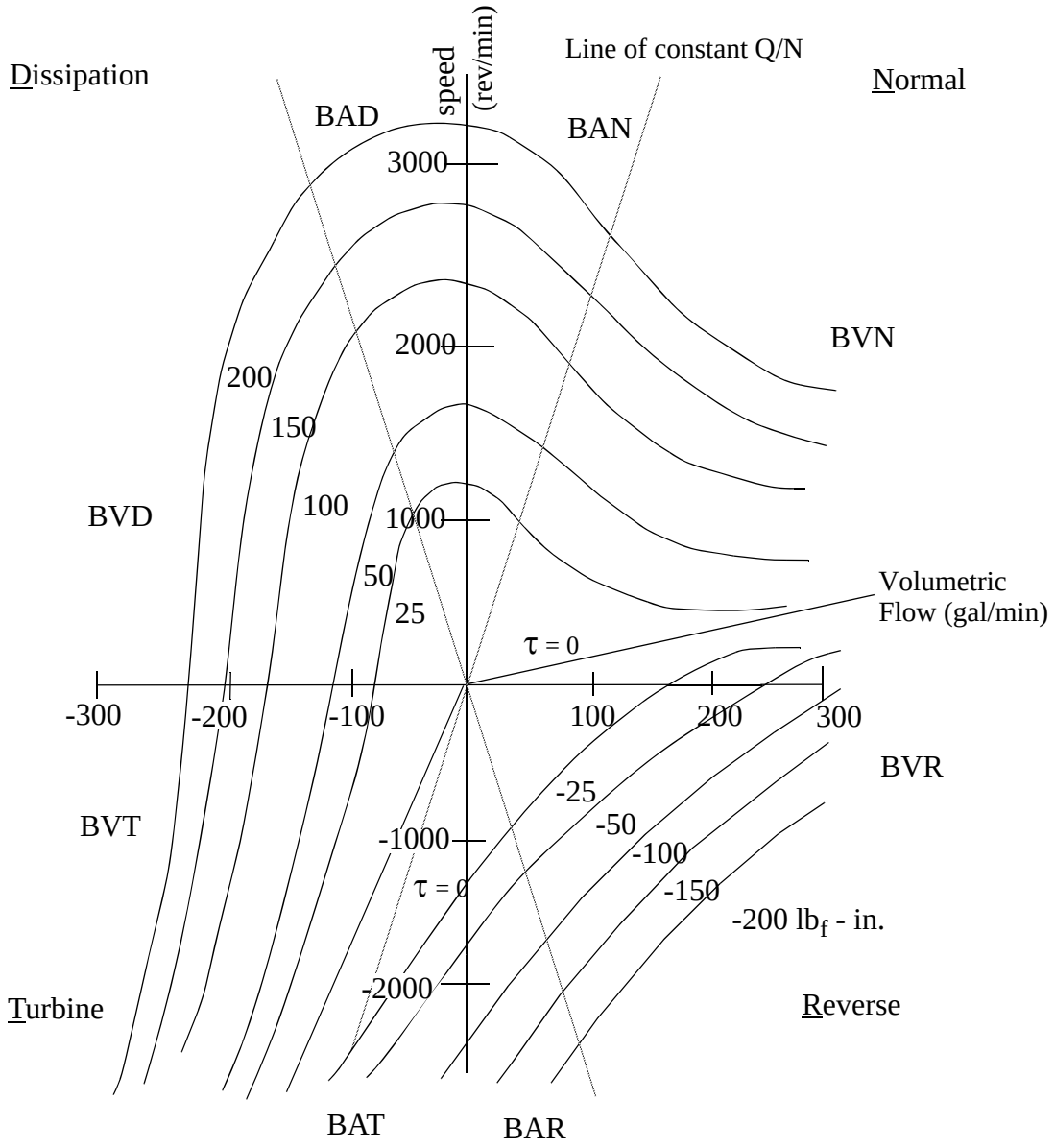


Figure 2.4-2 Four-quadrant torque curve for Semiscale MOD1 pump (ANC-A-3449).

octants. Each of these eight octants is named according to the convention listed in **Table 2.4-2**, for the purpose of constructing the homologous representation.

Table 2.4-2 Pump homologous curve definitions.

Regime number	Regime mode ID name	α	v	v/α	Independent variable[a]	Dependent[a] variable head	Dependent[a] variable torque
1	HAN BAN Normal pump	> 0	≥ 0	≤ 1	v/α	h/α^2	β/α^2
2	HVN BVN Normal pump	≥ 0	≥ 0	> 1	α/v	h/v^2	β/v^2
3	HAD BAD Energy dissipation	> 0	< 0	≥ -1	v/α	h/α^2	β/α^2
4	HVD BVD Energy dissipation	≥ 0	< 0	< -1	α/v	h/v^2	β/v^2
5	HAT BAT Normal turbine	< 0	≤ 0	≤ 1	v/α	h/α^2	β/α^2
6	HVT BVT Normal turbine	< 0	≤ 0	> 1	α/v	h/v^2	β/v^2
7	HAR BAR Reverse pump	< 0	> 0	≥ -1	v/α	h/α^2	β/α^2
8	HVR BVR Reverse pump	< 0	> 0	< -1	α/v	h/v^2	β/v^2

a. α = rotational ratio; v = volumetric flow ratio; h = head ratio; and β = torque ratio. Note: For the case $\alpha = 0$ and v = 0 in regime 2, h = 0 and $\beta = 0$.

The four-quadrant pump head and torque maps in **Figure 2.4-1** and **Figure 2.4-2** can be reduced to the homologous representation curves in two steps. First, the maps are made dimensionless by using the rated head, H_R, flow, Q_R, speed, ω_R, and torque, τ_R, to form corresponding dimensionless parameters $h = \frac{H}{H_R}$, $v = \frac{Q}{Q_R}$, $\alpha = \frac{\omega}{\omega_R}$, and $\beta = \frac{\tau}{\tau_R}$, respectively. Second, the data are plotted in terms of the

homologous parameter h/α^2 or h/v^2, v/α or α/v, and β/α^2 or β/v^2. The parameter used depends upon the octant in which the curve is being plotted. The choice is made so that the values are bounded (i.e., the denominators never vanish and, in the case of the capacity parameter, the range of variation is confined between plus and minus 1.0). **Figure 2.4-3** is the homologous head curve that is obtained from the head map in **Figure 2.4-1**. Note that not all points fall on a single curve. This is a result of the inexact nature of the similarity theory. Real pumps do not perform exactly according to the similarity relations due to leakage, viscous effects, etc.; however, the correspondence is surprisingly close, as evidenced by the tight clustering of points. The homologous curve for the torque data of **Figure 2.4-2** is shown on **Figure 2.4-4**. Since the data do not form a single curve, the design operating usual approach is to use least squares or other smoothing techniques to obtain curves passing through the point (1.0, 1.0). These curves must also be continuous at the point v/α or α/v equal to + 1.0. The legends on **Figure 2.4-3** and **Figure 2.4-4** have a key indicating which of the homologous parameters are used in each octant. All combinations of head, flow, speed, and torque can now be located on a corresponding segment of the homologous curve. Note that the impeller diameter parameter that appears in the dimensionless similarity parameters is not used in the homologous reduction of the four-quadrant representation; thus, special considerations are necessary for application of the data to a larger but geometrically similar pump. The advantage of using the homologous pump performance data representation in a computer code is obvious. Two-dimensional data arrays and two-dimensional interpolation are avoided, and, only two parameter tables and one-dimensional interpolation are required.

2.4.8.1.3 Homologous Data and Scaling--In most system simulation tasks, complete pump performance data are not available. Usually, only first-quadrant data are available (normal operation), and sometimes only the design or rated values are known. In the case of full-scale nuclear power plant pumps, it is difficult to test the pumps in all octants of operation or even very far from design conditions. The usual approach to obtain data for such systems is through the use of scaled-down pump tests.

The scaled pump test data can be for the same physical pump operated at reduced speed or for a pump scaled in size such that similarity is preserved. For the case of a pump scaled in size, it is necessary to maintain the similarity in specific head, capacity, speed, and torque parameters. (Note that the diameter was dropped in the development of the homologous performance model since a fixed configuration was considered.) Consideration of the diameter change must be implicitly included in the selection of “rated” parameters to properly account for changes in geometric scale. The homologous parameters, including the impeller diameter, are given in Equations (2.4-26), (2.4-27), and (2.4-29). When a change in scale is considered, an additional degree of freedom is possible, since only two parameters, the rated specific head and capacity, must be held constant. The specific speed is also held fixed whenever both specific head and capacity are kept fixed. There are many combinations of N and D for which this is possible.

The usual situation encountered in applications work is that homologous data exist for a similar pump, and the question arises, “Can we use these data to simulate our pump by adjusting the rated parameters?” The question can best be answered by the following statements. First, the best approach is to use the rated conditions corresponding to the pump used to generate the data. Second, the rated condition can be changed if the specific head and capacity are kept the same as for the pump used to generate the data. Similarity is assured since the modeled pump will have the same specific speed. The rated conditions

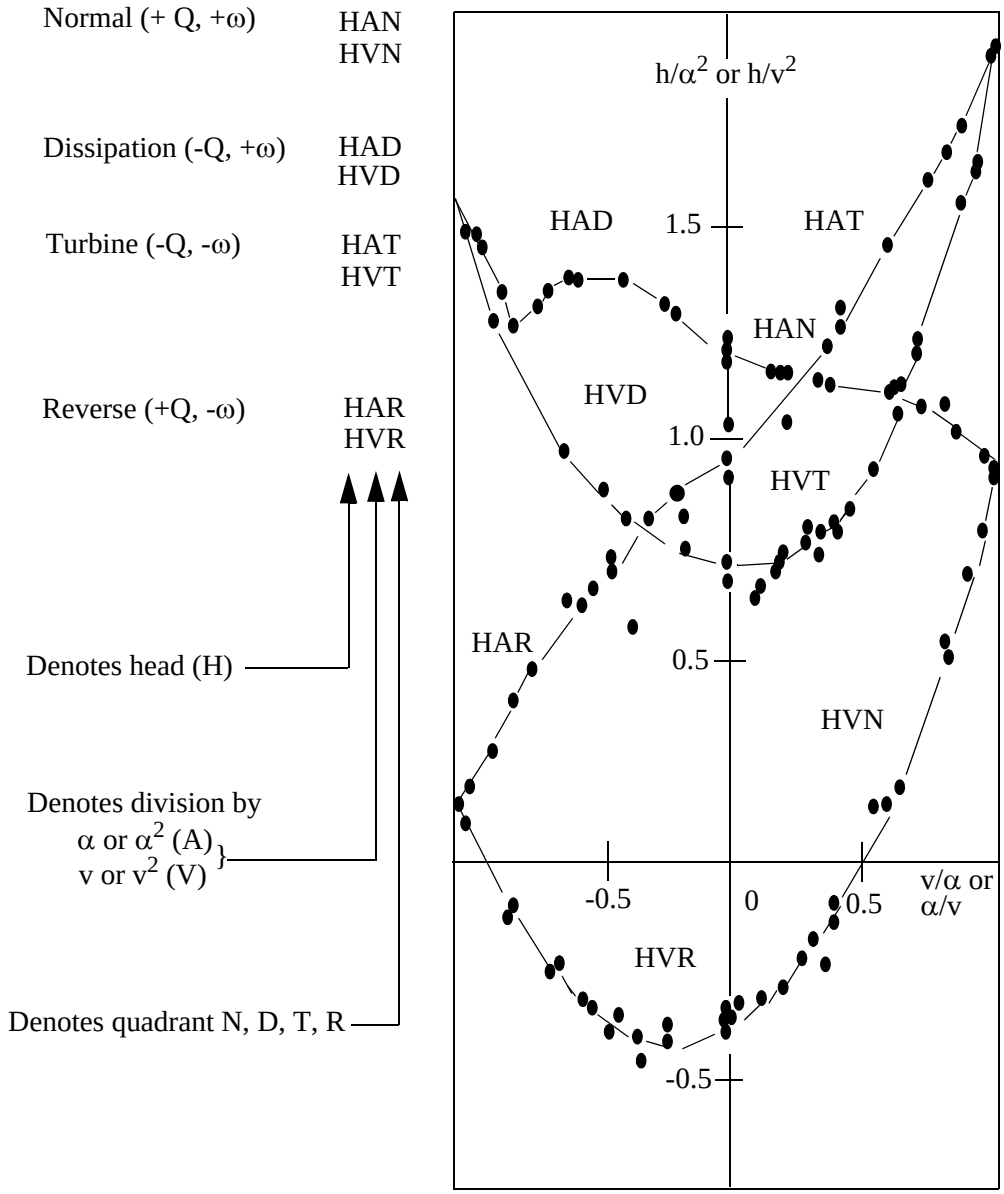


Figure 2.4-3 Homologous head curve.

by definition locate the region of pump operation on the homologous performance curve, at the design point or point of maximum efficiency. The rated conditions can then be safely adjusted in this way. They can also be adjusted using the impeller diameter as an additional parameter while still maintaining the rated specific speed, head, and capacity constant. However, this type of scaling implies a change in pump geometry and the extrapolation depends more heavily on the validity of the pump similarity relationships, which are only approximate. In all cases, the rated conditions must correspond to the same specific speed as the pump used to produce the data. However, the operating point does not have to correspond to the

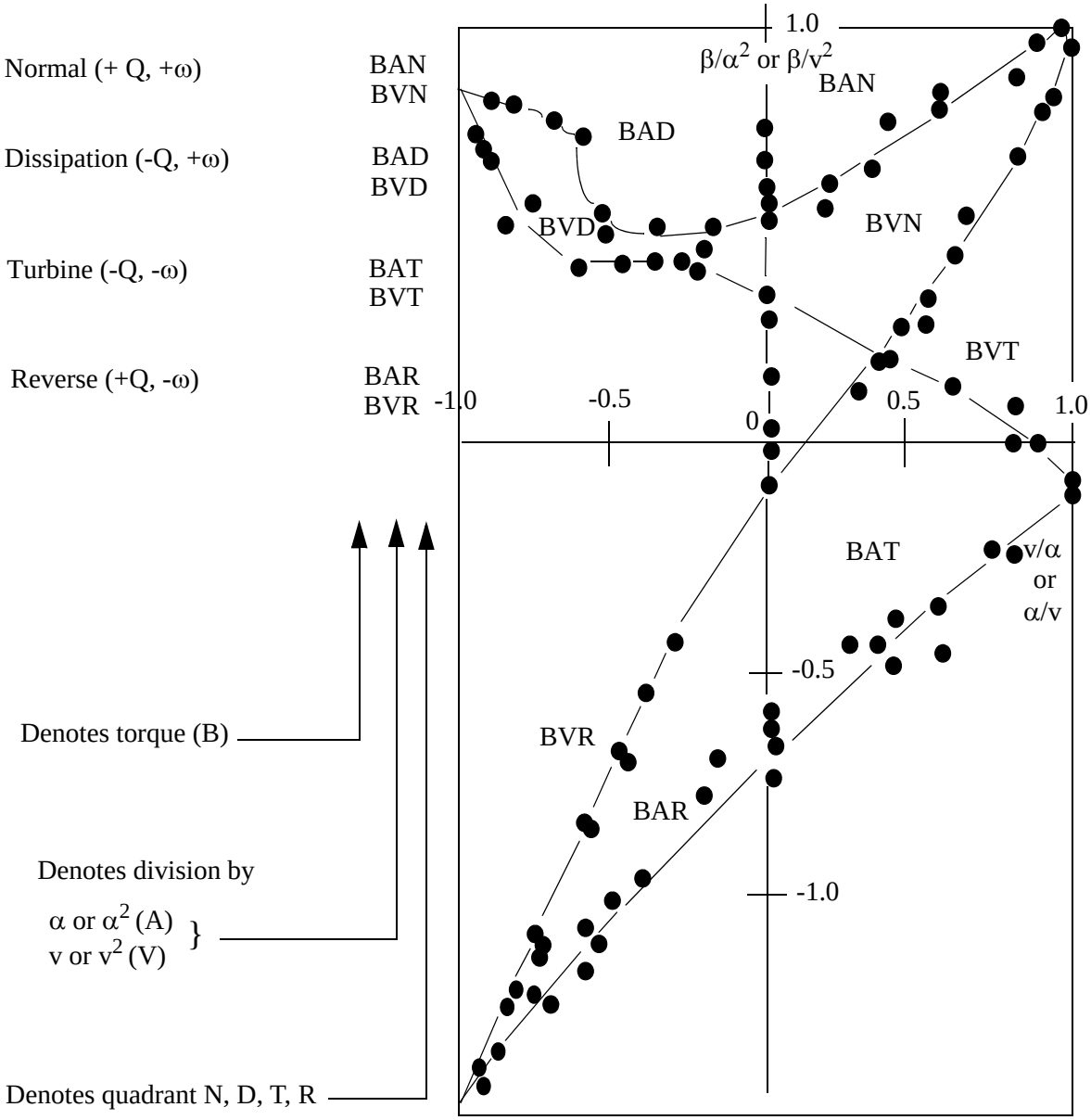


Figure 2.4-4 Homologous torque curve.

rated conditions. In such a case, the pump will operate at off-design conditions and efficiency will be less than the design value. Such off-design adjustments may be desirable to better match the modeled pump head-flow characteristics at the system operating point. This methodology was originally developed by Farman and Anderson[2.4-3] for the RELAP3 code, and then it became the accepted methodology in the RELAP4 and RELAP5 codes.

2.4.8.1.4 Two-Phase Performance Representation--The discussion above applies to a pump operated with a single-phase fluid of constant density. When pump performance operation with a

two-phase fluid is considered, the homologous representation of performance data has a less firm basis. An empirical modification of the homologous approach has therefore been developed based on Semiscale and Westinghouse Canada Limited (WCL) data. The RELAP5-3D© two-phase pump model is the same as that developed for RELAP4.[2.4-2] The approach is one in which the two-phase performance data are plotted, and a lowest performance envelope is constructed. This curve is called the fully degraded two-phase performance. The fully degraded performance and the single-phase performance data are used to form two-phase difference homologous performance curves for head or torque. The pump performance is then expressed in terms of the single-phase data and the difference data using a two-phase multiplier that can be a function of void fraction only or a function of void fraction and pressure. The pump head is expressed as

$$H = H_{1\phi} - M_H \, \Delta H \tag{2.4-30}$$

where ΔH is the head difference obtained from the single-phase to two-phase difference homologous curve. The function M_H is the two-phase multiplier, defined such that it is zero for the void fraction, α_g, equal to 0.0 and 1.0. It is recommended that, if appropriate data are available, the two-phase multipliers be entered as a function of void fraction and pressure. The pump torque is expressed in a similar way. Very little advice can be offered with respect to scaling of the two-phase performance data. Generally, it is assumed that the same similarity principles used for single-phase performance also hold for two-phase performance. A complete set of data was generated for a Semiscale pump, and these data are widely used for predicting two-phase performance of other pumps.

2.4.8.1.5 Pump Velocity Modeling--The pump computation for a time step begins by computing pump head and torque from the homologous data using pump angular velocity and volume conditions at the beginning of the time step. The head is used in the momentum equations. The remaining pump calculation determines the pump angular velocity at the end of the time step. The logic for computing pump angular velocity is complex, since stop logic, friction, an initializing calculation, the presence or absence of two tables, and two trips are involved. Additional capability is provided if the pump is associated with a shaft component. An optional card in the pump component input data specifies whether the pump is associated with a shaft. The remainder of this section defines pump capability when not associated with a shaft. In Section 4.2.3, the available shaft component capabilities are described and user suggestions are given.

Pump frictional torque (τ_{fr}) is modeled as a constant or as four-term function of the pump rotational velocity and is given by

$$\tau_{fr} = \pm\tau_{frn} \qquad \text{for } \left|\frac{\omega}{\omega_R}\right| < S_{PF} \tag{2.4-31}$$

$$\tau_{fr} = \pm\left(\tau_{fr0} + \tau_{fr1}\left|\frac{\omega}{\omega_R}\right|^{x1} + \tau_{fr2}\left|\frac{\omega}{\omega_R}\right|^{x2} + \tau_{fr3}\left|\frac{\omega}{\omega_R}\right|^{x3}\right) \qquad \text{for } \left|\frac{\omega}{\omega_R}\right| \geq S_{PF} \tag{2.4-32}$$

where ω is the pump rotational velocity; ω_R is the input rated pump rotational velocity; and τ_{frn}, τ_{fr0}, τ_{fr1}, τ_{fr2}, τ_{fr3}, x1, x2, x3, and S_{PF} (pump friction critical speed ratio) are input data. The pump frictional torque is negative if $\frac{\omega}{\omega_R} > 0$, and it is positive if $\frac{\omega}{\omega_R} < 0$.

The pump model has special capabilities to accommodate experimental systems. For example, the LOFT system primary pumps use a motor-generator, flywheel, fluid coupling, and an active control system in order to better represent full-size PWR pumps. Allowing a variable pump inertia provides a simple model of the LOFT pump rotational behavior. To facilitate LOFT usage, pump input provides for constant inertia or optionally allows input of variable inertia data. The variable pump inertia (I_p) is given by

$$I_p = I_{pn} \qquad \text{for } \left|\frac{\omega}{\omega_R}\right| < S_{PI} \tag{2.4-33}$$

$$I_p = I_{p0} + I_{p1}\left|\frac{\omega}{\omega_R}\right| + I_{p2}\left|\frac{\omega}{\omega_R}\right|^2 + I_{p3}\left|\frac{\omega}{\omega_R}\right|^3 \qquad \text{for } \left|\frac{\omega}{\omega_R}\right| \geq S_{PI} \tag{2.4-34}$$

where ω is the pump rotational velocity, ω_R is the input rated pump rotational velocity, and I_{pn}, I_{p0}, I_{p1}, I_{p2}, I_{p3}, and S_{PI} (pump inertia critical speed ratio) are input data.

A pump stop card containing limits on problem time, forward pump angular velocity, and reverse angular velocity may optionally be entered. The pump angular velocity is set to zero and remains zero for the remainder of the problem if any of the limits are exceeded. Selected tests can effectively be disabled by entering a very large number for the limits. If the problem time limit = 0, then the problem time test is ignored.

A time-dependent pump velocity table and an associated trip number may be entered. If the table is entered and the trip number is zero, the pump angular velocity is always determined from this table. If the trip number is nonzero, the table is used only when the trip is true. The default search variable for the time-dependent pump velocity table is time, but time-advanced quantities may be specified as the search variable. When time is the search variable by default, the search argument is time minus the time of the trip. When a time-advanced variable is specified as the search variable (even if it is time), the search argument is just the specified variable. The use of the pump velocity implies a pump motor to drive the pump at the specified velocity.

The following is a possible example of the use of a time-advanced variable as the search argument in the pump velocity table. The motor and its control system that drives a BWR recirculation pump could be modeled using the control system with one of the control variables representing the rotational velocity of the motor. The recirculation pump would be modeled as a hydrodynamic pump component. The torque exerted by the liquid on the pump would be one of the input variables to the control system model. Motor velocity would be supplied to the pump component by specifying the motor velocity as the search

argument of the time-dependent pump velocity table. The table would relate the motor rotational velocity to the pump rotational velocity. If the motor and pump were directly coupled, the search variables and dependent variables would be the same.

Whenever the time-dependent pump angular velocity table is not being used, the pump angular velocity is determined by the advancement in time of the differential equation relating pump moment of inertia, angular acceleration, and net torque. The net torque is the pump motor torque minus the homologous torque value and the frictional torque. If the pump trip is false, electric power is being supplied to the pump motor; if the trip is true, electric power is disconnected from the pump motor and the pump motor torque is zero. If a table of pump motor torque as a function of pump angular velocity is entered, the pump motor is directly specified and motor torque is obtained from the table, interpolating when needed. If the table is missing, the pump motor is implied and torque is assumed to be such that the net torque is zero. This is implemented in the program by simply setting the pump angular velocity at the end of the time step equal to that at the beginning of the time step. This latter option is usually used when the problem starts with the pump at its normal steady-state velocity; the pump is assumed to remain at this velocity until the pump trip, and the trip, once true, remains true for the rest of the problem.

2.4.8.2 Pump Modeling Examples. Two examples are discussed to illustrate pump operation. Consider a pump in a closed loop filled with liquid. At the start of the transient, all the liquid in the loop is at zero velocity but the pump is rotating in the positive direction. No pump motor torque table is used, the pump trip is initially false, and thus, the pump angular velocity is constant at the initial value until the pump trip becomes true. With the pump rotating at a constant angular velocity but the liquid at rest, the head is high and the liquid is accelerated. As the velocity of the liquid increases, wall friction and area change losses increase because of the dependence of these losses on liquid velocity. At the same time, the pump head obtained from the homologous data will decrease as the volumetric flow increases. A steady-state will be reached when the pump head and the loss effects balance. If no wall friction options are selected for the loop piping and no area losses are present, the liquid will accelerate until the pump head is zero. When steady-state is reached and the pump trip is then set true, the pump will begin to decelerate because the pump friction torque and the torque exerted by the liquid on the pump are no longer balanced by the pump motor torque. The liquid also begins to decelerate owing to loss effects. The interaction between the liquid and pump depends on the relative inertias and friction losses between the two. If the liquid tends to decelerate more rapidly than the pump, the pump will use its rotational kinetic energy to maintain liquid velocity. If the pump tends to decelerate more rapidly than the liquid, the pump, depending on its design as reflected in the homologous data, may continue to act as a pump or the kinetic energy of the liquid may tend to maintain pump angular velocity.

The second example is similar to the first example except that the initial pump angular rotational velocity is zero and a pump motor torque curve for an induction motor is used. From the curve, the torque is positive at zero angular velocity and increases slowly as the velocity increases to a value slightly below the synchronous speed. Then, the torque decreases sharply to zero at the synchronous speed and continues to negative torque. At the initial conditions, the net torque is positive, the pump angular velocity increases, and the liquid is accelerated. If the pump torque is sufficiently high, the pump velocity increases to slightly below the synchronous speed where the developed torque matches the frictional torque and the torque imposed by the liquid. As the liquid accelerates, the angular velocity decreases slightly to meet the

increased torque requirements. The angular velocity decrease is very small owing to the steep slope of the torque versus angular velocity near the synchronous speed. Thus, once the pump approaches the synchronous speed, the transient behavior of the second example is similar to the first example.

2.4.8.3 Built-in Pump Data. RELAP5-3D© contains built-in, single-phase homologous data for a Bingham Pump Company pump with a specific speed of 4,200 and a Westinghouse Electric Corporation pump with a specific speed of 5,200. The built-in Westinghouse curves are typical curves for Westinghouse coolant pumps and were sent to the INL from Westinghouse for use in blowdown calculations in the LOFT project.[a] Two-phase difference homologous data are also associated with these pumps, but the data curves are identical and were obtained from two-phase tests of the Semiscale pump. (The data curves are stored as data statements in subroutine RPUMP.) No built-in, two-phase multiplier tables are entered. Specification of built-in, single-phase homologous data does not require specification of the built-in, two-phase difference homologous data, or vice versa.

If multiple pump components are used and some tables are common to more than one component, then user effort and computer storage can be saved by entering the data for only one component and specifying that other components use that data. This holds true for built-in data, since built-in data are treated as input data and stored in the pump component data when requested. There are no component ordering restrictions when one pump component references tables in another pump component. Thus, a pump component may reference a pump component numbered higher or lower than itself. Also, a pump component may reference another pump component that references another pump component, as long as a pump component with data entered is eventually reached.

2.4.8.4 Pump Edit Parameters. Major output edits include pump performance information in addition to the quantities common to all volumes and junctions. Pump angular velocity, head, torque, octant number, and motor torque are edited. Pump angular velocity, head, torque, motor torque, and inertia are available as minor edit variables. The pump torque is the sum of torque from homologous data and friction effects. Pump motor torque is zero if the motor is tripped or if no motor is directly specified or implied.

2.4.9 Jet Pump

A jet pump is modeled in RELAP5-3D© using the JETMIXER component. In a jet pump, the pumping action is caused by the momentum mixing of the high-speed drive line flow with the slower suction line flow. **Figure 2.4-5** contains a schematic showing the typical nodalization used for a jet pump mixing section.

2.4.9.1 Input Requirements. The input for a JETMIXER component is the same as that for a BRANCH component, with the following modifications:

---

a. Letter from W. C. Gangloff, PWR Systems Division, Westinghouse Electric Corporation, to R. F. Farman, Idaho Nuclear Corporation, National Reactor Testing Station, E-L-035, December 7, 1970.

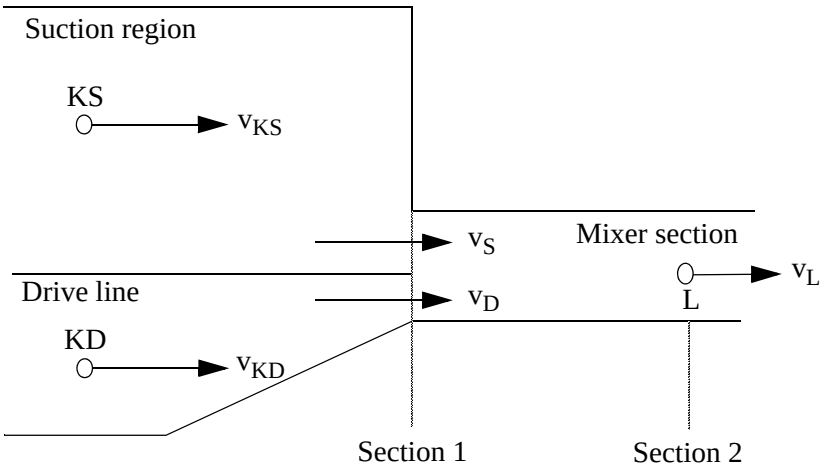


Figure 2.4-5 Schematic of mixing junctions.

1. For a BRANCH component, the junctions connected to that branch can be input with the branch or as separate components. For a JETMIXER, three (and only three) junctions, representing the drive, suction, and discharge, must be input with the JETMIXER component, i.e., NJ = 3. If NJ is not equal to 3, an input error message is printed.

2. The three junction card sequences must be numbered as follows: Cards CCC1101 and CCC1201 represent the drive junction. Cards CCC2101 and CCC2201 represent the suction junction. Cards CCC3101 and CCC3201 represent the discharge junction in the mixing section.

3. The drive and suction junctions must have their TO connection codes referring to the JETMIXER volume, and the discharge junction must have its FROM connection code referring to the JETMIXER volume. If this is not the case, an input error message is printed. The drive and suction junctions must be connected to the inlet side of the JETMIXER volume, and the discharge junction must be connected to the outlet of the JETMIXER volume. If this is not the case, an input error message is printed.

2.4.9.2 Recommendations. Although the junction and volume areas for a JETMIXER are not restricted, the JETMIXER will properly model a jet pump only if the drive and suction junctions flow areas sum to the JETMIXER volume area.

The drive and suction junctions can be modeled with smooth or abrupt area changes. If they are modeled as smooth junctions, then the appropriate forward and reverse loss coefficients must be input by the user. They should be obtained from standard references for configurations similar to those of the jet pump being modeled. The use of smooth junctions gives the user more explicit control over the resistance

coefficients. In either case, it should be remembered that the turning losses associated with reverse flow through the suction junction are automatically included in all code calculations.

The JETMIXER component volume is intended to represent the mixing region of the jet pump. The diffuser section of a jet pump normally follows the mixing section. The diffuser section is not an integral part of the JETMIXER component and must be modeled using one or more additional volumes. Several volumes with slowly varying cross-sections and the smooth junction option can be used to model the diffuser region.

2.4.9.3 Additional Guidelines. It has been customary to identify jet pump operations in terms of two dimensionless parameters. These are the M and N parameters, defined as follows:

The M ratio (flow ratio) is the suction flow rate, W_S, divided by the drive flow rate, W_D, and is given by

$$M = \frac{W_S}{W_D} \quad . \tag{2.4-35}$$

The N ratio (head ratio) is the increase in dynamic pressure for the suction-discharge path divided by the loss of dynamic pressure for the drive-discharge path, and is given by

$$N = \frac{\left(P + \frac{1}{2}\rho v^2 + \rho g H\right)_{Dis} - \left(P + \frac{1}{2}\rho v^2 + \rho g H\right)_S}{\left(P + \frac{1}{2}\rho v^2 + \rho g H\right)_D - \left(P + \frac{1}{2}\rho v^2 + \rho g H\right)_{Dis}} \quad . \tag{2.4-36}$$

Figure 2.4-6 shows an expanded view of the normal operating region (first quadrant) with several curves representing different flow resistances. This figure can be used as a guide for modeling different jet pump geometries. Each curve shows the M - N performance generated with base-case loss coefficients plus a single additional loss coefficient (K = 0.2) added to either the drive, suction, or discharge junction. This figure gives an indication of the quantitative change in performance caused by the respective drive, suction, or discharge losses. Using this figure, one can, with a few preliminary runs, design a code model for a specific jet pump if the performance data are available. If no specific performance data are available, we recommend that standard handbook losses be applied.

2.4.9.4 Output. There is no special output printed for the JETMIXER component. We recommend that control variables be used to set up the M and N parameters for minor edit purposes and that these parameters be printed with every edit.

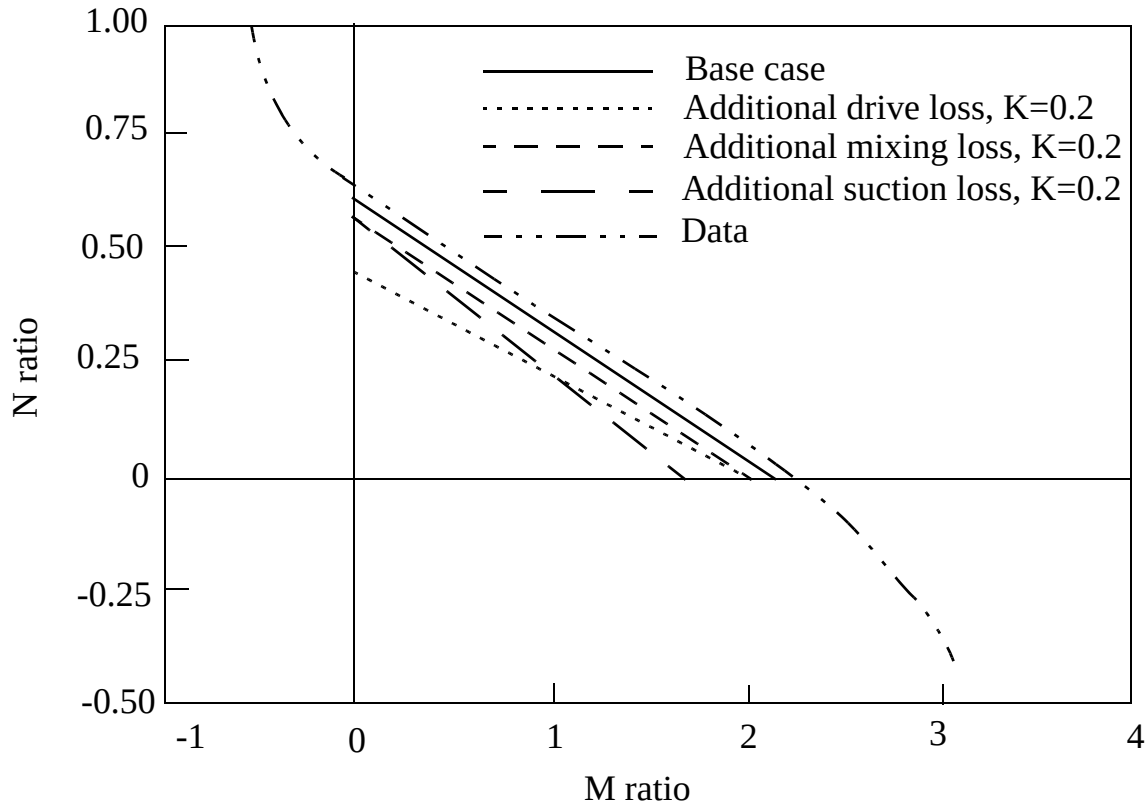


Figure 2.4-6 Jet pump model design.

2.4.10 Valves

In RELAP5-3D©, eight valves are modeled that are of six types. The types of valves provided are check valves, trip valves, inertial swing check valves, motor valves, servo valves, and relief valves. A single model for each type of valve is provided except for the check valves. For check valves, three models are provided, each of which has different hysteresis effects with respect to the opening/closing forces. Of the six types of valves, the check valves and trip valves are modeled as instantaneous *on/off* switches. That is, if the opening conditions are met, then the valve is instantly and fully opened; if the closing conditions are met, the valve is instantly and fully closed. The remaining four types of valves are more realistic models in that opening/closing rates are considered. In the case of the inertial swing check valve and the relief valve, the dynamic behavior of the valve mechanism is modeled.

Fundamentally, a valve is used to regulate flow by varying the flow area at a specific location in a flow stream. Hence, in the RELAP5-3D© scheme, a valve is modeled as a junction component that gives the user a means of varying a junction flow area as a function of time and/or hydrodynamic properties. Valve action is modeled explicitly and therefore lags the hydrodynamic calculational results by one time step. A closed valve is treated as a time dependent junction with no flow. In order for the user to more fully use the valve models, some characteristics and recommendations for each valve are discussed in the following subsections.

2.4.10.1 Check Valves. Check valves are on/off switches, and the on/off action is determined by the formulation presented in Volume 1 of this manual. In turn, it is the characteristic of these formulations that determines the kind of behavior modeled by each type of check valve.

2.4.10.1.1 Static Pressure Controlled Check Valve--The check valve logic in Section 3 of Volume I describes the operation of a static pressure controlled check valve. If the equation is positive, the valve is instantaneously and fully opened, and the switch is *on*. If the equation is negative, the valve is instantaneously and fully closed, and the switch is *off*. If the equation is zero, an equilibrium condition exists, and no action is taken to change the existing state of the valve. Hence, in terms of pressure differential, there is no hysteresis. However, because the valve model is evaluated explicitly in the numerical scheme, the actual valve actuation will lag one time step behind the pressure differential. In terms of fluid flowing through the valve in a transient state, it is obvious that if the valve is closed and then opens, the flow rate is zero; but when a pressure differential closes the valve, the flow rate may be either positive, negative, or zero. Hence, with respect to flow, a hysteresis effect will be observed. Also, in the strictest sense, this type of valve is not a check valve, since the model allows reverse flow.

2.4.10.1.2 Flow Controlled Check Valve--Section 3 in Volume 1 shows the model of a check valve in the strictest sense in that flow is allowed only in the positive or forward direction, and the model is again designed to perform as an *on/off* switch. If the valve is closed, it will remain closed until the static pressure differential becomes positive, at which time the valve is instantaneously and fully opened, and the switch is *on*. Once the valve is opened, it will remain open until flow is negative or reversed, regardless of the pressure differential. Hence, with respect to pressure differential, a hysteresis effect may be observed. With respect to flow, it defines a negligible hysteresis effect, since flow is zero when the valve opens, and closes if flow becomes infinitesimally negative. However, since valve actuation lags one time step behind the pressure and flow calculation, a significant flow reversal may be calculated before the valve model completes a closed condition.

2.4.10.1.3 Dynamic Pressure Controlled Check Valve--Section 3 in Volume 1 shows the model of a dynamic pressure-actuated valve also designed to perform as an *on/off* switch. If the valve is closed, there is no flow through the valve, hence the valve must be opened by static pressure differential. For this condition, the valve is opened instantaneously and fully, and the switch is *on*. Once the valve is opened, the fluid is accelerated, flow through the valve begins, and the dynamic pressure aids in holding the valve open. Since the valve cannot close until the closing back pressure, PCV, exceeds the junction static and dynamic pressure, there is a hysteresis effect both with respect to the opening and closing pressure differential and with respect to the fluid flow. These hysteresis effects are also determined by the sign of PCV, as input by the user. If PCV is input as positive, positive or forward flow through the valve will be allowed, and negative or reverse flow will be restricted. In this sense, the valve performs as a check valve. However, if PCV is input as negative, it will aid in opening the valve, and significant negative or reverse flow must occur before the valve will close. In this sense, the valve will not perform as a check valve. In addition, valve actuation lags one time step behind the pressure and flow calculations in the numerical scheme.

2.4.10.1.4 Check Valve Closing Back Pressure Term PCV--In Section 3 in Volume 1, the term PCV is used; in the input requirements, this term is designated as the *closing back pressure*. However, to be precise, PCV is a constant representing an actuation set point. If positive, PCV behaves as a back pressure acting to close the valve. In both the static and dynamic pressure-controlled valves, PCV acts both

as an actuation set point for opening a closed valve and as a closing force for closing an open valve. For the flow-controlled valve, the back pressure acts only as an actuation set point for opening a closed valve.

2.4.10.2 Trip Valve. The trip valve is also an on/off switch that is controlled by a trip such that when the trip is *true* (i.e., *on*), the valve is *on* (i.e., instantly and fully open). Conversely, when the trip is *false* (i.e., *off*), the valve is *off* (i.e., instantly and fully closed).

Since trips are highly general functions in RELAP5-3D© and since trips can be *driven* by control systems, the on/off function of a trip valve can be designed in any manner the user desires. The user should remember, however, that trips, control systems, and valves are explicit functions in RELAP5-3D© and hence lag the calculational results by one time step.

2.4.10.3 Inertial Swing Check Valve. The inertial valve model closely approximates the behavior of a real flapper-type check valve. To direct the model to neglect flapper mass and inertial effects, the user simply inputs the flapper mass and moment of inertia as small numbers. Flapper open angles are positive in the positive junction flow direction. The code assumes that gravity always acts in the vertically downward direction, so that gravity can act to either open or close the valve, depending on the implied junction direction. The minimum flapper angle must be greater than or equal to zero.

2.4.10.4 Motor and Servo Valves. The interaction of both motor and servo valves with fluid flow are identical but the means of positioning the valves are different. Both valves use a normalized stem position to position the valve. The normalized stem position ranges between 0.0 for the closed position to 1.0 for the fully open position. The flow area corresponding to a normalized stem position is determined from the normalized flow area, which also ranges from 0.0 for fully closed to 1.0 for fully open. A general table can be used to describe the normalized flow area for a given normalized stem position. If the general table is not used, the normalized flow area is set to equal the normalized stem position. Two models are provided to effect flow changes based on valve flow area. If the abrupt area change flag is set, the abrupt area change model is used to determine flow losses, and the valve flow area is treated as the orifice area in the abrupt area change model. If the abrupt area change flag is not set, a CSUBV table must be entered. This table contains forward and reverse flow coefficients as a function of normalized valve area. The model using CSUBV coefficients should usually be used when the valve is designed for regulating flow.

The motor valve assumes that the valve stem is positioned by a motor. The valve position can be stationary, or the valve can be moving at a constant rate in the opening or closing direction. The rate is a user-input quantity in terms of normalized position change per second. The motor is controlled by an open trip and a closed trip. The valve stem position is stationary when both trips are false; when the open trip is true, the valve stem moves according to the valve change rate in the opening direction; when the close trip is true, the valve stem moves according to the valve change rate in the closing direction. The code terminates if both trips are simultaneously set true. Section 4.1.4 shows trip logic for the open trip that could be used to position a valve to regulate flow such that an upstream pressure is held within a set range.

The servo valve uses the value of a control variable to indicate the normalized valve stem position. A typical application would be regulating vapor/gas flow to the turbine to maintain a desired quantity such as primary system temperature or secondary side steam generator pressure. The control system, perhaps using

a STEAMCTL (specialized proportional-integral controller), would compare the current value of the primary temperature or steam generator pressure to the desired value, and from the difference of the values compute an appropriate valve position. The servo valve, using the control output, would position the valve and thus regulate the flow.

2.4.10.5 Relief Valve. A scheme was designed to input the terms required to define a typical relief valve geometry and dynamic parameters. This scheme is consistent with the RELAP5-3D© input philosophy in that extensive checking is performed during input processing, and error flags are set to terminate the problem if input errors are encountered. Error messages are also printed to inform the user that the data entered were in error. The specific input description is detailed in Appendix A.

2.4.11 Separator

Figure 2.4-7 contains a schematic showing the typical nodalization used for a separator and the adjoining bypass and downcomer regions. If there is any possibility of recirculation flow through a bypass region, we recommend that this flow path be included. In general, there will be a mixture level at some location in the downcomer volumes.

The RELAP5-3D© ideal simple separator model can be looked on in the following way, that is, when the input maximum volume fractions (VGMAX and VFMAX) are both 1.0. The flux through the liquid outlet is all liquid when the volume fraction of liquid is above a critical value. Below that critical value, a mixture of vapor/gas and liquid are fluxed out the liquid outlet. A similar process occurs for the vapor/gas outlet. When the volume fraction of vapor/gas is above a critical value, only vapor/gas is fluxed through the vapor/gas outlet. When the volume fraction of vapor/gas is below this critical value, a mixture of vapor/gas and liquid is fluxed through the vapor/gas outlet. The critical values are given names VUNDER for the liquid outlet, and VOVER for the vapor/gas outlet. This behavior can be shown in **Figure 2.4-8**.

The volume on the bottom furnishes a mixture of vapor/gas and liquid to the separator. For the separator liquid outlet, if the liquid level drops below the outlet baffle on the left, vapor/gas will also come out the junction. For the separator vapor/gas outlet, if the liquid level rises above the outlet baffle on the right, liquid will also come out the junction. Thus the critical values of VUNDER and VOVER are given by the formulas

$$\text{VUNDER} = \frac{A_{fj}}{A_t} \text{ and VOVER} = \frac{A_{gj}}{A_t} \tag{2.4-37}$$

where

A_{fj} = the area open to the liquid

A_t = the total area

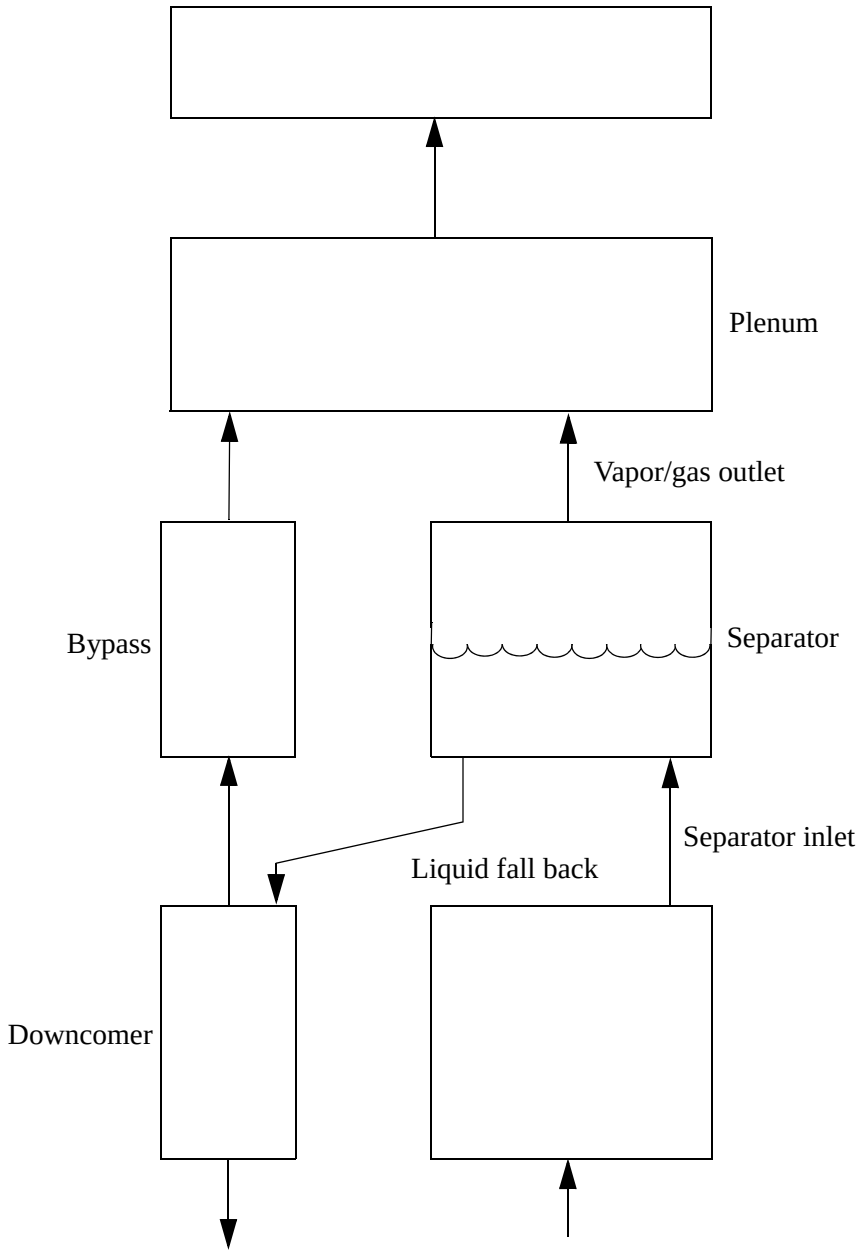


Figure 2.4-7 Schematic of separator.

A_{gj} = the area open to the vapor/gas.

When the liquid level drops below the baffle on the left, the volume fraction of vapor/gas that is fluxed through the liquid outlet junction is a linear function of the liquid level height. A similar relationship is used for the vapor/gas outlet when the liquid level rises above the baffle on the right. These linear relationships are such that if the separator is empty of liquid, then pure vapor/gas comes out the

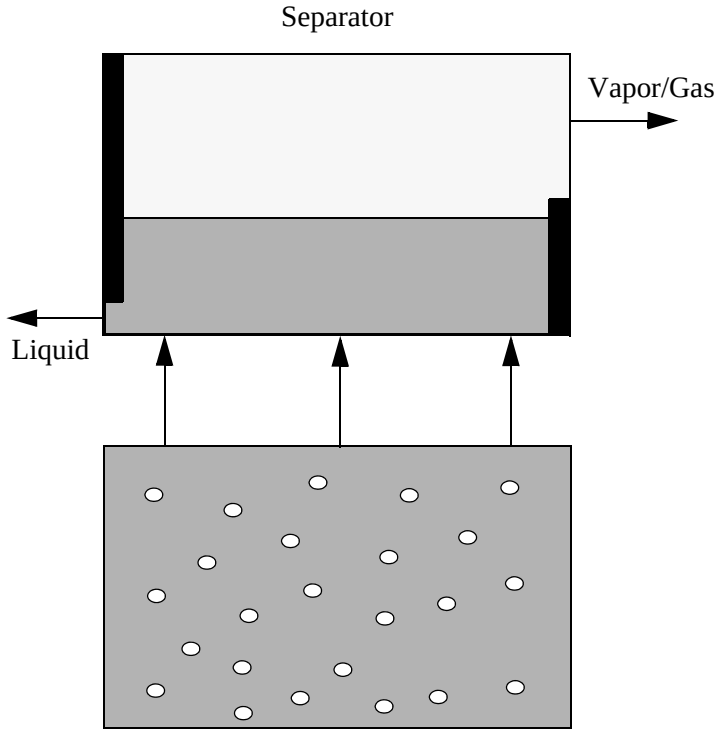


Figure 2.4-8 Physical picture of a separator.

liquid outlet, and conversely, if the separator is full of liquid, then pure liquid comes out the vapor/gas outlet. The behavior of the separator can now be characterized by **Figure 2.4-9** and **Figure 2.4-10** where the y-axis shows the volume fraction fluxed out the outlet (liquid or vapor/gas) and the x-axis shows the volume fraction (liquid or vapor/gas) in the separator volume.

2.4.11.1 Input Requirements. The input for a SEPARATR component is the same as that for a BRANCH component, with the following modifications:

1. For a BRANCH component, the junctions connected to the branch can be input with the branch or as separate components. For a SEPARATR, the three junctions, representing the vapor/gas outlet, liquid fall back, and separator inlet, must be input with the SEPARATR component, i.e., NJ = 3.

2. The three junction card sequences must be numbered as follows: Cards CCC1101 and CCC1201 represent the vapor/gas outlet junction, Cards CCC2101 and CCC2201 represent the liquid fall back junction, and Cards CCC3101 and CCC3201 represent the separator inlet junction.

3. The FROM connection for the vapor/gas outlet junction must refer to the x-coordinate outlet face of the separator for the old format (CCC010000) and must refer to any of the 6

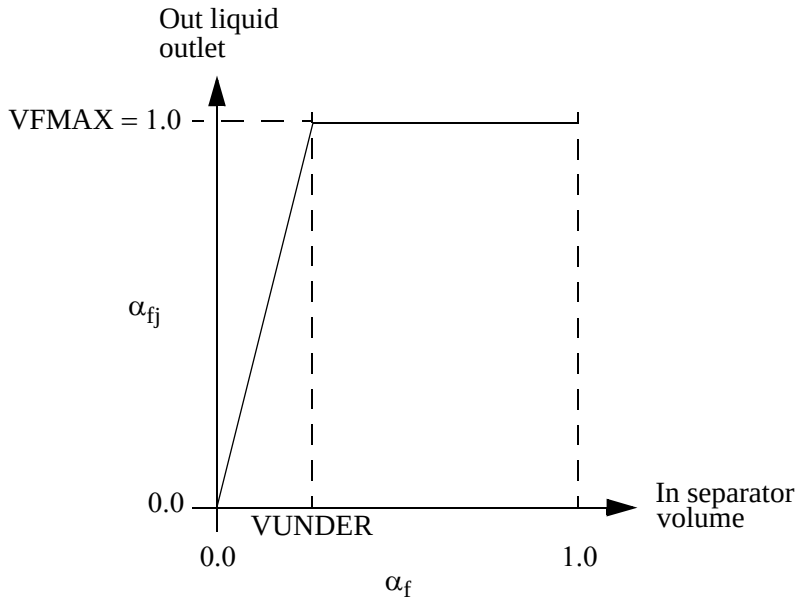


Figure 2.4-9 Separator volume fraction of liquid fluxed out the liquid outlet.

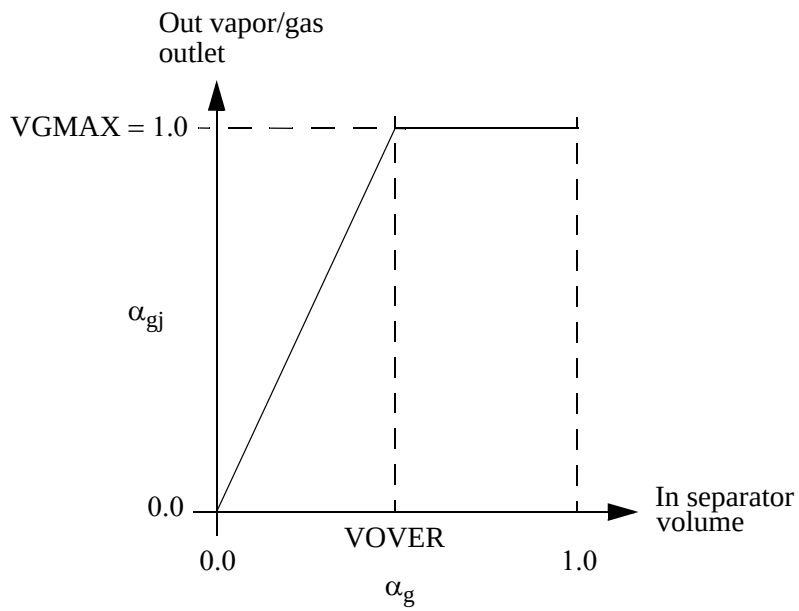


Figure 2.4-10 Separator volume fraction of vapor/gas fluxed out the vapor/gas outlet.

volume faces of the separator except the x-coordinate inlet face for the expanded format (CCC010002 through CCC010006). The FROM connection for the liquid return junction must refer to the x-coordinate inlet face of the separator for the old format (CCC000000) and must refer to any of the 6 volume faces of the separator except the x-coordinate outlet face for the expanded format (CCC010001, CCC010003 through CCC010006). The separator inlet junction must be connected to the x-coordinate inlet face of the separator for the old format (CCC000000) and must be connected to any of the 6 volume faces of the separator except the x-coordinate outlet face for the expanded format (CCC010001, CCC010003 through CCC010006).

4. The type of separator desired is specified on Card CCC0002. If the mechanistic separator or dryer options are chosen, additional input data may be entered on the CCC050X cards for the mechanistic separator model and on the CCC0600 card for the dryer model. Default data are provided for both the mechanistic separator and dryer models.

5. A word, W7(R), is added to the SEPARATR component junction geometry Cards CCCN101 through CCCN109 for the simple separator option. For the vapor/gas outlet, Word W7(R) specifies VOVER. For the liquid fall back junction, Word W7(R) specifies VUNDER. No input should be entered for Word W7(R) on the separator inlet junction.

6. A word, W3(R), is on the SEPARATOR component junction maximum volume fraction Cards CCCN901 through CCCN909 for the simple separator option. For the vapor/gas outlet, Word W3(R) specifies VGAMX. For the liquid fallback junction, Word W3(R) specifies VFMAX. No input should be entered for Word W3(R) on the separator input junction.

2.4.11.2 Recommendations for the Simple Separator Option. The smooth or abrupt junction option can be used for the separator. Separators in general have many internal surfaces that lead to flow resistances above that of an open region. For this reason, additional energy loss coefficients may be required at the appropriate separator junctions. These should be obtained from handbook values or adjusted to match a known pressure drop across the separator. In some cases, it is necessary to use large loss coefficients (~100) in order to remove void oscillations in the separator volume. In addition, we recommend that choking be turned off for all three junctions. The nonhomogeneous options should be used for the vapor outlet and liquid fall back junctions.

An important parameter that influences the operation of any heat exchanger/separator combination is the equivalent mixture level in the downcomer region. This level is primarily determined by the rate of flow in the liquid return junction, which in turn is affected by the liquid level in the separator and the vapor flow out of the separator. The liquid return flow and liquid level in the separator are affected by the user-input volume fraction limits VOVER and VUNDER and the user input maximum volume fractions VGMAX and VFMAX. Because of the simple black-box nature of the separator, these user inputs should be adjusted to obtain the desired operating mixture level in the downcomer region. The default volume fraction limits (VOVER = 0.5 and VUNDER = 0.15) and maximum volume fractions (VGMAX = 1.0 and VFMAX = 1.0) are intended to be preliminary.

The black-box nature of the separator, along with the use of VOVER, VUNDER, VGMAX, and VFMAX may result in some changes to the user-input initial conditions. If the user inputs a phasic mass flow rate for both the vapor/gas outlet and liquid fall back junctions, the code will in many cases alter the phasic mass flow rates so that they no longer match those inputted. This is due to the use of the piecewise linear donor junction volume fractions used (see Volume 1 of this manual). Depending on the relations of α_{gK}, VOVER, and VGMAX as well as α_{fK}, VUNDER, and VFMAX, it may be necessary to scale back the phasic mass flow rates to achieve the desired input phasic mass flow rates. Once the transient calculation begins, the phasic mass flow rates and phasic volume fractions will most likely change from the initial value and some adjustment of VOVER, VUNDER, VGMAX, and VFMAX may be required.

The final recommendation concerns the use of a bypass volume. If there is any possibility of a recirculation flow through a bypass-like region, we recommend such a flow path be included. The inclusion of such a flow path has generally improved the performance predictions.

2.4.11.3 Recommendations for the Mechanistic Separator and Dryer Options. The mechanistic separator and dryer models are new in RELAP5-3D©. They are intended to model the centrifugal separator and chevron dryer components in a BWR reactor. Until a base of user experience is obtained, we recommend default input data for the models be used. The user should explicitly model the separator standpipe as a separate volume or set of volumes because the separator component volume is intended to model the volume within the separator barrel and discharge passages. Likewise, the dryer volume should encompass the physical volume inside the dryer skirt between the elevations of the dryer inlet and outlet elevations. The separator inlet quality can be adjusted to the desired operating point by modifying the form loss coefficient in the separator liquid discharge junction. The liquid discharge line from a dryer to the downcomer should be modeled as a separate volume or set of volumes so that the liquid removed by the dryer may be injected into the downcomer at the correct elevation below the two-phase mixture level in the downcomer. The void fraction within the dryer component can also be adjusted by use of the liquid discharge junction form loss coefficient. The separator component may represent any number of physical separators. It is required that the geometry (i.e., volume and junction flow areas) of the separator component be the volume and flow areas of all of the physical separators represented by the RELAP5-3D© separator components and that the number of separators represented by the RELAP5-3D© separator component be specified in Word 2 on Card CCC0002 in the separator component input data.

2.4.12 Turbine

A turbine is a device that converts the energy contained in a high-pressure fluid to mechanical work. Four different stage group types can be implemented: (a) a two-row impulse stage group, which is normally only used as the first stage of a turbine for governing purposes; (b) a general impulse-reaction stage group with a fixed reaction fraction needed as input; (c) a constant efficiency stage group to be used for very simple modeling or as a preliminary component during the model design process; and (d) a variable efficiency turbine that allows the user to specify turbine efficiency as a function of normalized speed and load (shaft power). A simple efficiency formula for each of the turbine types is given in Volume I, where all the terms are defined.

The mean stage radius needed in the efficiency formulas may not be known from the actual turbine design diagrams. We recommend the mean stage radius, R, be obtained from the efficiency formulas. If the turbine model is used with a constant efficiency factor, the stage radius is not needed (except for startup) and 1.0 can be entered. If the turbine stage is a general impulse-reaction stage, then the maximum efficiency, η_o, is obtained when

$$\frac{v_t}{v} = \frac{0.5}{1-r} \quad . \tag{2.4-38}$$

Using $v_t = R\omega$ and the input values v, r, and ω at the design operating point, Equation (2.4-38) gives for R,

$$R = \frac{0.5v}{\omega(1-r)} \quad . \tag{2.4-39}$$

This is the recommended mean stage radius that is consistent with the assumed efficiency formula. For a two-row impulse stage, the maximum efficiency occurs when

$$\frac{v_t}{v} = 0.25 \quad . \tag{2.4-40}$$

Expressing v_t as $R\omega$ gives

$$R = \frac{0.25v}{\omega} \tag{2.4-41}$$

as the mean stage radius consistent with the efficiency formula.

Turbine frictional torque τ_{fr} is modeled as a five-term function of the turbine rotational velocity and is given by

$$\tau_{fr} = \pm\left(\tau_{fr0} + \tau_{fr1}\left|\frac{\omega}{\omega_R}\right|^{x1} + \tau_{fr2}\left|\frac{\omega}{\omega_R}\right|^{x2} + \tau_{fr3}\left|\frac{\omega}{\omega_R}\right|^{x3} + f_i\omega\right) \tag{2.4-42}$$

where ω is the turbine rotational velocity; ω_R is the rated turbine rotational velocity; and τ_{fr0}, τ_{fr1}, τ_{fr2}, τ_{fr3}, x1, x2, x3, and f_i are input data. The turbine frictional torque is negative if $\frac{\omega}{\omega_R} > 0$, and it is positive if $\frac{\omega}{\omega_R} < 0$.

The moment of inertia, I_t, of the turbine may be entered as a constant or as a cubic function that depends on the speed ratio; that is

$$I_t = I_{tn} \qquad \text{for } \left|\frac{\omega}{\omega_R}\right| < S_{TI} \tag{2.4-43}$$

and

$$I_t = I_{t0} + I_{t1}\left|\frac{\omega}{\omega_R}\right| + I_{t2}\left|\frac{\omega}{\omega_R}\right|^2 + I_{t3}\left|\frac{\omega}{\omega_R}\right|^3 \qquad \text{for } \left|\frac{\omega}{\omega_R}\right| \geq S_{TI} \tag{2.4-44}$$

where I_{tn}, I_{t0}, I_{t1}, I_{t2}, I_{t3}, and S_{TI} (turbine inertia critical speed ratio) are input data.

For a TURBINE component, the primary fluid inlet junction must be input with the TURBINE component as the first junction. If a fluid extraction (bleed) junction is desired, it must be input with the TURBINE component as the second junction. Thus, NJ must be either 1 or 2. Cards CCC1101 and CCC1201 represent the inlet junction, and Cards CCC2101 and CCC2201 represent the extraction bleed junction (if desired). The TO connection for the inlet junction must refer to the inlet of the TURBINE (old format is CCC000000, and expanded format is CCC010001).

Horizontal stratification effects are not modeled in the TURBINE component. Thus, the horizontal stratification flag must be turned off (v = 0). If several TURBINE components are in series, the choking flag should be left on (c = 0) for the first component but turned off for the other components (c = 1). The area changes along the turbine axis are gradual, so the smooth junction option should be used at both the inlet and outlet junctions. No special modeling has been included for slip effects, nor are there any data that could be used as a guide. Thus, the inlet and outlet junctions must be input as homogeneous junctions (h = 1 or 2). If an extraction (bleed) junction is present, it should be a crossflow junction.

The standard wall friction calculation is based upon the wetted perimeter. Because of all the internal blading surfaces, the wall friction based upon the volume geometry will not give a meaningful calculation. The turbine volume must be input using the zero wall friction option.

For some off-design cases, choking can take place at the nozzle and stator throats in a turbine. The junction velocities must represent the maximum nozzle velocities if the critical flow model is to be used.

Hence, the junction areas used in the TURBINE component should represent the average nozzle throat or minimum area for the stage group if proper critical flow modeling is desired.

It is much easier to initialize a turbine component for steady-state conditions using a stand-alone model rather than a system model. The boundary conditions should be applied using time dependent volumes and single junctions on either end of the model. If a time dependent junction is applied at the inlet, the pressure in the time dependent volume will generally not match the pressure in the adjacent control volume at steady state. Any discrepancy in pressure will cause variations in the fluid density and velocity that may persist for several volumes downstream. The use of multiple frictionless volumes upstream and downstream of the turbine components can be used to verify that the calculated conditions are steady with respect to distance.

The constant efficiency option should be used in the initial stand-alone calculations. Once the desired steady state has been obtained, the more mechanistic turbine models, such as the general impulse-reaction stage group and the two-row impulse stage group, should be applied. An optional user-specified efficiency may also be used (specify efficiency as a function of normalized speed and load); this is called a Type 3 turbine in the input manual (see Volume II, Appendix A). A multiplier (from a user-specified control variable) on this optional user-specified efficiency may be also be specified for a Type 3 turbine (see Volume II, Appendix A). The mean stage radius should be calculated using Equations (2.4-39) and (2.4-41) from the rotational speed and the junction velocity, which is obtained from the steady-state calculation with constant efficiency.

Stage pressure, enthalpy, flow rate, and efficiency are generally available from turbine cycle diagrams. The form loss coefficients at the inlet junction to each stage should be calculated as follows.

The momentum equation for the input junction of each stage (assuming zero wall friction), is

$$(1-\eta)(P_1 - P_2) = 0.5K\dot{\rho}v_{j1}^2 \tag{2.4-45}$$

where the subscript j1 refers to the junction under consideration, K is the form loss coefficient and the dot above the density indicated that it is donored from the upstream volume. Solving for K yields

$$K = \frac{(1-\eta)(P_1 - P_2)}{0.5\dot{\rho}v_{j1}^2} \tag{2.4-46}$$

Several turbine stages may be combined into one turbine component if the efficiency of each stage is similar. However, if the pressure drop across the RELAP5-3D© component is too large, the power extracted from the fluid may not match the value from the turbine cycle diagram because Equation (3.5-87) in Volume I of this manual is approximate. The approximation improves as the pressure drop across the stage decreases and the upstream and downstream fluid densities converge. Equation (3.5-87) in Volume I

of this manual is generally accurate to within a few percent even for relatively large pressure changes. If this level of accuracy is not sufficient, the approximation will improve if each stage is modeled explicitly.

Note that turbines are usually designed to run with large velocities in the nozzles. The turbine may be the component that gives the maximum Courant number in the system. For this reason, the turbine component may limit the time step size. This can be mitigated if the turbine volumes are used with an exaggerated length. This will not affect any steady-state results, but it will give slightly inaccurate storage terms during a transient. The transient storage terms are generally small, so this should not be a problem.

2.4.13 Accumulator

An accumulator is a *lumped* parameter component modeled by two methods. First, the component is considered to be an accumulator as long as some of the initial liquid remains in the component. In this state, the accumulator is modeled using the special formulations discussed in Volume I of this manual, which includes models for the accumulator tank, tank wall, surge line, and outlet check valve junction. However, second, when the accumulator empties of liquid, the code automatically converts the component to an equivalent single-volume with a single outlet junction and continues calculations using the normal solution algorithms. In performing this conversion, the accumulator wall heat transfer model is retained but the volume flow area (A), hydraulic diameter (D_h), and elevation change (Δz) are reset to

$$A = \frac{V}{\Delta x_{TK} + \Delta x_L} \tag{2.4-47}$$

$$D_h = \frac{4V}{\pi(D_{TK}\Delta x_{TK} + D_L\Delta x_L)} \tag{2.4-48}$$

$$\Delta z = \Delta z_{TK} + \Delta z_L \tag{2.4-49}$$

respectively. In these equations, subscript TK is for the tank, and subscript L is for the surgeline/standpipe. In addition, the accumulator mass transfer model converts to the normal mass transfer model scheme.

In setting up an accumulator component, the user must remember that at the input processing level, the code assumes that the accumulator is initially *off*; that is, flow through the accumulator junction is zero. It is further assumed that the standpipe/surgeline is initially full of liquid and that the tank liquid level is as defined by the user. These assumptions are also true for RESTART runs if the user renodalizes the accumulator. Hence, the user must be careful to define the initial accumulator pressure lower than the injection point pressure, including elevation head effects. The tank geometry may be either cylindrical (see **Figure 2.4-11**) or spherical (see **Figure 2.4-12**). In the input description (Appendix A), the standpipe/surgeline inlet refers to the end of the pipe inside the tank itself (see **Figure 2.4-13**). Also, the noncondensable (nitrogen) used in the accumulator is that defined for the entire system being modeled.

Hence, the user must be sure to input the correct noncondensable name (nitrogen) on Card 110, as discussed in Appendix A.

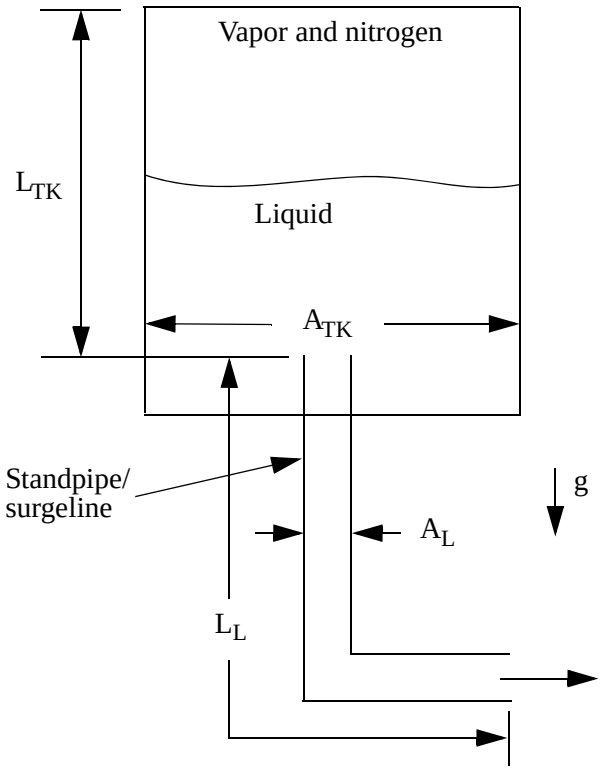


Figure 2.4-11 Schematic of a cylindrical accumulator.

No other junctions (except the accumulator junction) should be connected to an accumulator volume. There are 4 possible accumulator configurations, as shown in **Figure 2.4-14**. The inclination angle (W5) and elevation change (W6) in Cards CCC0101 through CCC0109 can be either positive or negative, but both must have the same sign. The elevation drop of the surgeline and standpipe (W4) in Card CCC2200 is positive for a decrease in elevation from the standpipe/surgeline inlet to the injection point, and it is negative for an increase in elevation from the standpipe/surgeline inlet to the injection point.

2.4.14 Annulus

The annulus component is identical to a pipe component (Section 2.4.6), except the annulus component must be vertical and the annular-mist flow regime is different. If the user specifies this component, all the liquid is in the film and none is in the drops when the flow regime is annular-mist. The annulus component should be used to model a vertical annular region (i.e., reactor vessel downcomer or annular downcomer region in a U-tube steam generator).

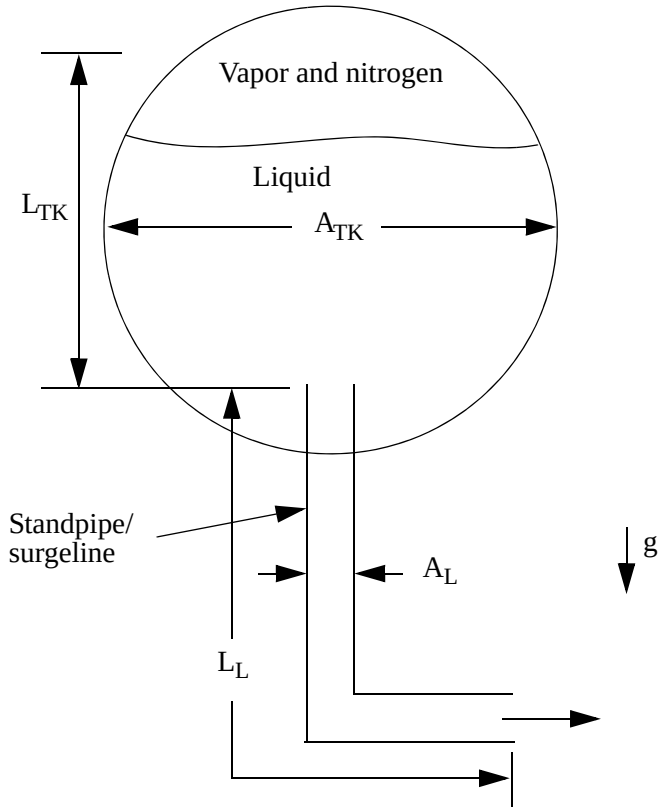


Figure 2.4-12 Schematic of a spherical accumulator.

2.4.15 ECC Mixer

An ECC mixing component is a specialized branch that requires three junctions with a certain numbering order. The physical extent of the ECCMIX is a length of the cold leg pipe centered around the position of the ECC injection location. The length of this segment should be about three times the inside diameter of the cold leg pipe. Junction No. 1 is the ECC connection; junction No. 2 is the cold leg cross-section through which flow enters this component in normal reactor operation; and junction No. 3 is the one that leads to the reactor vessel. The geometrical description of the ECCMIX component is very similar to that of the JETMIXER component, except for the specification of an angle for the ECC pipe connection. The modeling details of the ECCMIX component are given in Section A7.7 of the input requirements in Appendix A of this volume. The ECCMIX component calculations are evoked only if there is subcooled ECC injection and if there is any vapor to be condensed in that component. Otherwise, the ECCMIX component is treated as an ordinary BRANCH component.

2.4.16 Multi-Dimensional Component

The multi-dimensional component (indicated by MULTID in the input cards) defines a one-, two-, or three-dimensional array of volumes and the internal junctions connecting the volumes. The

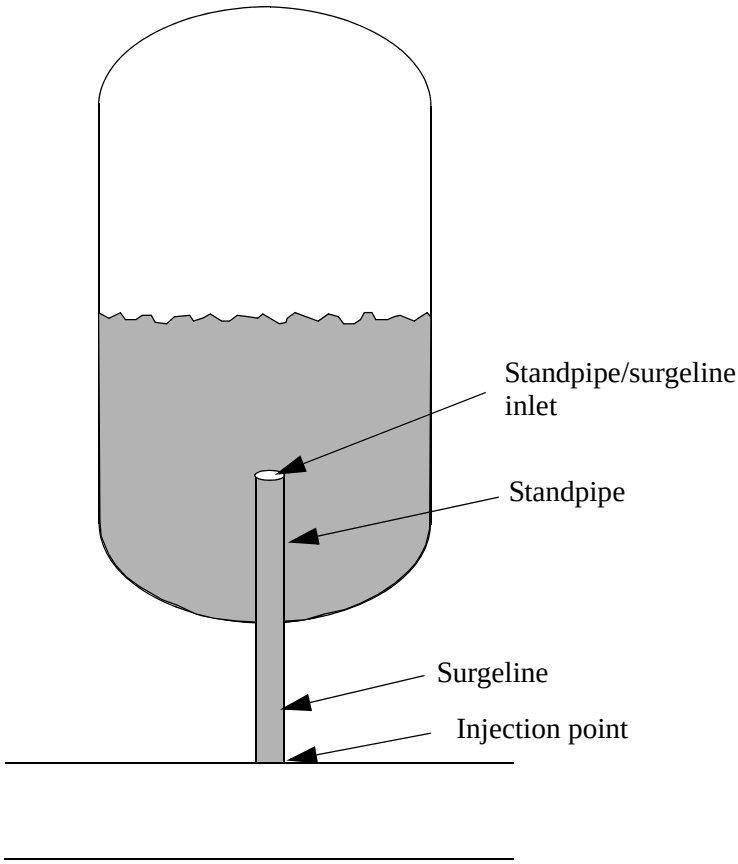


Figure 2.4-13 Schematic of an accumulator showing standpipe/surgeline inlet.

multi-dimensional component is described as a three-dimensional component but can be reduced to two or one dimensions by defining only one interval in the appropriate coordinate directions. The geometry can be either Cartesian (x,y,z) or cylindrical (r,θ,z). In cylindrical geometry, the r-direction can start at zero or nonzero, and θ can cover 360 degrees (i.e., full circle, annulus) or can cover less than 360 degrees (i.e., semicircle, wedge). Input checking uses 360 $\pm$ 0.0005 degrees for the region that represents 360 degrees.

An orthogonal, three-dimensional grid is defined by mesh interval input data in each of the three coordinate directions. The edges of the hydrodynamic volumes are defined by the grid lines.

The 3-D component is designed primarily for reactor applications, particularly in the vessel (i.e., core, downcomer) and steam generator. These reactor components have solid structures in the fluid path (i.e., core, steam generator) or have a short length in the radial direction (i.e., downcomer) that result in the form loss, wall friction, and interphase friction models being the primary source terms in the momentum equations. For these applications, the viscous stress and turbulence terms are not as important and are not included in the RELAP5-3D© 3-D model. Since these terms are not present in the code at this time, the RELAP5-3D© 3-D model should not be used to model large open tanks.

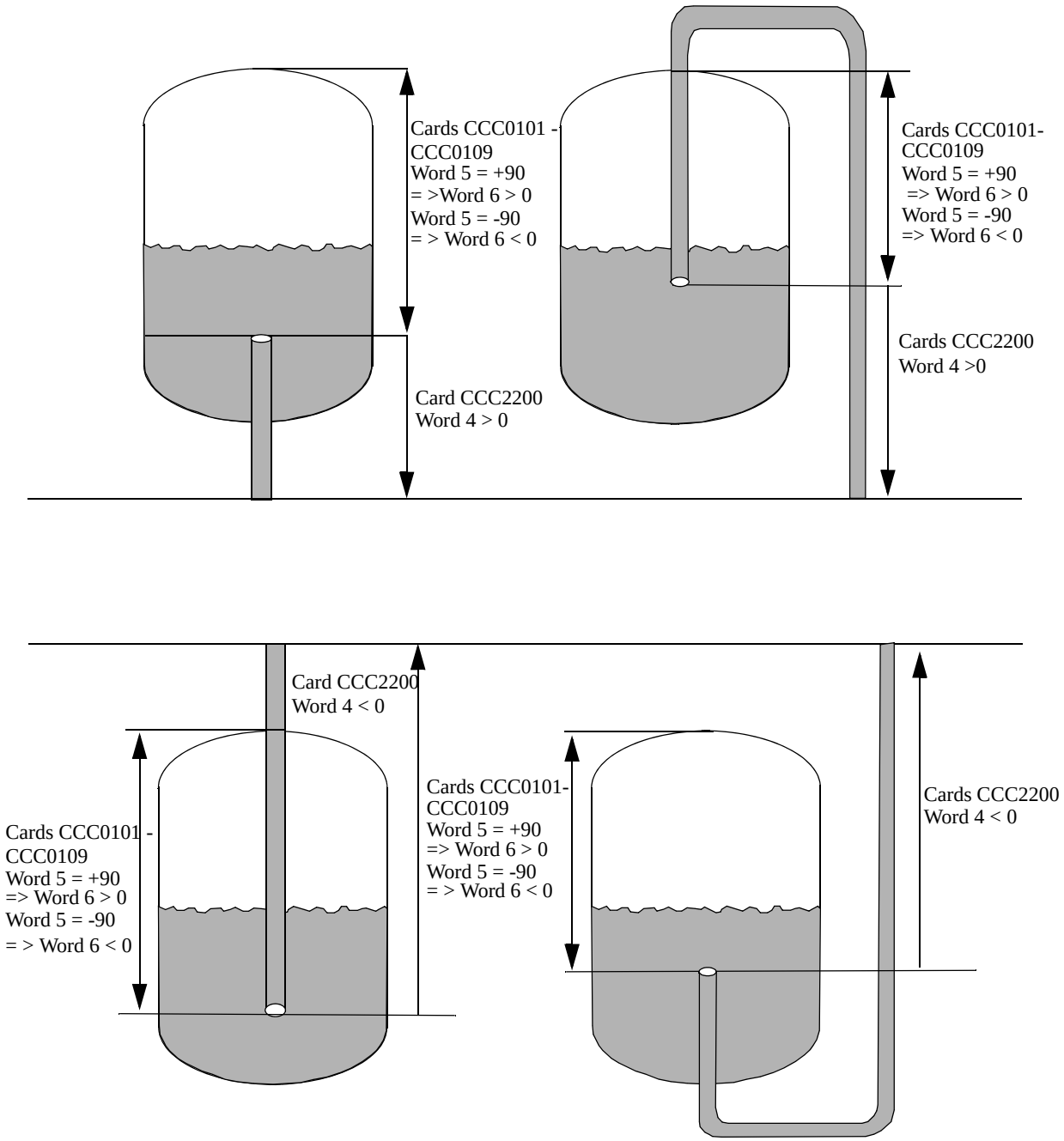


Figure 2.4-14 Possible accumulator configurations.

The 3-D component can be connected to 1-D components externally via either a normal junction or a crossflow junction, depending on the actual flow paths. The 1-D to 3-D external junction connection to an external 3-D face should be restricted to 1 junction per external face. The 3-D component can also be connected to 3-D components externally via either a normal junction or a crossflow junction. The 3-D to 3-D connection is restricted to the same direction (i.e., radial to radial, axial to axial, etc.).

The 3-D cylindrical component can be modeled either as a solid cylinder or a hollow cylinder. It can also be modeled as a cylindrical wedge. All of these geometries are implemented.

The 3-D cylindrical component can be used to model a pipe. When it is used to model a horizontal pipe that is connected to 1-D components, the 3-D cylindrical component needs to be rotated properly about the local z-axis to ensure there is no artificial gravity-induced flow due to the connections to the 1-D component.

The volume factors and junction areas factors must be specified explicitly for the 3-D component. A totally blocked internal 3-D junction (i.e., the junction area factor is zero) is treated as a time dependent junction with no flow. In the junction initial condition cards, the junction face number must be specified explicitly.

For pure radial, frictionless flow in and out of a 3-D solid cylinder, the pressure profile within the 3-D component is not sensitive to the number of radial nodes, i.e., a 3-ring model produced as good pressure results as an 8-ring model. The user does not have to increase radial nodes to increase accuracy.

For a multid component (no drops option), as with the annulus component, all the liquid is in the film and none is in the drops when the flow regime is annular-mist.

It is recommended that the momentum flux be omitted in the cross direction (perpendicular to the main upward flow direction) if either of the following two modeling approaches are used:

- 1-D components (i.e., pipes, etc.) with cross junctions
- MULTID component, 1-D option (1-D momentum equations per Card CCC0001, Word 7 = 1 or 3).

2.4.17 Pressurizer

The pressurizer component is identical to a pipe component (Section 2.4.6), except the user must input the surgeline connection junction identifier. The surgeline connection junction is not part of the pressurizer component. There are optional input for a constant interfacial heat transfer coefficient for liquid in the vertically stratified flow regime and the mixture level tracking flow regime, a constant interfacial heat transfer coefficient for vapor/gas in the vertically stratified flow regime and the level tracking flow regime, a multiplier on the film thickness in the annular-mist flow regime, and a multiplier on the interfacial heat transfer coefficients for both liquid and vapor/gas in the vertically stratified flow regime and the level tracking flow regime. There is optional input to specify the drop size in the

annular-mist flow regime and optional input (sprayline connection junction identifier) to activate a spray induced, enhanced pool condensation model. The sprayline connection junction is not part of the pressurizer component. It is recommended that the user activate the thermal stratification model in the volumes in the bottom of the pressurizer to prevent the mixing of cold liquid entering the bottom of the pressurizer through insurge events with the hotter upper liquid layers of the liquid pool where interfacial condensation is occurring.

2.4.18 Feedwater Heater

The feedwater heater is a specialized branch component. It represents a horizontally oriented tube-in-shell heat exchanger the heats feedwater passing through the shell in a tube bundle by condensing steam on the shell side (**Figure 2.4-15**). Input data are used to define the steam inlet and condensate drain junctions and an optional condensate inlet junction. Optional input data can be entered to define the relationship between the void fraction on the shell side and the corresponding water level. One or more pipe components are used to represent the tube bundle. For the heat slabs representing the tube bundle walls, data are required to define the elevations of the top and bottom of the tube bundle relative to the bottom of the shell.

.

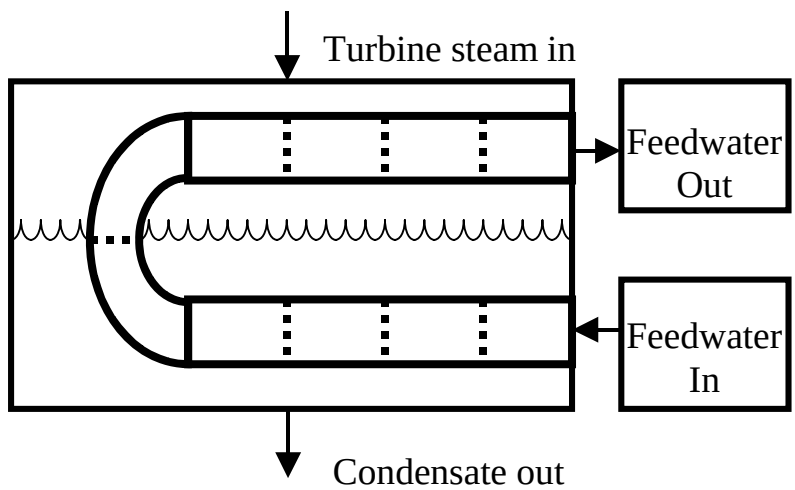


Figure 2.4-15 Schematic of a feedwater heater.

2.4.19 Compressor

The compressor component model can be separated into models for hydrodynamics, compressor-fluid interaction, and compressor driving torque. The compressor component input provides information for the hydrodynamic and compressor-fluid interaction models. A compressor can be connected to a shaft that is a specialized component within the control system. A shaft component is used when the compressor is driven by a turbine or by an electric motor with a control system to regulate speed. The compressor also may optionally include input for an electric motor to drive the compressor.

2.4.19.1 Compressor Model Description. The hydrodynamic model of a compressor component consists of one volume and one or two associated junctions. The coordinate directions of the junctions are aligned with the coordinate direction of the volume. One junction is connected to the inlet and is called the suction junction; and must be supplied with the compressor component. Optionally, a second junction can be specified with the compressor component, which is connected to the outlet and is called the discharge junction. The compressor head, torque, and angular velocity are computed using upstream junction mass flow rate, density, and sound speed. The head developed by the compressor is treated like a body force in the momentum equations for the inlet junction. With the exception of the head term, the hydrodynamic model for the compressor volume and junctions is identical to that for normal volumes and junctions.

The compressor model is similar to the pump model in RELAP5-3D©. It performs the same function on a gas as the pump performs on single-phase and two-phase fluids. It is anticipated that the compressor will usually be driven by a turbine on the same shaft, although the other capabilities presently available in the pump, i.e., the speed table, the motor torque table, and the coastdown feature are also be available in the compressor. The compressor head curve specification is formulated as pressure ratio, instead of directly as head as is done in the pump model and the compressor torque is calculated based on the compressor characteristic curves and the stage adiabatic efficiency. In addition, the compressor consists of an inlet junction and a control volume, and optionally, an outlet junction. The outlet connection can also be an ordinary single junction or branch component junction, or an other non-compressor component junction, or it can be another compressor component inlet junction. This feature allows for cascading compressor components into stages, if desired, to improve the accuracy of the representation.

2.4.19.2 Compressor Performance Modeling. Interaction of the compressor and the fluid is described by empirically developed performance maps relating compressor pressure ratio and efficiency to the mass flow rate and compressor angular (rotational) velocity. **Figure 2.4-16** is an example of a compressor performance map. The performance map is an xy representation that has a mass flow rate parameter (v) as the abscissa and a head-related parameter (in this case pressure ratio $\frac{P_2^T}{P_1^T}$) as the ordinate. Lines of constant speed (α) and lines of constant efficiency are represented parametrically on the compressor performance map.

The operating range of a compressor is the region between the surge point and the choke point. Compressor surge is characterized by some aerodynamic instability, caused by aerodynamic stall occurring either in the impeller or in the diffuser. This results in a flow direction reversal for a short time interval, during which time the backpressure drops, and flow assumes its proper direction. The reversal of flow also causes a reversal in the direction of forces, especially thrust forces, which can damage or destroy the compressor. Choking is a condition of maximum mass flow rate through a compressor because of sonic flow at the minimum area point of the compressor. The compressor model in RELAP5-3D© does not impose a limitation within the coding to prevent operation beyond (i.e., to the left and above) the surge line. Therefore, the user should carefully analyze the results to ensure operation in this region does not occur. Also, the RELAP5-3D© choking model will probably not predict choking at the correct point, because the actual flow areas within the compressor blade region are not included in the model. Therefore, the user should ensure that operation beyond the choking point does not occur.

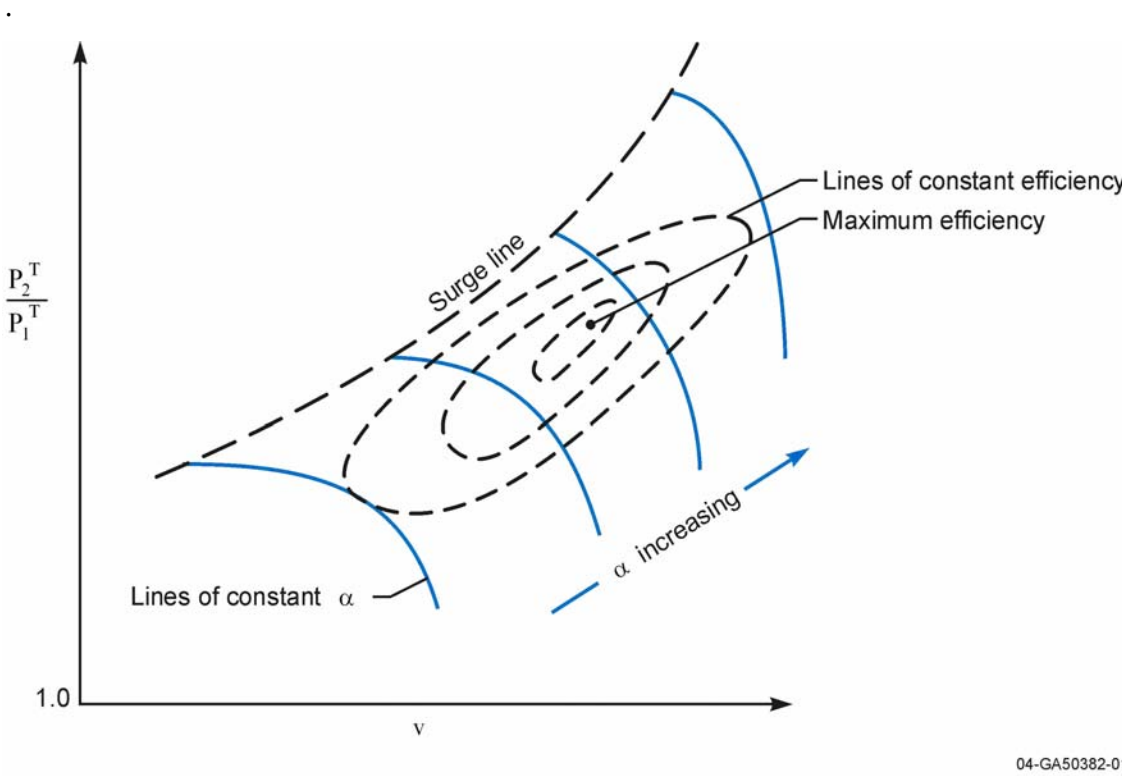


Figure 2.4-16 Compressor performance map.

In RELAP5-3D, compressor performance maps are read in as a series of tables, one for each value of angular velocity. Each table consists of triplets of numbers for mass flow rate, pressure ratio, and efficiency. Angular velocity and mass flow rate are entered as relative corrected values, corrected to rated stagnation sound speed and density, as given by

$$\alpha = \frac{\left(\frac{N}{a_0}\right)}{\left(\frac{N}{a_0}\right)_{Rated}} \tag{2.4-50}$$

and

$$v = \frac{\left(\frac{\dot{m}}{\rho_0 a_0}\right)}{\left(\frac{\dot{m}}{\rho_0 a_0}\right)_{Rated}} \tag{2.4-51}$$

respectively. See Volume I, Section 3.5.12.1 for further information. The tables are read using simple linear interpolation using the relative corrected angular (rotational) velocity and relative corrected mass flow rate

parameters as independent variables. The potential exists for large interpolation errors when using this method. Errors in the efficiency values are especially prone to interpolation errors, because of the nonlinearity in the adiabatic efficiency curves. As shown in **Figure 2.4-16**, the lines of constant adiabatic efficiency are "islands" having roughly elliptical shapes on the performance map when a mass flow rate parameter is the variable of the abscissa and a head-related parameter is the variable of the ordinate. Because of the potential for interpolation errors, the user should supply intermediate lines of constant corrected angular (rotational) velocity as necessary to obtain acceptable interpolation accuracy. Additionally, the user may need to supply extrapolated values to ensure that operation within valid operating regions will not result in an interpolation failure. The user should use caution when applying the extrapolated values so that operation of the model above and to the left of the surge line does not occur. As noted above, compressor surge is a region of instability within the compressor, but the surge line is not imposed as a limit within the RELAP5-3D© model. In addition, the user should be aware that the compressor model does not include the physical areas that physically exist within the compressor machine, so accurate representation of local velocities is not available. Therefore, a further limitation of the RELAP5-3D© compressor model is that it will not correctly predict choking within the compressor model.

2.4.20 References

2.4-1. E. Buckingham, "Model Experiments and the Forms of Empirical Equations," *Transactions of the ASME, 37*, 1915, p. 263.

2.4-2. Aerojet Nuclear Company, *RELAP4/MOD5: A Computer Program for Transient Thermal-Hydraulic Analysis of Nuclear Reactors and Related Systems, User's Manual, Volume 1, RELAP4/MOD5 Description*, ANCR-NUREG-1335, Idaho National Engineering Laboratory, September 1976.

2.4-3. R. F. Farman and N. R. Anderson, "A Pump Model for Loss-of-Coolant Accident Analysis," *Topical Meeting on Water Reactor Safety*, CONF-730304, March 1973, pp. 569-580.

3 Heat Structures

Heat structures represent the selected, solid portions of the thermal-hydrodynamic system. Being solid, there is no flow, but the total system response depends on heat transferred between the structures and the fluid, and the temperature distributions in the structures are often important requirements of the simulation. System components simulated by heat structures include fuel rods, pipe walls, core barrels, pressure vessels, and heat exchanger tubing. In simulations that do not involve core damage, heat structures can represent fuel pins, control rods, and other structural components. Temperatures and heat transfer rates are computed from the one-dimensional form of the transient heat conduction equation for non-reflood and from a two-dimensional form of the transient heat conduction equation for reflood. In core damage simulations, the SCDAP/RELAP5-3D© code[3.0-1] should be used.

A heat structure is identified by a number, CCCG0NN. The subfield, CCC, is the heat structure number and is analogous to the hydrodynamic component number. Since heat structures are usually closely associated with a hydrodynamic component, it is suggested that the hydrodynamic component number and the CCC portion of the attached heat structures be the same number. Since different heat structures can be attached to the same hydrodynamic component, such as fuel pins and a core barrel attached to a core volume, the G portion can be used to distinguish the different types of heat structures. The combined field, CCCG, is the heat structure-geometry number, and input data are organized by this heat structure geometry number. Up to 99 individual heat structures may be defined using the geometry described for the heat structure geometry number. The individual heat structures are numbered consecutively starting at 01; this number is the subfield, NN, of the heat structure number. The heat structure input requirements are divided into input common to all heat structures with the heat structure geometry number, Cards 1CCCG000 through 1CCCG499, and input needed to uniquely define each heat structure, 1CCCG501 through 1CCCG999.

3.0.1 Reference

3.0-1. The SCDAP/RELAP5-3D© Development Team, *SCDAP/RELAP5-3D© Code Manual, Volumes I, II, III, IV, V,* INEEL-EXT-02-00589, Idaho National Engineering and Environmental Laboratory, Rev. 2.2, October 2003.

3.1 Heat Structure Geometry

For one-dimensional form of the transient heat conduction equation, temperature distributions in heat structures are assumed to be adequately represented in rectangular, cylindrical, or spherical coordinates. The spatial dimension of the calculation is along any one of the coordinates in rectangular geometry and is along the radial coordinate in cylindrical or spherical geometry. The one-dimensional form assumes no temperature variations along the other coordinates. **Figure 3.1-1** illustrates placement of mesh points at which temperatures are computed. The mesh point spacing is taken in the positive direction from left to right. RELAP5-3D© uses the terms “left” and “right” to describe the opposite sides of a heat structure. This has no particular geometric interpretation for rectangular geometry heat structures but does for cylindrical and spherical heat structures. By convention, the right surface represents the outside diameter and the left

surface represents the inside diameter. For solid cylinders and spheres, therefore, the left surface is actually a point (a cylinder or sphere with zero inside diameter). For all three geometries (rectangular, cylindrical, and spherical), the left boundary coordinate must be less than the right boundary coordinate because the mesh increments are positive. A composition is a material with associated thermal conductivity and volumetric heat capacity. Mesh points must be placed such that they lie on the two external boundaries and at any interface between different compositions. Additional mesh points may be placed at desired intervals between the interfaces or boundaries. There is no requirement for equal mesh intervals between interfaces, and compositions may vary at any mesh point. Experience has shown that with too small a mesh interval,

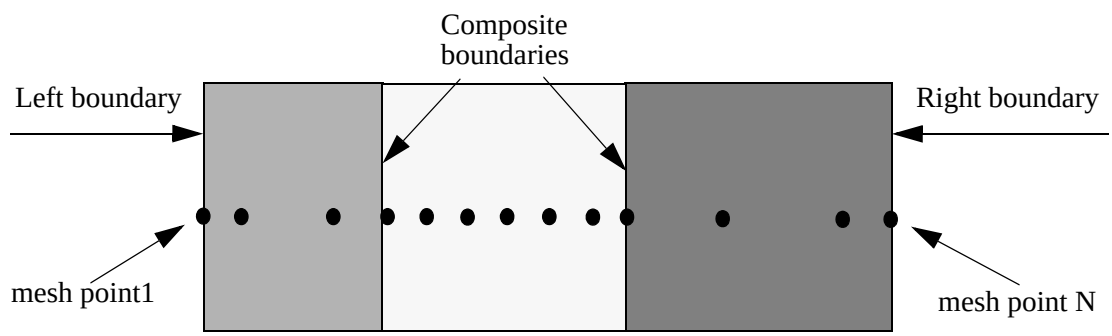


Figure 3.1-1 Mesh point layout.

oscillations can occur. This is particularly evident when modeling a gap, where it is recommended only one mesh interval be used.

The heat structure input processing provides a convenient means to enter the mesh point spacing and composition placement. Each composition is assigned a three-digit, nonzero number (these numbers need not be consecutive). For each composition specified, corresponding thermal property data are needed to define the thermal conductivity and volumetric heat capacity as functions of temperature; the temperature-dependence can be described by tabular data or by a set of functions. Built-in (stored within the program) typical thermal property data as a function of temperature are also available; this is for fuel rod gap gas, carbon steel, stainless steel, uranium dioxide, and zircaloy. These are discussed in Section 4.5 of Volume I and Section 4.7.5 of Volume V of the manual. The data were stored within the program as a user convenience and should be reviewed for applicability to the specific structures being modeled. Defining thermal property data for compositions not specified in any heat structure is not considered an error but does waste storage space. Input editing includes the thermal properties, and a list of the built-in data can be obtained by assigning the built-in materials to unused composition numbers in any input-check run. The thermal property data must span the temperature range of the problem. Problem advancement is terminated if temperatures are computed outside the range of the data.

Heat structures can have an internal volumetric heat source that can be used to represent nuclear, gamma, or electrical heating. The internal volumetric source $S(\bar{x},t)$ for the one-dimensional form of the transient heat conduction equation is assumed to be a separable function of space and time; it is of the form

$$S(\bar{x},t) = P_f Q(x) P(t) \tag{3.1-1}$$

where

P_f	=	a scaling factor
$Q(x)$	=	a space distribution function in the x direction
$P(t)$	=	power.

The space distribution function is assumed to be constant over a mesh interval but may vary from mesh interval to mesh interval. Only the relative distribution of the space function is important, and it may be scaled arbitrarily. For example, given a heat structure with two zones, the first zone having twice the volumetric internal heat generation of the second, the input space distribution factors for the two zones could be 2.0 and 1.0, 200.0 and 100.0, or any numbers with the 2-to-1 ratio. Zeros can be entered for the space distribution if there is no internal heat source. If zeros are entered, the internal heat source will be zero.

As discussed in Volume I, the integral of the internal volume source for the one-dimensional form of the heat conduction equation over the two sub-volumes on either side of a mesh point gives

$$\iiint_V S(\bar{x}, t)dV \approx P_f P(t)(Q_{lm}\delta_{lm}^V + Q_{rm}\delta_{rm}^V) \ . \tag{3.1-2}$$

The term P_f is the factor that relates the reactor power (or power from a table or control variable) to the heat generation rate for a particular heat structure. Word 2 in the heat structure cards 1CCCG701 through 1CCCG799 is used to enter P_f. The term P(t) is the time varying function that may be reactor power, power from a table, or power from a control variable. The volume weights δ_{lm}^V and δ_{rm}^V are shown in Volume I, Section 4.1. The space-dependent terms for the left and right intervals, Q_{lm} and Q_{rm}, are calculated from

$$Q_{lm} = \frac{Q_{lm,\,input}}{\sum_m Q_{lm,\,input}\delta_{lm}^V + \sum_m Q_{rm,\,input}\delta_{rm}^V} \tag{3.1-3}$$

$$Q_{rm} = \frac{Q_{rm,\,input}}{\sum_m Q_{lm,\,input}\delta_{lm}^V + \sum_m Q_{rm,\,input}\delta_{rm}^V} \tag{3.1-4}$$

where the summation is over all the space mesh points for a particular heat structure. The space-dependent input terms for the left and right intervals, $Q_{lm,\,input}$ and $Q_{rm,\,input}$, are entered in Word 1 ($Q_{i,input}$ for the ith interval) of heat structure cards 1CCCG301 through 1CCCG399. As shown in Equations (3.1-3) and (3.1-4), the input terms $Q_{lm,\,input}$ and $Q_{rm,\,input}$ are relative values only; that is they are normalized by the summations in the denominator to obtain the final Q_{lm} and Q_{rm} terms. It should also be noted that the input terms $Q_{lm,\,input}$ and $Q_{rm,\,input}$ are multiplied by the volumetric weights δ_{lm}^{V} and δ_{rm}^{V} in the summations in the denominator.

The mesh point spacings, composition placement, and source space distribution are common to all the heat structures defined with the heat structure geometry number, and only one copy of this information is stored. If a heat structure geometry has this data in common with another heat structure, input preparation and storage space can be saved by referencing the data in the other component. There are no ordering restrictions as to which heat structure geometry may reference another; one heat structure geometry may reference another, which in turn references a third, etc., as long as a defined heat structure is finally reached.

An initial temperature distribution may be entered for each heat structure geometry. This initial distribution is common to all heat structures defined with the same heat structure geometry number, but storage space for temperatures is assigned to each heat structure. Referencing initial temperature distributions in other heat structure geometries is allowed. Optionally, an initial temperature distribution may be entered for each heat structure.

The input temperature distribution can be used as the initial temperature distribution, or initial temperatures can be obtained from a steady-state heat conduction calculation using initial hydrodynamic conditions and zero-time power values. The input temperature distribution is used as the initial temperature guess for iterations on temperature-dependent thermal properties and boundary conditions. If a good temperature guess is not known, setting the temperature of any surface connected to a hydrodynamic volume equal to the volume temperature assists the convergence of the boundary conditions. The iteration process is not very sophisticated, and convergence to 0.01 K occasionally is not obtained. Input of a better initial distribution, especially surface temperatures, usually resolves the problem.

3.2 Heat Structure Boundary Conditions

Boundary condition input specifies the type of boundary condition, the possible attachment of a heat structure surface to a hydrodynamic volume, and the relating of the one-dimensional or two-dimensional heat conduction solution to the actual three-dimensional nature of the structure. Each of the two surfaces of a heat structure may use any of the boundary conditions and may be connected to any hydrodynamic volume. Any number of heat structure surfaces may be connected to a hydrodynamic volume, but only one hydrodynamic volume may connect to a heat structure surface. When a heat structure is connected to a hydrodynamic volume, heat transferred from or to the heat structure is added to or subtracted from the internal energy content of the hydrodynamic volume. For both left and right surfaces, a positive heat transfer rate represents heat flow out of the surface.

There is an input option to decouple a heat structure from the hydrodynamic components. Decoupling a heat structure means that the heat structure responds to the hydraulic conditions in the hydrodynamic volumes to which it is connected, but the energy removed from (or added to) the surface of the heat structure by convection is not added to (or removed from) the hydrodynamic volumes. Decoupling is useful for a CSAU methodology and for debugging.

A non-convective boundary condition is used for a radiation/conduction enclosure heat flux boundary condition and for a symmetry/insulated boundary condition. The radiation/conduction enclosure model is discussed in Volume I, Sections 4.13 and 4.14. A symmetry or insulated boundary condition specifies no heat transfer at the surface, that is, a zero temperature gradient at the surface. This condition should be used in cylindrical or spherical coordinates when the radius of the left-most mesh point is zero, though the numerical techniques impose the condition regardless of the boundary condition specified. If a rectangular geometry is modeled with both surfaces attached to the same hydrodynamic volume, with the same boundary conditions, and having symmetry about the structure midpoint, storage space and computer time can be saved by describing only half of the structure. The symmetry boundary condition is then used at one of the surfaces, and the heat surface area is doubled. This boundary condition can also be used when a surface is very well insulated. The heat transfer mode (for this case when the non-convective boundary condition is selected by the user) listed in the printed output is zero.

When a heat structure is connected to a hydrodynamic volume, a set of heat transfer correlations can be selected by the user (Cards 1CCCG501 through 1CCCG599 and 1CCCG601 through 1CCCG699, Word 3 = 1 or 1nn) as boundary conditions. The correlations cover the various modes of heat transfer from a surface to the fluid, and the reverse heat transfer from the fluid to the surface. The heat transfer modes (for this case when the heat transfer correlations are selected by the user) listed in the printed output are:

Mode 0 Convection to noncondensable-vapor-liquid mixture.

Mode 1 Single-phase liquid convection at supercritical pressure.

Mode 2 Single-phase liquid convection, subcooled wall, low void fraction.

Mode 3 Subcooled nucleate boiling.

Mode 4 Saturated nucleate boiling.

Mode 5 Subcooled transition boiling.

Mode 6 Saturated transition boiling.

Mode 7 Subcooled film boiling.

Mode 8 Saturated film boiling.

Mode 9 Single-phase vapor/gas convection or supercritical pressure with the void fraction greater than zero.

Mode 10 Condensation when the void fraction is less than one (For FWHTR heat slabs, a combination of condensation above the water level and liquid convection below it).

Mode 11 Condensation when the void fraction equals one (For FWHTR heat slabs, condensation on a heat slab completely above the water level).

Mode 12 Heat flux non-positive from nucleate boiling.

If the noncondensable quality is greater than 10^{-9}, 20 is added to the mode number. If the structure is a reflood structure, 40 is added. Thus, the mode number can vary from 0 to 72.

Generally, the hydrodynamic volume will not be a time-dependent volume. Caution should be used in specifying a time-dependent volume, since the elevation and length are set to zero, and the velocities in an isolated time-dependent volume will be zero. Note that the current version of the code does not allow an isolated standard volume or an isolated time-dependent volume.

Users now have the option to be more specific about the type of hydraulic volume a heat slab is next to. Most of the default heat transfer coefficients originate from data taken inside vertical pipes. Users can now specify that the fluid is flowing between parallel plates, in a vertical or horizontal rod or tube bundle, or below a flat plate. When modeling a vertical bundle, the rod or tube pitch-to-diameter ratio should be input. This has the effect of increasing the convective part of heat transfer such that users can input the true hydraulic diameter and get reasonable predictions.

Other boundary condition options that can be selected are as follows: setting the surface temperature to a volume fraction averaged fluid temperature of the boundary volume, obtaining the surface temperature from a control variable, obtaining the surface temperature from a time-dependent general table, obtaining the heat flux from a time-dependent general table, or obtaining heat transfer coefficients from either a time- or temperature-dependent general table. For the last option, the associated sink temperature can be a volume fraction averaged fluid temperature of a boundary volume, or it can be obtained from a time-dependent general table, or it can be obtained from a control variable, or it can be set to zero. These options are generally used to support various efforts to analyze experimental data and do not contain all the physics present in the boundary condition options that use the heat transfer correlations. The user needs to use caution when using the heat flux boundary condition. If the heat flux is too large (positive or negative), a numerical failure may result. The heat transfer mode (for this case when the other boundary condition options are selected by the user) listed in the printed output is zero.

Another boundary condition that can be selected is the alternate heat structure - fluid coupling model. This model increases the accuracy of the coupled solution of energy equations for the fluid and heat structures with vapor/gas coolants.

The various boundary condition options are input through heat structure input cards. The following table (**Table 3.2-1**) shows the available options for Cards 1CCCG501 through 1CCCG599 and 1CCCG601 through 1CCCG699, Words 1 and 3, where the surface temperature (T_w) and the sink temperature (T_{sink})

Table 3.2-1 Boundary condition options.

Word 3	**Word 1 = 0 Non-convective or T_w = 0 or T_{sink} = 0**	**Word 1 > 0 Boundary Volume**	**Word 1 < 0 General Table or Control Variable**
0	Non-convective $q_w = 0$ $h_w = 0$	------	------
1 or 1nn	------	correlations for h_{wf} and h_{wg}	------
1000	$T_w = T_{sink} = 0$	$T_w = T_{sink} = \alpha_g T_g + \alpha_f T_f$	$T_w = T_{sink}$ from table or control variable in Word 1
1xxx	$T_w = f(time)$ $T_{sink} = 0$	$T_w = f(time)$ $T_{sink} = \alpha_g T_g + \alpha_f T_f$	$T_w = f(time)$ T_{sink} from table or control variable in Word 1
2xxx	$q_w = f(time)$ $T_{sink} = 0$ $q_{wg} = (1/2)q_w$	$q_w = f(time)$ $T_{sink} = \alpha_g T_g + \alpha_f T_f$ $q_{wg} = \alpha_g q_w$	$q_w = f(time)$ T_{sink} from table or control variable in Word 1 $q_{wg} = (1/2)q_w$
3xxx	$h_w = f(time)$ $h_{wg} = (1/2)h_w$ $q_w = h_w(T_w - T_{sink})$ $T_{sink} = 0$ $q_{wg} = (1/2)q_w$	$h_w = f(time)$ $h_{wg} = \alpha_g h_w$ $q_w = h_w(T_w - T_{sink})$ $T_{sink} = \alpha_g T_g + \alpha_f T_f$ $q_{wg} = \alpha_g q_w$	$h_w = f(time)$ $h_{wg} = (1/2)h_w$ $q_w = h_w(T_w - T_{sink})$ T_{sink} from table or control variable in Word 1 $q_{wg} = (1/2)q_w$
4xxx	$h_w = f(T_w)$ $h_{wg} = (1/2)h_w$ $q_w = h_w(T_w - T_{sink})$ $T_{sink} = 0$ $q_{wg} = (1/2)q_w$	$h_w = f(T_w)$ $h_{wg} = \alpha_g h_w$ $q_w = h_w(T_w - T_{sink})$ $T_{sink} = \alpha_g T_g + \alpha_f T_f$ $q_{wg} = \alpha_g q_w$	$h_w = f(T_w)$ $h_{wg} = (1/2)h_w$ $q_w = h_w(T_w - T_{sink})$ T_{sink} from table or control variable in Word 1 $q_{wg} = (1/2)q_w$

are discussed in Volume I, Section 4.1 of the manual.

A factor must be entered to relate the one-dimensional heat conduction representation to the actual heat structure. Two options are available for entry: either a heat transfer surface area or a

geometry-dependent factor. For rectangular geometry, the factor is the surface area; there is no difference in the options. In cylindrical geometry, the heat structure is assumed to be a cylinder or a cylindrical shell, and the factor is the cylinder length. For a circular pipe where a hydrodynamic volume represents the flowing part of the pipe and a heat structure represents the pipe walls, the factor equals the hydrodynamic volume length. For a hydrodynamic volume representing a core volume with fuel pins or a heat exchanger volume with tubes, the factor is the product of the hydrodynamic volume length and the number of pins or tubes. In spherical geometry, the heat structure is assumed to be a sphere or a spherical shell, and the factor is the fraction of the sphere or shell. For a hemisphere, the factor would be 0.5. Except for solid cylinders or spheres where the inner surface area is zero, one surface area can be inferred from the other and the mesh point spacing information. Nevertheless, both surface areas must be entered and an input error will exist if the surfaces are not consistent. Consistency is defined to be such that the difference between the calculated left and right factors (or the input left or right factors) must be $\leq 10^{-5}$ times the sum of the calculated left and right factors (or the input left and right factors). This requirement is easily met with the second option of entering a geometry-dependent factor, since the factor is the same for the left and right boundary.

3.3 Heat Structure Sources

Volumetric heat sources for heat structures consist of the product of an input scaling factor, an input space-dependent function, and a time function. The time function may be total reactor power, fission power, or fission product decay power from the reactor kinetics calculation; may be a control variable; or may be obtained from a general table of power versus time. Input data provide for three scaling factors. The first scaling factor is applied to the power to indicate the internal heat source generated in the structure. This means that in steady-state, heat equal to the first scaling factor times the time function power value would be generated in the heat structure and transferred out through its left and right surfaces. If P(t) is the power in Watts and P_f is the first scaling factor, then P_f P(t) is the heat generated in Watts. Within the program, the space-dependent input term is divided by the summation of the space-dependent input term times the volume weight for each interval to allow for the arbitrary scaling of that term as described in Volume II, Section 3.1.

After this scaling, the internal volumetric heat source (S) is in the required units of Watts/m^3. The volumetric heat source of an interval can be converted to the internal heat source (in Watts) of an interval by multiplying by the volume of the interval.

The other two scaling factors provide for the direct heating of the fluid in the hydrodynamic volumes attached to the surfaces. Heat equal to the factor times the power value is added to the internal energy of the fluid in the hydrodynamic volume without passing through the heat source. If P(t) is the power in Watts and P_{dl} is the left direct heat scaling factor, then P_{dl} P(t) is the heat added directly to the fluid in the left boundary volume. If P(t) is the power in Watts and P_{dr} is the right direct heat scaling factor, then P_{dr} P(t) is the heat added directly to the fluid in the right boundary volume. The total direct heating added to a volume is the sum of the direct heating from all structures connected to the volume. Zeros are entered where no heat source or hydrodynamic volumes exist.

In a reactor problem, if a single table or control variable represents the total reactor power generated and if this power is totally accounted for in the RELAP5-3D© model, then the sum of the P_f, P_{dl}, and P_{dr} scaling factors over all the heat structures that references that table or control variable should generally equal one. The summing to one is not required, and no checks are performed by the code. The user may find it convenient to apply a different table or control variable to each powered heat structure, in which case the sum of the P_f, P_{dl}, and P_{dr} scaling factors should be one for each powered heat structure.

In many instances, the power will be applied not only to the heat structures representing the fuel but also to the heat structures representing such items as the downcomer and pressure vessel walls to model gamma heating.

3.4 Heat Structure Changes at Restart

At restart, heat structures may be added, deleted, or replaced. Since heat structure input data are organized with respect to a heat structure geometry, all heat structures with the heat structure geometry number are affected.

Composition and general table data can also be added, deleted, or replaced at restart. A transient or steady-state problem terminated by a heat structure temperature out of range of the thermal property data can be restarted at the restart prior to the termination by replacing the thermal property data.

3.5 Heat Structure Output and Recommended Uses

Up to six sections of heat structure output are printed at major edits. The first section prints one line of heat transfer information for each surface of each heat structure. Each line provides the heat structure number; a left or right surface indicator; the connected hydrodynamic volume or, if none, zero; surface temperature; the heat transfer rate; the heat flux; the critical heat flux; the critical heat flux multiplier; the mode of heat transfer; and the heat transfer coefficient. The first line for each heat structure also includes the heat input to the structure, the net heat loss from the structure, and the volume-average temperature for the structure. The critical heat flux multiplier is the value used to multiply the value from the CHF table.

The second section prints the mesh point temperatures for each heat structure. This section can be suppressed by an input option.

The other optional sections include output on reflood, metal-water reaction, rupture, and radiation/conduction enclosure.

For the heat structure additional boundary cards (1CCCG801 through 1CCCG899 and 1CCCG901 through 1CCCG999), it is suggested to use zero for the heat transfer hydraulic diameter (D_{he}) (i.e., heated equivalent diameter, thermal diameter) for a uniformly heated tube or channel. When zero is used, the heat transfer hydraulic diameter is set the same as the volume hydraulic diameter (D_h) at the boundary volume, which should be determined by the user from

$$D_h = 4 \bullet \frac{\text{flow area}}{\text{wetted perimeter}} \quad . \tag{3.5-1}$$

For tube and channel tests, the same scaling method used in the hydraulic calculation should be used in the heat transfer calculation.

If the heat structure does not represent uniformly heated pipe or channel walls, the default should not be taken. The heat transfer hydraulic diameter should be determined by the user from

$$D_{he} = 4 \bullet \frac{\text{flow area}}{\text{heated perimeter}} \quad . \tag{3.5-2}$$

3.6 Reflood Model

The reflood model is designed so that it can be activated at low pressure (less than 1.2×10^6 Pa) with nearly empty conditions (average void fraction in a connected stack of hydrodynamic volumes > 0.9) or dryout beginning (average void fraction in a connected stack of hydrodynamic volumes > 0.1) or by user command through a trip. The model considers a heat-structure geometry composed of 1 to 99 heat structures as a reflood unit. As there is no input specification for the length of a structure (except for the heat structure surface), such length is inferred from the length of the boundary volume connected to the heat structure. It is the user's responsibility to make certain that the length of a heat structure corresponds to the length of its connected volume for reflood calculations.

Additional suggestions concerning the use of the reflood model are listed below:

1. The appropriate user-specified maximum number of axial fine mesh intervals is 8 to 32. No significant differences have been found in using 16 to 128 axial nodes for 0.18-m (0.6 ft) long heat structures.

2. The appropriate length of hydrodynamic volumes is 0.15 to 0.61 m (0.5 to 2.0 ft).

3. The maximum user-specified time step size is 0.01 to 0.05 seconds.

4. Each reflood unit should have its own flow channel and parallel flow channels should be connected by crossflow junctions.

The number of heat-structure geometries that can be specified for a reflood calculation is limited only by computer storage capacity. The heat transfer modes that appear in the mode column of the major edit are the same as those that appear when reflood is not activated except that 40 is added (see Section 3.2).

4 Trips and Controls

4.1 Trips

Extensive trip logic has been implemented in RELAP5-3D©. Each trip statement is a single logical statement, but because trip statements can refer to other trip statements, complex logical statements can be constructed.

There are two aspects to trip capability: (a) to determine when a trip has occurred, and (b) to determine what to do when a trip occurs. In the modular design of RELAP5-3D©, these two aspects have been separated. The term trip logic refers only to the first aspect and includes the input processing of the trip statements and the transient testing to set trip status. The action to be taken when a trip occurs is considered to be part of a particular model, and that aspect of trip coding is associated with the coding for the model. Examples of the second aspect of trips are the effects of trips on pump models and check valves.

Trip capability provides for variable and logical trips. Both types of trips are logical statements with a false or true result. A trip is false (that is off, not set, or has not occurred) if the result is false. A trip is true (that is on, is set, or has occurred) if the result is true. Trips can be latched or unlatched. A latched trip, once true (set), remains true (set) for the remainder of the problem execution, even if conditions change such that the logical statement is no longer true. An unlatched trip is tested at each time step, and the conditions can be switched at any step.

A TIMEOF quantity is associated with each trip. This quantity is always -1.0 for a trip with the value false. When a trip is switched to true, the time at which it switches replaces the value in TIMEOF. Linear interpolation, along with limits between the current time step and the previous time step, is used to obtain a more accurate time at which the trip switched to true. For a latched trip, this quantity once set to other than -1.0 always retains that value. An unlatched trip may have several TIMEOF values other than -1.0 during a simulation. Whenever an unlatched trip switches to false, TIMEOF becomes -1.0; when true again, the new time of switching to true is placed in TIMEOF. The TIMEOF quantities are used to effect delays in general tables, time-dependent volumes, time-dependent junctions, and pump speed tables, and can be referenced in the control system.

Two card formats are available for entering trip data. All trips for a problem must use the same format. At restart, the same format must be used for trip modifications unless all trips are deleted (Card 400) and desired trips are reentered. The default format uses Cards 401 through 599 for variable trips and Cards 601 through 799 for logical trips. The trip number is the same as the card number. Up to 199 variable trips and up to 199 logical trips can be defined. An alternate format is selected by entering Card 20600000. Trip data are entered on Cards 206NNNN0, where NNNN is the trip number. Trip numbers 1 through 1000 are variable trips, and trip numbers 1001 through 2000 are logical trips. The alternate format allows 1,000 trips each for variable and logical trips.

As trips are input, the default initial value is false. Optionally, the TIMEOF quantity may be entered. If -1.0 is entered, the trip is false; if 0 or a positive number is entered, the trip is true, and the entered quantity is the time the trip turned true. This quantity must be less than or equal to the time of restart. For a new problem, 0.0 must be entered.

Several options are available on restart. If no trip data are entered, trips are defined at restart with the values at restart. It is possible to delete all trip definitions and enter completely new definitions. Individual trips can be deleted or redefined, and new trips can be inserted. Individual trips can be reset to false. At restart, a latched trip can be reset. Detailed discussions and examples of the use of trips are presented in Section 4 of Volume V of this code manual.

4.1.1 Variable Trips

A variable trip evaluates a comparison statement relating two variables and a constant using one of the relationships, equal (EQ), not equal (NE), greater than or equal (GE), greater than (GT), less than or equal (LE), or less than (LT). The variables currently allowed are listed in the Input Requirements (Appendix A). Most variables advanced in time are allowed, and any variable that is permanently stored can be added to the list. The only restriction on the two variables is that they have the same units if neither variable is a control variable. Thus, a hydrodynamic volume temperature can be compared to a heat structure temperature, but a pressure cannot be compared to a velocity. The variable trip statement is

$$\text{NUM} \quad \text{VAR1} \quad \text{OP} \quad \text{VAR2} + \text{CONSTANT} \quad \begin{bmatrix} \text{L} \\ \text{N} \end{bmatrix} \quad \text{TIMEOF} \tag{4.1-1}$$

where NUM is the card number; VAR1 and VAR2 each consist of two words that identify a variable, the first word being alphanumeric for the variable type, the second word being a number associated with the particular variable; OP is the comparison operation; CONSTANT is a signed number to be added to VAR2 before comparison; and either L or N is used to indicate a latched or unlatched trip. TIMEOF is the optional initialization value. A special form NULL,0 is used to indicate that no variable is to be used. VAR2 must be NULL,0 if VAR1 is to be compared only to the constant. Either VAR1 or VAR2 may also be TIMEOF, trip number. The trip number may refer to either a variable or a logical trip. When a variable trip statement references a TIMEOF variable whose value is -1.0 (i.e., the trip is false), the evaluation of the variable trip is bypassed. Thus, the value of the variable trip remains the same as it value on the previous time step.

Three examples of variable trips are

501	P,3010000	LT	NULL,0	1.5+5	N
502	P,5010000	GT	P,3010000	2.0+5	N
510	TIME,0	GE	NULL,0	100.0	L

Trip 501: Is the pressure in volume 3010000 < 1.5 bar (1 bar = 10^5 Pa)?

Trip 502: Is the pressure difference between volumes 5010000 and 3010000 > 2.0 bar?

Trip 510: Is the current advancement time $\geq$ 100 s?

Use of the equal (EQ) or not equal (NE) operator should be avoided because fractions expressed exactly in decimal notation may not be exact in binary notation. As an example, assume a time step of 0.01. After ten advancements, the time should be 0.10, but an equality test of time equal to 0.10 would probably fail. An analogous situation is dividing 1 by 3 on a three-digit decimal calculator, obtaining 0.333. Adding 1/3 three times should give 1.000, but 0.999 is obtained.

4.1.2 Logical Trips

A logical trip evaluates a logical statement relating two trip quantities with the operations AND, OR (inclusive), or XOR (exclusive). **Table 4.1-1** defines the logical operations where 0 indicates false, 1 indicates true. The table shows the result for each combination of true and false for two operands for each of the logical operations. Each trip quantity may be the original value or its complement. (Complement means reversing the true and false values; that is, the complement of true is false.)

Table 4.1-1 Logical operations.

Operands/Operation	**AND**				**OR**				**XOR**			
Trip1	0	0	1	1	0	0	1	1	0	0	1	1
Trip2	0	1	0	1	0	1	0	1	0	1	0	1
Result	0	0	0	1	0	1	1	1	0	1	1	0

The logical trip statement is:

$$\text{NUM} \quad \pm\text{TRIP1} \quad \text{OP} \quad \pm\text{TRIP2} \quad \begin{bmatrix} \text{L} \\ \text{N} \end{bmatrix} \quad \text{TIMEOF} \tag{4.1-2}$$

where NUM is the card number, TRIP1 and TRIP2 are either variable or logical trip numbers, OP is the logical operator, L or N are for latched or unlatched trips, and TIMEOF is the optional initialization value. A positive trip number means the original trip value; a negative number means the complement value. Examples of logical trips are

601 501 OR 502 N

602 601 AND 510 N

620 -510 OR -510 N

Trip 602 involves a previous logical trip and illustrates the construction of a complex logical statement. With the definitions given in the examples above and using parentheses to indicate the order of logical evaluations, Trip 602 is equivalent to {(Pressure 3010000 < 1.5 bar) OR [Pressure 5010000 > (Pressure 3010000 + 2.0 bar)]} AND (Time $\geq$ 100 s). Trip 620 is the complement of Trip 510, and the AND operation in place of the OR operation would also give the same result. Additional examples of trips are presented in Volume V of this code manual.

4.1.3 Trip Execution

The trip printout for a new problem at time equal to 0 seconds shows trips as they were entered at input. On restarted problems, the trip printout at the restart time shows input values for new and modified trips and the values from the original problem for the unmodified trips.

Trip computations are the first calculation of a time step. Thus, trip computations use the initial values for the first time step and the results of the previous advancement for all other advancements. Because trips use old values, they are not affected by repeats of the hydrodynamic and heat structure advancements.

Trips are evaluated in order of trip numbers; thus, variable trips are evaluated first, then logical trips (refer to the discussion of trips in Volume I). Results of variable trips involving the TIMEOF quantity and logical trips involving other trips can vary, depending on their position relative to other trips. As an example, consider

6XX -650 OR -650 N

which just complements Trip 650. Also, assume Trip 650 switches to true this time step, and, thus, 650 was false and 6XX was true previous to trip evaluation. At the end of trip evaluation, 6XX is true if 6XX is < 650 and false if 6XX is > 650. If Trip 650 remains true for the following time step, Trip 6XX with 6XX < 650 becomes false one time step late. Similarly, TIMEOF quantities can be one time interval off. This can be minimized by ordering TIMEOF tests last and defining logical trips before they are used in logical statements.

4.1.4 Trip Logic Example 1

Techniques from Boolean algebra can assist in formulating the logical trip statements. Consider a motor-operated valve that operates such that if the valve stem is stationary, it remains stationary until a specified pressure exceeds 12 bar or drops below 8 bar. The valve starts opening when the pressure exceeds 12 bar and continues opening until the pressure drops below 11 bar. The valve starts closing when the pressure drops below 8 bar and continues closing until the pressure exceeds 9 bar. The motor valve requires two trips, one to be true when the valve should be opening, the other to be true when the valve should be closing.

The following procedure is used to derive the open trip logic. A Boolean variable has one of two possible values, false (0) or true (1). Define as Boolean variables V_0 which is to be true when the valve should be opening; V_1 as the current value of the valve motion; P_1 true when the pressure is > 11 bar; and P_2 true when the pressure is > 12 bar. **Table 4.1-2** is a truth table that has been constructed by listing all possible combinations of the three input variables, V_1, P_2, and P_1, and the desired output, V_o. The number in the rightmost column is the number resulting from assuming the input values form a binary number; this is done to ensure that all combinations are listed. From the truth table, the following expression can be written,

$$V_0 = (\bar{V}_1 \otimes P_2 \otimes P_1) \oplus (V_1 \otimes \bar{P}_2 \otimes P_1) \oplus (V_1 \otimes P_2 \otimes P_1) \tag{4.1-3}$$

where $\otimes$ indicates AND, $\oplus$ indicates OR, and the bar indicates the complement. The expression is derived by combining (with OR operations) terms from each line having a true value in the output column. Each term consists of the combining of each input variable with AND operations, using the direct variable if the value is true and the complement if the value is false. **Table 4.1-2** shows that two of the combinations are impossible. This is because if P_2 is true, P_1 must also be true; that is, if the pressure is > 12 bar, it is also > 11 bar. Because of the relationship between P_2 and P_1,

$$P_2 \otimes P_1 = P_2 \qquad P_2 \oplus P_1 = P_1 \tag{4.1-4}$$

Using the Boolean identities from **Table 4.1-3**, the logical expression can be reduced to

$$V_0 = (\bar{V}_1 \otimes P_2) \oplus [(V_1 \otimes P_1) \otimes (\bar{P}_2 \oplus P_2)] = (\bar{V}_1 \otimes P_2) \oplus (V_1 \otimes P_1) \tag{4.1-5}$$

Table 4.1-2 Truth table examples.

Output	**Input**			
$\mathbf{V_0}$	$\mathbf{V_1}$	$\mathbf{P_2}$	$\mathbf{P_1}$	**Num**
0	0	0	0	0
0	0	0	1	1
impossible	0	1	0	2
1	0	1	1	3
0	1	0	0	4
1	1	0	1	5

Table 4.1-2 Truth table examples. (Continued)

Output	**Input**			
V_0	**V_1**	**P_2**	**P_1**	**Num**
impossible	1	1	0	6
1	1	1	1	7

Table 4.1-3 Boolean algebra identities.[a]

$A \otimes A = A$	$A \oplus A = A$	$A \otimes 0 = 0$	$A \oplus 0 = A$
$A \otimes \overline{A} = 0$	$A \oplus \overline{A} = 1$	$A \otimes 1 = A$	$A \oplus 1 = 1$
$A \otimes B = B \otimes A$		$A \oplus B = B \oplus A$	
$A \otimes (B \oplus C) = (A \otimes B) \oplus (A \otimes C)$			
$A \oplus (B \otimes C) = (A \oplus B) \otimes (A \oplus C)$			

a. $\otimes$ denotes AND; $\oplus$ denotes OR; bar above quantity denotes complement.

The following trip input implements the logic. Trips 601 through 603 implement the rightmost expression in Equation (4.1-5). Trip 603 specifies the open trip in a motor valve. The trip logic is written as follows:

```
501     P,1010000     GT    NULL,0     11.0+5    N    (P1)

502     P,1010000     GT    NULL,0     12.0+5    N    (P2)

601     -603 AND 502        N          (FIRST TERM OF EQ)

602      603 AND 501        N          (SECOND TERM OF EQ)

603      601 OR 602         N          (OPEN TRIP)
```

The close trip logic can be written similarly.

4.1.5 Trip Logic Example 2

Another consideration in constructing trip logic involves the behavior of the “TIMEOF,nnn or TIMEOF,nnnn” minor edit variables when it is used in the logic of a variable trip, for example, if a delay time is desired for motor valve opening or closing. The behavior of TIMEOF is unique when it is used as one of the two variables of a trip. If one of the two variables is TIMEOF and its value is -1.0 (i.e., its

associated trip is false), the logical comparison to the second trip variable is not made. This prevents directly comparing the value of “-1.0” to a “TIME,0” value, which in most cases would produce a trip the turned true at the wrong time. However, bypassing the logical comparison under this circumstance has the additional effect of latching any trip containing a TIMEOF variable. The following example illustrates this point.

Consider the case where a motor valve is to be opened if the pressure exceeds 12 bars and is to be closed if the pressure is less than 11 bar. A delay time of 4 seconds is required prior to valve opening once the pressure exceeds the opening setpoint, and a delay of 10 seconds is required prior to valve closing once the pressure has decreased below the closing setpoint. Notice that TRIPS 502 and 504 are not used to directly control the opening and closing of the motor valve. The additional logical trips, TRIP 601 and TRIP 602, are required to produce the desired response.

```
501  P,10100000  GT  NULL,0  12.0+5  N  (OPENING SETPOINT)

502  TIME,0  GE  TIMEOF,501  4.0  N  (TIME DELAY)

601 501  AND  502   N  (OPENS VALVE)

503  P,1010000  LT   NULL,0  11.0+5  N  (CLOSING SETPOINT)

504  TIME,0   GE  TIMEOF,503  10.0  N  (TIME DELAY)

602  503  AND  504  N  (CLOSES VALVE)
```

Initially, TRIP 501 is false, TRIP 502 is false, TRIP 503 is true, and TRIP 504 is false. IF TRIP 503 is still true at 10 seconds, TRIP 504 will become true. Note, however, that if TRIP 501 remained false, and if TRIP 502 were tested when the TIMEOF,501 value were -1.0, it would turn true when TIME,0 > 3 seconds, which is not the desired response. This is the reason the comparison logic is bypassed if either argument of a trip is a TIMEOF variable and its value is -1.0. However, bypassing the comparison has the further effect that, after a trip containing a TIMEOF variable becomes true, it will not thereafter become false, because when the TIMEOF variable becomes -1.0, it will no longer be tested and the trip will retain is previous (true) state. In the above example, if TRIP 502 turned true (4 seconds after P,1010000 exceeded the setpoint), it would remain true regardless of subsequent behavior of TRIP 501. If TRIP 504 subsequently became true (10 seconds after P,1010000 falls below 11 bar), TRIP 502 and TRIP 504 would be true simultaneously. If these trips were being used to directly control a motor valve, this situation would cause the code to fail. Therefore, in the above example, TRIP 601 and TRIP 602 are used to directly control the opening and closing of the motor valve.

4.2 Control Components

The control system provides the capability to evaluate simultaneous algebraic and ordinary differential equations. The capability is primarily intended to simulate control systems typically used in

hydrodynamic systems, but it can also model other phenomena described by algebraic and ordinary differential equations. Another use is to define auxiliary output quantities (such as differential pressures) so they can be printed in major and minor edits and be plotted.

4.2.1 Basic Control Components

The control system capability consists of several types of control components, each type of component defining a control variable as a specific function of time-advanced quantities. The time-advanced quantities include hydrodynamic volume, junction, pump, valve, heat structure, reactor kinetics, and trip quantities, and the control variables themselves, including the control variable being defined. Permitting control variables to be input to control components allows complex expressions to be developed from components that perform simple, basic operations. The basic control components are listed below, followed by a brief review of the evaluation procedure. Familiarity with the control system numerical techniques documented in Section 6 of Volume I is recommended. In the definitions that follow, Y_i is the control variable defined by the i-th control component; A_j, R, and S are real constants input the by user; I is an integer constant input by the user; V_j is a quantity advanced in time by RELAP5-3D© and can include Y_i; t is time; and s is the Laplace transform variable. Superscripts involving the index n denote time levels. Some components include a definition in Laplace transform notation. The name in parentheses is the name used in the input data to select the type of component.

<u>Constant (CONSTANT).</u>

$$Y_i = S \ .$$

<u>Addition-Subtraction (SUM).</u>

$$Y_i = S(A_0 + A_1V_1 + A_2V_2 + ...) \ .$$

<u>Multiplication (MULT).</u>

$$Y_i = S\,V_1V_2 \ .$$

<u>Division (DIV).</u>

$$Y_i = \frac{S}{V_1} \text{ or } S_1\frac{V_2}{V_1} \ .$$

<u>Integer Exponentiation (POWERI).</u>

$$Y_i = SV_1^{\,I} \ .$$

<u>Real Exponentiation (POWERR).</u>

$Y_i = SV_1^R$.

Variable Exponentiation (POWERX).

$Y_i = SV_1^{V_2}$.

Table Lookup Function (FUNCTION).

$Y_i = S\,F(V_1)$

where F is a function defined by a general table lookup and linear interpolation.

Standard Functions (STDFNCTN).

$Y_i = S\,F(V_1, V_2, V_3, ...)$

where F can be $|V_1|$, $\exp(V_1)$, $1n(V_1)$, $\sin(V_1)$, $\cos(V_1)$, $\tan(V_1)$, $\tan^{-1}(V_1)$, $(V_1)^{1/2}$, $MAX(V_1, V_2, ...)$, and $MIN(V_1, V_2, ...)$. Only MAX and MIN may have multiple arguments.

Delay (DELAY).

The delay component is defined by

$Y_i = SV_1(t - t_d)$

where t_d is the delay time. A user-input h determines the number of time-function pairs in the table used to store past values of V_1. The maximum number of time-function pairs is h + 2. The delay table time increment is $\frac{t_d}{h}$. The delayed function is obtained by linear interpolation using the stored past history. As time is advanced, new time values are added to the table. Once the table is filled, new values replace values that are older than the delay time.

Unit Trip (TRIPUNIT).

$Y_i = SU(\pm tr)$.

Trip Delay (TRIPDLAY).

$Y_i = ST_r(t_r)$.

In the two definitions above, t_r is a trip number and, if negative, indicates that the complement of the trip is to be used; and U is 0.0 or 1.0, depending on trip t_r (or its complement if t_r is negative) being false or true. T_r is -1.0 if the trip is false, and the time the trip was last set true if the trip is true. The trip delay result is -S if the trip is false and can be values between 0 and St (t is time) if the trip is true. The trip delay can be limited to values between 0 and St (instead of -S and St) by use of the optional minimum value for the component.

Integration (INTEGRAL).

$$Y_i = S\int_0^t V_1 dt \text{ : or } Y_i(s) = \frac{SV_1(s)}{s} \quad .$$

Differentiation (DIFFERNI or DIFFERND).

$$Y = S\frac{dV_1}{dt} \text{ : or } Y_i(s) = SsV_1(s) \quad .$$

Use of DIFFERNI is *not* recommended, and, if possible, any differentiation should be avoided. See the discussion in Volume I of this manual.

Proportional-Integral (PROP-INT).

$$Y_i = S\left(A_1V_1 + A_2\int_0^t V_1 dt\right) \text{ or } Y_i(s) = S\left(A_1 + \frac{A_2}{s}\right)V_1(s) \quad .$$

Lag (LAG).

$$Y_i = \int_0^t \frac{(SV_1 - Y_i)}{A_1} dt \text{ : or } Y_i(s) = S\left(\frac{1}{1 + A_1 s}\right)V_1(s) \quad .$$

Lead-Lag (LEAD-LAG).

$$Y_i = \frac{A_1 SV_1}{A_2} + \int_0^t \frac{(SV_1 - Y_i)}{A_2} dt \text{ : or } Y_i(s) = S\left(\frac{1 + A_1 s}{1 + A_2 s}\right)V_1(s) \quad .$$

Inverse Kinetics (INVKIN).

The output of this component is computed as

$$Y = S\left(\frac{\frac{\Lambda}{\beta}\left(\frac{dV_1}{dt}\right) + \sum_{i=1}^{N_d} D_i(t_o + \Delta t)}{V_1}\right)$$

where

$$D_i(t_o + \Delta t) = e^{-\lambda_i \Delta t} D_i(t_o) + \frac{\beta_i}{\beta \lambda_i}(1 - e^{-\lambda_i \Delta t})\frac{dV_1}{dt}$$

Each control component generates an equation, and, together, the components generate a system of nonlinear simultaneous equations. The solution of the simultaneous equations is approximated by simply evaluating the equation for each component in order of increasing component numbers and using the currently available information. Evaluation of algebraic control components uses only currently defined values, but evaluation of components involving integration and differentiation use both old (V^n) and new (V^{n+1}) values. For time-advanced variables other than control variables, both the old and new quantities are available. If a control variable is defined (by appearing on the left side of an equation) before it appears on the right side, the correct old and new variables are available. If a control variable appears on the right side before it is defined, or if it appears in the defining equation, the new and old values are off by a time step. That is, V^{m+1} uses V^m and V^m uses V^{m-1}. For good results, the user should try to define a control variable before using it. This is not always possible, as shown in the second example in Section 4.2.2.

Except for a CONSTANT component, each control component may optionally specify a minimum, a maximum, or both. After the component is evaluated by its defining equation, the value is limited by the minimum and maximum values if they are specified.

The control system input provides for an initial value and an initial value (initialization) control flag to indicate whether the initial value is to be computed during the initialization phase of input processing. The initialization of all other systems, such as trips, hydrodynamics, heat structures, and reactor kinetics, precedes that for control systems. If one of those systems needs an initial value of a control system variable, the input value is used. Thus, the control variable value is the input value when used in servo valve initialization, initialization of time-dependent volumes and junctions if control variables are specified as search arguments, initialization of heat structures when a control variable is specified as a heat source, and computation of bias reactivity when control variables contribute to reactivity. However, the input edit and first major edit, after introduction of a control variable, show the value after initialization.

Except for the SHAFT component, RELAP5-3D© treats control system variables as dimensionless quantities. No unit conversion of the input scaling factors or multiplier constants is done when British input units are specified, and no unit conversion is done on output when British output units are specified. All dimensioned variables are stored within the program in SI units, and the units for variables that can be

used in control components are stated in the input description. The user may assume any desired unit for each control variable. It is the user's responsibility to enter appropriate scale factors and multiplier constants to achieve the desired units and to maintain unit consistency.

Two card formats are provided for input of control system data, but only one format may be used in a problem. The default format uses Card 205CCCNN, where CCC is the control component number and NN is a card sequence number. The card format limits the number of control components to 999. The alternate format using Card 205CCCCN can be selected by entering Card 20500000. With the alternate format, only one digit is used for card sequencing, and up to 9999 control components can be used with the four digit CCCC. Control variables are printed in major edits, can be specified for minor edits, and can be plotted.

4.2.2 Control System Examples

Two examples of control system use are given. (See Section A.14 of Appendix A for input format descriptions.) Input for the examples are shown except that symbols enclosed in parentheses are sometimes used where the actual input would need a number. Also, all examples use control component numbers beginning with one.

The first example is the computation of total flow rate in a volume from

$$W = (\alpha_f \rho_f v_f + \alpha_g \rho_g v_g)A \tag{4.2-1}$$

where

α	=	volume fraction
ρ	=	density
v	=	velocity
A	=	flow area
g	=	vapor/gas
f	=	liquid.

Two multiplication components and one addition-subtraction component are used. The time-advanced quantities, α, ρ, and v, are specified as V_1, V_2, and V_3, respectively, in the two multiplication components, one for each phase. The area A is entered as the scaling factor. An addition-subtraction component adds the results from the multiplication components with $A_0 = 0$, $A_1 = A_2 = S = 1.0$, and V_1 and V_2 are the control variables defined by the multiplication components. For the present numerical scheme, the products should be defined first. This control system is assumed to generate

a quantity for plotting only, so initial values are entered as zeros and initialization is selected. For volume number 123010000, input data using the default format would be the following:

```
20500100  FFLOW   MULT   (A)   0.0   1
20500101  VOIDF,123010000   RHOF,123010000
20500102  VELF,123010000
20500200  GFLOW   MULT   (A)   0.0   1
20500201  VOIDG,123010000   RHOG,123010000
20500202  VELG,123010000
20500300  TFLOW   SUM   1.0   0.0   1
20500301  0.0   1.0,CNTRLVAR, 1   1.0, CNTRLVAR, 2
```

The second example is to solve

$$A_2\ddot{x} + A_1\dot{x} + A_{10}\dot{x}x + A_0x + B\int_0^t x\,dt = C \quad . \tag{4.2-2}$$

Assignment of control variables, Y_i, are made to derivative, integral, and product terms, as listed below. In addition, each line shows equivalent expressions derived from algebraic manipulation and definition of an integral.

$$Y_1 = \dot{x}x = Y_3Y_4 \tag{4.2-3}$$

$$Y_2 = \ddot{x} = \frac{1}{A_2}(C - A_1Y_3 - A_{10}Y_1 - A_0Y_4 - BY_5) \tag{4.2-4}$$

$$Y_3 = \dot{x} = \int_0^t \ddot{x}\,dt = \int_0^t Y_2\,dt \tag{4.2-5}$$

$$Y_4 = x = \int_0^t \dot{x}\,dt = \int_0^t Y_3\,dt \tag{4.2-6}$$

$$Y_5 = \int_0^t x\,dt = \int_0^t Y_4\,dt \quad . \tag{4.2-7}$$

The control components are defined by the rightmost expression. Thus, the third-order, nonlinear equation is defined by a multiplication, an addition-subtraction, and three integration components. Note that the above expressions cannot be rearranged so that all control variables are defined on the left before being used as operands on the right. The above order is recommended for the current numerical scheme. Assuming zero as the initial value for all the quantities, no initialization (no initial conditions calculation), and that the integral should be limited between zero and one (no reason except to demonstrate the input), input cards in the alternate format would be the following:

```
20500010  XD1*X  MULT  1.0  0.0  0
20500011  CNTRLVAR,3  CNTRLVAR,4
20500020  XD2  SUM  (1.0/A2)  0.0  0
20500021  (C)  (-A1),  CNTRLVAR,3  (-A10),  CNTRLVAR,1
20500022  (-A0),  CNTRLVAR,4  (-B),  CNTRLVAR,5
20500030  XD1  INTEGRAL  1.0  0.0  0
20500031  CNTRLVAR,2
20500040  X  INTEGRAL  1.0  0.0  0
20500041  CNTRLVAR,3
20500050  "INT OF X"  INTEGRAL  1.0  0.0  0 3,0.0,1.0
20500051  CNTRLVAR,4
```

4.2.3 Shaft Control Component (SHAFT)

The shaft component is a specialized control component that computationally couples motor, turbine, pump, and generator components analogously to a shaft mechanically coupling these devices. The primary purpose for the shaft component is to couple multiple turbine hydrodynamic components to represent a multi-stage turbine with vapor/gas extraction and liquid drain lines and to allow the turbines to drive a pump or generator. Computations associated with the shaft are advanced in time in the same manner as other control components. The shaft component evaluates the rotational velocity equation as

$$\sum_i I_i \frac{d\omega}{dt} = \sum_i \tau_i - \sum_i f_i \omega + \tau_c \qquad (4.2\text{-}8)$$

where I_i is the moment of inertia from component i, ω is the rotational velocity, τ_i is the torque from component i, f_i is the friction from component i, and τ_c is an optional torque from a control component. The summations are over the pump, compressor, generator, motor, or turbine components that might be connected to the shaft and the shaft itself. The rotational velocity is considered positive when rotating in the normal operating direction. A torque is positive when it would accelerate the shaft in the positive direction. In their normal operating modes, motors and turbines would generate positive torque; and pumps, compressors, and generators would have negative torque.

Each component contains its own model, data, and storage for inertia, friction, and torque and has storage for its rotational velocity. For example, the pump model allows cubic expressions for inertial and friction. The friction expression shown in Equation (4.2-8) is used for the shaft itself and the generator component. Each component also has a disconnect trip number. If zero (no trip), the component is always connected to the shaft. If a trip is specified, the component is connected when false and disconnected when true. Any disconnected component is advanced separately and, thus, can have a different rotational velocity than the shaft. All connected components have the same rotational velocity.

The shaft equation is advanced explicitly by

$$\sum_i I_i^n \frac{(\omega^{n+1} - \omega^n)}{\Delta t} = \sum_i \tau_i^n - \sum_i f_i^n \omega^n + \tau_c \tag{4.2-9}$$

where superscripts indicate time levels. Inertias, torques, and friction are evaluated using old time information. The torque from the control system, τ_c, would be in terms of new time values for quantities other than control variables and would use new or old time values for control variables, depending on their component numbers relative to the shaft component number. Except when a generator component is involved, the shaft component calculations consist of solving Equation (4.2-9) for ω^{n+1} separately for each component disconnected from the shaft (if any) and for the shaft and the connected components as one system. For separated components, the new rotational velocity is stored with the component data, and the summations are only over terms within the component. (Each component has only one term, except the pump/motor component, which has two terms.) For the shaft and the connected components, the new rotational velocity is stored as the rotational velocity of the shaft and each connected component.

The following sections discuss the components that can be connected to a shaft.

4.2.3.1 Motor Component. No separate motor component exists in RELAP5-3D©. A motor capability is an optional feature of a pump component, and input describing the motor features are entered as part of the pump input. Specifying a pump as being connected to a shaft includes the motor if it is described in the pump input.

A motor model can also be described though the control system and its torque applied to the shaft through a control variable [τ_c in Equation (4.2-9)].

4.2.3.2 Pump Component. A pump need not be connected to a shaft, since the pump component optionally includes a model for advancing the angular velocity equation. A review of the pump when not associated with a shaft follows, so that the pump with a shaft can be described by their differences.

4.2.3.2.1 Pump Not Associated with Shaft--A pump rotational velocity table and associated trip may be entered. If a rotational velocity table is entered, its use depends on the optional trip. If the trip is not entered, the table is always used; if the trip is entered, the table is used when the trip is true and not used when the trip is false. The dependent variable of the table is rotational velocity. The search variable

may be time or any other variable allowed in minor edits, including control variables. This allows a model for pump velocity to be computed by the control system. A motor is implied by the table, since a torque is needed to match the friction and hydrodynamic torque and to accelerate the pump velocity from the previous time-step value. The torque from this implied motor is labeled by MTR.TORQUE in the pump output of major edits.

When the pump speed table is not being used or is not entered, the pump rotational velocity equation is used,

$$I \frac{d\omega}{dt} = \tau_m + \tau_h \tag{4.2-10}$$

where I is the moment of inertia of the pump, ω is the rotational velocity, τ_m is the pump motor torque, and τ_h is the sum of the frictional and hydrodynamic torques. An operational pump trip may be specified. If not specified (trip number is zero) or if specified and false, electric power is supplied to the pump motor. If the trip is true, the pump breaker has tripped. (This is the origin of the name trips for the Boolean logic in RELAP4, and the name has been continued in RELAP5-3D©.) No electric power is supplied to a tripped pump, and, thus, the motor torque, τ_m, is zero.

A pump motor is directly specified when a table of pump motor torque versus rotational velocity is entered. An induction motor can be modeled by entering a function similar to that shown in Volume I. The key features of an induction pump are the negative slope of the torque with respect to velocity near the synchronous velocity, and the fact that the torque is zero at the synchronous velocity. In steady-state, the velocity is slightly less than the synchronous speed such that a positive torque balances the negative torque imposed by the pump. Pump transients such as pump startups from rest to operating speeds can be modeled. A simple AC or DC motor could also be modeled by a table that would have only positive torque values and negative slope. The motor torque table is not searched when the pump trip is true, since the motor torque is always zero when the pump is tripped. The motor torque is labeled by MTR.TORQUE in the pump output in major edits.

If a motor torque table is not entered, a pump motor is implied. When the pump trip is true, the torque from the implied pump motor is zero. If the trip is not entered or is false, a motor torque is assumed that is equal to the sum of frictional and hydrodynamic torques, resulting in no change to the rotational velocity over the time step. In this mode, the field labeled MTR.TORQUE has the same magnitude as the pump torque but has opposite sign.

The implied pump motor is normally used in cases where the pump is initially operating at normal velocity and, if tripped, is never restarted. Note that with the implied motor, if the pump trip is set true (pump tripped), the pump is free to change velocity. If the pump trip is reset to false (pump trip reset), the rotational velocity remains at the previous time step velocity; it is not reset to the initial velocity. To return to the initial velocity, the pump rotational velocity table can be used.

Optional input can prevent reverse rotation and stop the pump based on elapsed time and exceeding a maximum rotational speed in either direction.

4.2.3.2.2 Pump Associated With Shaft--Optional pump component input can be entered to associate the pump component with a shaft component. When a pump is associated with a shaft component, the rotational velocity is computed by the shaft component logic and not by the pump logic. The following describes the differences in pump logic when the pump is associated with a shaft.

The pump speed table cannot be entered. The options to prevent reverse velocity and to stop the pump based on time or velocity also cannot be used.

With one exception, the motor torque computation using either the motor torque table or the implied motor with a shaft component is identical to that without a shaft. If no components other than the pump are attached to the shaft, the moment of inertia of the pump-shaft combination is equal to that of the pump alone. Identical results can be obtained with or without using the shaft. The shaft must have a nonzero moment of inertia; to have the inertia of a pump alone equal that of the pump-shaft combination, some of the pump inertia must be apportioned to the shaft.

The one exception noted above is that with an implied pump motor (no motor torque table entered) and no pump trip entered (trip number is zero), the implied motor torque is always zero. This same situation without the shaft generates motor torque sufficient to maintain constant velocity. This option with the shaft forces the pump motor torque always to be zero. It would be used when a turbine is attached to the shaft, or the torque is computed by the control system.

The pump and shaft components offer several options; and, in some cases, the same model can be specified in more than one manner. Some general application recommendations follow. For motor-driven pumps that are either on or off (untripped or tripped), use the pump component without a shaft. For a variable-speed pump where the speed is computed by the control system, use a pump component with a *one-to-one* velocity table. The one-to-one table is a stratagem for forcing pump velocity to be equal to a control variable. Specify the search variable to be the control variable containing the velocity and enter a two-point velocity table. The independent and dependent variable for each point are the same. The first point is for the minimum possible velocity; the second point is for the highest velocity expected. The output from the table lookup and interpolation is just the input search argument. For a motor-driven, variable-speed pump where the torque is computed by the control system, use the shaft component. For a turbine-driven pump, use a shaft with the pump and turbine stages attached.

4.2.3.3 Compressor Component. A compressor need not be connected to a shaft, since the compressor component optionally includes a model for advancing to angular velocity equation. The input for a compressor has features similar to a pump. See Section 4.2.3.2 for a pump (for both associated with a shaft and not associated with a shaft) as this section has many features that are appropriate to a compressor.

4.2.3.4 Turbine Component. A turbine component is a hydrodynamic component consisting of one volume and has additional modeling to compute torque based on volume conditions and rotational velocity. One junction may connect to the turbine volume inlet to represent the steam line. Multiple

junctions may connect to the outlet to represent vapor/gas exit, extraction vapor/gas for regenerative heating of feedwater, and drain lines to remove liquid. A small turbine, such as might be used to drive a pump, is usually modeled by one turbine component. The turbine used to drive the electrical generator typically has steam extraction points and drain lines and, thus, is usually modeled by two or more turbine components. The shaft component is the only mechanism for providing the rotational velocity common to each turbine component and summing the torque developed in each turbine component. The shaft is also the only mechanism to couple the turbine to a pump or generator.

4.2.3.5 Generator Component. The generator component consists of the minimum model to load a turbine. Because of the simple model and its small input data requirements, it has been made an option of the shaft component.

The generator model allows two operating modes. One mode is having the generator connected to a large electrical grid; the generator, the shaft, and other connected components are forced to the synchronous speed. The other mode is tripped, and the rotational velocity then responds to the torques applied to the shaft. When the generator is connected to the grid, the torque necessary to maintain synchronous velocity is computed and the generator power is that torque times the synchronous velocity. If the torque is negative, the generator is in its normal mode of generating electricity. If the torque is positive, the generator is acting as a synchronous motor and power is being drawn from the grid to maintain the synchronous velocity. When the generator is tripped, the generator torque is zero.

A generator can be connected to a pump through the shaft component. This allows a synchronous motor-pump combination, which is yet another pump-motor option that can yield results identical to the pump without a shaft using an implied motor.

4.2.3.6 Pump, Generator, and Shaft Sample Problem. **Table 4.2-1** shows input data for a sample problem to test pump, generator, and shaft components. The test problem consists of two identical but separate loops. Each loop has a pump and a pipe connecting the pump discharge to the pump suction. The normal wall friction model is used, and an orifice is included for additional dissipation. The loops are filled with subcooled liquid at zero velocity. The two pumps are driven differently. The first pump uses an implied pump motor operating at normal speed. The liquid is accelerated to near steady-state velocity within a few seconds. A true steady-state is not possible, since there is no provision for removing dissipation heat. The pump is then tripped, the pump coasts down, and flow velocities diminish. The second pump uses a pump motor torque table representing an induction motor with the rotational velocity initially zero. The pump accelerates to near the synchronous velocity, and, in turn, the liquid velocity is accelerated similarly to the first loop. The second loop is tripped similarly to the first loop.

In the second problem, the pumps are driven identically but using a different mechanism. The first pump uses a shaft and a generator acting as a motor. The second pump uses a shaft and control system to develop the torque. A general table duplicates the motor torque table, and a unit trip applies the trip action. Identical results are obtained in the two cases.

.

Table 4.2-1 Input data for a sample problem to test pump, generator, and shaft.

```
=two loops with pumps
*
* The first loop is similar to the pump problem. The second loop uses pump motor torque data to
* represent an induction motor. The pump is initially at rest. The pump accelerates to near synchronous
* speed, and fluid is accelerated. Reaching near steady-state, pump trip and decreasing pump speed and
* fluid velocity are similar to pump. The second problem is identical to the first except that shaft and
* generator (acting as a motor) components are used.
*
100 new transnt
102 british british
104 none
201   1.0 1.0-6 0.010 15001 1 20 1000
202 40.0 1.0-6 0.200 15001 1 20 1000
301 p 1010000
302 p 1040000
303 p 1060000
304 p 1070000
305 p 1100000
306 p 1150000
307 p 1180000
308 p 2010000
309 velfj 1010000
310 velfj 1070000
311 velfj 1180000
312 velfj 2010000
313 velfj 2020000
314 pmpvel 002
```

Table 4.2-1 Input data for a sample problem to test pump, generator, and shaft. (Continued)

```
315 pmphead 002
316 pmptrq 002
351 p 3010000
352 p 3040000
353 p 3060000
354 p 3070000
355 p 3110000
356 p 3150000
357 p 3180000
358 p 4010000
359 velfj 3010000
360 velfj 3070000
361 velfj 3180000
362 velfj 4010000
363 velfj 4020000
364 pmpvel 004
365 pmphead 004
366 pmptrq 004
501 time 2 q null 0 20.2 1
10000 loop pipe
10001 19
10101 0.0376, 19
10201 0.0376, 6 0.01, 7 0.0376, 18
10301 2.0, 19
10601 0.0, 4 90.0, 9 0.0, 14 -90.0, 19
10801 0, 0, 19
11001 0, 19
11101 0, 6 100, 7 0, 18
```

Table 4.2-1 Input data for a sample problem to test pump, generator, and shaft. (Continued)

```
11201 3, 2244.780, 540.0, 0, 0, 0, 19
11301 0, 0, 0, 18
20000 loop pump
20101 0.0468 0 0.1660 0 0 0 0
20108 1010000 .0376 0 0 0
20109 1000000 .0376 0 0 0
20200 3 2264.78 540.0 0
20201 0 0 0 0
20202 0 0 0 0
20301 -1 0 -2 -1 -1 501 -1
20302 3560.0 0.66573 180.0 192.0 34.8 38.3 62.3 0 6.7 0 0 0
23000 0
23001 0.0, 0.0 0.1 0.0 0.15, 0.05 0.24, 0.8 0.3, 0.96 0.4, 0.98
23002 0.6 0.97 0.8, 0.9 0.9 0.8 0.96, 0.5 1.0, 1.0
23100 0
23101 0.0, -.17 0.0001, -.017 0.0006, 0.0 0.1, 0.0 0.15, 0.05
23102 0.24, 0.56 0.8, 0.56 0.96, 0.45 1.0, 0.0
30000 loop2 pipe
30001 19
30101 0.0376, 19
30201 0.0376, 6 0.01, 7 0.0376, 18
30301 2.0, 19
30601 0.0, 4 90.0, 9 0.0, 14 -90.0, 19
30801 0, 0, 19
31001 0, 19
31101 0, 6 100, 7 0, 18
31202 3, 2265.780, 540.0, 0, 0, 0, 19
31301 0, 0, 0, 18
```

Table 4.2-1 Input data for a sample problem to test pump, generator, and shaft. (Continued)

```
40000 loop2 pump
40101 0.0468 0 0.1600 0 0 0 0
40108 3010000 .0376 0 0 0
40109 3000000 .0376 0 0 0
40200 3 2244.78 540.0 0
40201 0 0 0 0
40202 0 0 0 0
40301 2 2 2 0 -1 501 1
40302 3560.0 0.0 180.0 192.0 34.8 38.3 62.3 35.0 6.7 0 0 0
46001 1440., 1.00 2160., 1.10 2880., 1.50
46002 3526., 2.80 3672., -2.70 4320., -1.90
46003 5040., -1.20 5760., -1.05 6480., -1.00
46004 7200., -0.98
.end of first case
=two loops with pumps using shaft component
20301 -1 0 -2 -1 -1 0 1
20302 3560.0 0.66573 180.0 192.0 34.8 37.0 62.3 0 6.7 0 0 0
20309 20
40301 2 2 2 -1 -1 0 0
40302 3560.0 0.0 180.0 192.0 34.8 38.0 62.3 0 6.7 0 0 0
40309 10
46001
46002
46003
46004
20500100 mtr.trq function 47.4536282 0 0
20500101 cntrlvar, 10 10
20500200 trip tripunit 1.0 1.0 0
```

Table 4.2-1 Input data for a sample problem to test pump, generator, and shaft. (Continued)

```
20500201 -501
20500300 torque mult 0.7375621495 0 0
20500401 cntrlvar, 3
20501000 shaft4 shaft 1.0 0.0 0
20501001 3 0.3 0.0 pump, 4
20201000 reac-t 0 0.10471975512 1.0
20201001 1440.0, 1.00 2160.0, 1.10 2880.0, 1.50
20201002 3528.0, 2.80 3672.0, -2.70 4120.0, -1.90
20201003 5040.0, -1.20 5760.0, -1.05 6480.0, -1.00
20201004 7200.0, -.098
20502200 shaft2 shaft 1.0 2370.0 1
20502001 0 1.0 0.0 pump, 2 generatr, 20
20502006 1800.0 2370.0 0.3 0.0 501 0
.end of job
```

5 Reactor Kinetics

The reactor kinetics capability can be used to compute the power behavior in a nuclear reactor. The power is computed using the space-independent or point kinetics approximation, which assumes that power can be separated into the product of space and time functions. This approximation is adequate for those cases in which the space distribution remains nearly constant. There is also an option to use a multi-dimensional neutron kinetics option.

Reactor kinetics data may be entered for new or restart problems. In restart problems, reactor kinetics data completely replaces previous reactor kinetics data if present; thus, all needed data must be entered even if they duplicate existing data.

5.1 Power Computation Options

For point kinetics, data for the six generally accepted delayed neutron groups are built into the code. Optionally, yield ratios and decay constants for up to 50 groups may be entered.

The total reactor power is the sum of immediate (prompt and delayed neutron) fission power and the power from decay of fission fragments. The immediate (prompt and delayed neutron) power is that released at the time of fission and includes power from fission fragment kinetic energy, prompt gammas, and neutron moderation. Decay power is generated as the fission products undergo radioactive decay. The user can specify one of three options for computing reactor power: fission power only; fission and fission product decay product power; or fission, fission product decay, and actinide decay power. Actinide decay power is the power resulting from production of ^{239}U by neutron absorption in ^{238}U and subsequent two-stage beta decay to ^{239}Pu.

Four sets of fission product decay data are built into the code. The default set is an approximation to the eleven-group 1973 ANS Proposed Standard.[5.1-1] The other set of data is the exact 1979 ANSI/ANS Standard.[5.1-2,5.1-3,5.1-4] The 1979 Standard specifies data for three isotopes, ^{235}U, ^{238}U, and ^{239}Pu, using 23 groups for each isotope. To use the three-isotope data, the user must furnish the fraction of power produced by each isotope. An option exists to use only the ^{235}U isotope data from the 1979 Standard. The third set of data is the exact 1994 ANSI/ANS Standard[5.1-5]. The 1994 Standard specifies data for four isotopes, ^{235}U, ^{238}U, ^{239}Pu, and ^{241}Pu, using 23 groups for each isotope. The fourth set of data is the exact 2005 ANSI/ANS Standard[5.1-6]. The 2005 Standard specifies data for four isotopes, ^{235}U, ^{238}U, ^{239}Pu, and ^{241}Pu, using 23 groups for each isotope. To use the four isotope data, the user must furnish the fraction of power produced by each isotope. An option exists to only use the ^{235}U isotope data for the 1994 or 2005 Standards. Actinide data are from the 1979 Standard, the 1994 Standard, and the 2005 Standard, which are the same for actinides. An input fraction is applied to both the fission product and actinide yield data. For fission products, the factor is usually 1.0 for best-estimate calculations; and 1.2 has been used for conservative calculations with the approximation to the 1973 data. For actinide data, the factor is the ratio of ^{238}U atoms consumed per ^{235}U atoms fissioned; but additional conservative factors can be applied.

Fission product and actinide data can also be entered by the user. Earlier versions of the code did not include the G factor, which is part of the 1979 Standard, the 1994 Standard, and the 2005 Standard. This factor can now be optionally included and should be for long-term transients.

The built-in data for delayed neutrons, fission products, and actinides are recommended and are listed in the reactor kinetic input edit when used. Use of the fission power plus fission product decay power is recommended, as is actinide decay power if an appreciable amount of ^{238}U is present. The 2005 Standard is recommended because it is an approved standard. The four-isotope option is recommended unless the power fractions for each isotope are not available.

The reactor kinetics major edit output lists total reactor power, immediate (prompt and delayed neutrons) fission power, decay (fission product and actinide) power, reactivity, and reciprocal period. Either the total power, immediate (prompt and delayed neutrons) fission power, or decay (fission product and actinide) power can be specified as the time-varying part of the heat source in heat structures.

5.1.1 References

5.1-1. American Nuclear Society Proposed Standard, ANS 5.1, *Decay Energy Release Rates Following Shutdown of Uranium-Fueled Thermal Reactors*, October 1971, revised October 1973.

5.1-2. *American National Standard for Decay Heat Power in Light Water Reactors*, ANSI/ANS-5.1-1979, August 1979.

5.1-3. V. E. Schrock, "A Revised ANS Standard for Decay Heat from Fission Products," *Nuclear Technology, 46*, 1979, pp. 323-331.

5.1-4. V. E. Schrock, "Evaluation of Decay Heating in Shutdown Reactors," *Progress in Nuclear Energy, 3*, 1979, pp. 125-156.

5.1-5. *American National Standard for Decay Heat Power in Light Water Reactors*, ANSI/ANS-5.1-1994, August 1994.

5.1-6. *American National Standard for Decay Heat Power in Light Water Reactors*, ANSI/ANS-5.1-2005, April 2005.

5.2 Reactivity Feedback Options

Five reactivity feedback options are provided. One assumes separability of feedback effects; the others use three- or four-dimensional table lookup and linear interpolation. The defining equations are given in Volume I of this manual. Note that the sign of the feedback terms is positive. Negative quantities must be entered where negative feedback is desired. All options include an input reactivity, r_o, a bias reactivity, r_B, and sums over scram curves and control variables.

The quantity r_o is an input quantity and is the reactivity corresponding to the assumed steady-state reactor power at time equal to zero. This quantity must be ≤ 0. A nonzero quantity indicates that a neutron source is present. For most applications, $r_o = 0$ is acceptable.

The bias reactivity, r_B, is calculated during input processing such that $r(0) = r_o$. The purpose of the bias reactivity is to ensure that the initial reactivity is still equal to the input reactivity after including the feedback effects. Without this quantity, the user would have to manually adjust a scram curve or control variable to obtain the input value of initial reactivity or have a step input of reactivity as the transient starts. The bias reactivity, r_B, is printed out during input processing.

The scram curves are obtained from general tables defining reactivity as a function of time. Each table can have an associated trip number. If the trip number is not entered or zero, time is the search argument. If the trip number is nonzero, the search argument is -1.0 if the trip is false. If the trip is true, the search argument is time minus the time at which the trip last turned true. These tables can be used to describe reactivity changes from rod motion.

Control variables can be defined to represent power control systems or to implement alternate feedback models. However, reactor kinetics advancement precedes control system evaluation; thus, feedback from control variables is delayed one time step.

The separable option uses two tables, one defining reactivity as a function of volume moderator fluid density and the other defining reactivity as a function of heat structure volume average fuel temperatures. The tables allow nonlinear feedback owing to moderator fluid density and fuel temperature changes. A constant temperature coefficient allows for linear moderator fluid temperature feedback, and an additional linear fuel temperature feedback is provided. The separable option is so named because of the assumption that each feedback mechanism is independent and the total reactivity is the sum of the individual effects. The separable option does not directly allow boron feedback, but boron effects can be modeled through the control system.

Data for the separable option can be obtained from reactor operating data, reactor physics calculations, or a combination of the two. The required moderator fluid temperature coefficient is not the usually quoted quantity. Assume the moderator feedback is a function of fluid density and fluid temperature, $r(\rho, T)$, and fluid density is a function of temperature, $\rho(T)$. The usual temperature coefficient is the total derivative, $\frac{dr}{dT}$. The input requires partial derivatives. The moderator fluid density feedback is $\left.\frac{\partial r}{\partial \rho}\right|_T$; the moderator fluid temperature coefficient is $\left.\frac{\partial r}{\partial T}\right|_\rho$.

The three- and four-dimensional table lookup and interpolation option uses three or four quantities as the independent variables. The four-dimensional table includes the effects of boron; the three-dimensional table does not include boron effects. Two suboptions allow a choice of independent variables. One choice is reactivity as a function of moderator fluid density, void weighted moderator temperature, heat structure

volume average fuel temperature, and boron density. This option uses the same variables as the separable option plus boron effects if four variables are used. The other option uses void fraction, liquid moderator temperature, heat structure volume average fuel temperature, and boron concentration as independent variables. Feedback effects in light water power reactors are usually expressed in terms of these quantities. The multi-dimensional interpolation allows nonlinearities and interaction of feedback effects but burdens the user with obtaining a larger amount of reactivity data. As with the separable option, required data can be obtained from plant data or reactor physics calculations. As discussed in Volume I, a data point must be entered for each combination of coordinate values. Accurate reactivity data need only be entered for points near zero reactivity. Once the shutdown reactivity decreases below -2.0 dollars, little change in fission energy release occurs with further decreases in reactivity. Thus, in sections of the multi-dimensional table where reactivity is known to be very much shut down, data can be determined from extrapolation and need not be accurate. Similarly, some parts of the table may contain large values of reactivity. The user does not expect the transient to use this portion of the table, but the code input requires all tabular points to be entered. Again, accurate data need not be entered; if the transient should enter this area, the large power rises will be evident and the user can investigate the modeling difficulty. In some instances, a coordinate value is introduced to ensure accuracy in one section of a table, but the detail is not needed in other parts of the table. Where the detail is not needed, data could be obtained at a more coarse mesh, and the user can interpolate to meet the input requirements of the code.

Usually, several hydrodynamic volumes are used to represent the coolant channels in a reactor core and several heat structures represent the fuel pins. Weighting factors are input to specify the reactivity contribution of each hydrodynamic volume and heat structure to the total. Reactivity feedback is usually defined such that the weights for volumes and heat structures each should sum to one. The code does not check that the weights sum to one.

The use of the weights is different between the separable and table options. In the separable option, a reactivity effect is first computed for a volume or for a heat structure. Then its contribution to the total reactivity is obtained by multiplying the effect by the weighting factor. This order is reversed for the table option.

Weighted-averaged independent variables for table lookup and interpolation are obtained by using volume or heat structure values and the weighting factors. Table evaluation for total feedback uses the averaged values. It is possible to define a table equivalent to the separable data. However, slightly different transient results would be obtained using the equivalent data owing to the difference in application of the weighting factors.

In steady-state problems, the user usually wishes to specify reactor power. If reactivity feedback data are entered, reactor power will vary as the reactor system moves toward a steady-state condition. To prevent this, a control system could be defined to adjust reactivity to maintain constant power. A simpler alternative is to omit reactivity feedback in steady-state, and the reactor power will remain constant at the input value. At the restart to start the transient, the original reactor kinetics data plus feedback data can be entered.

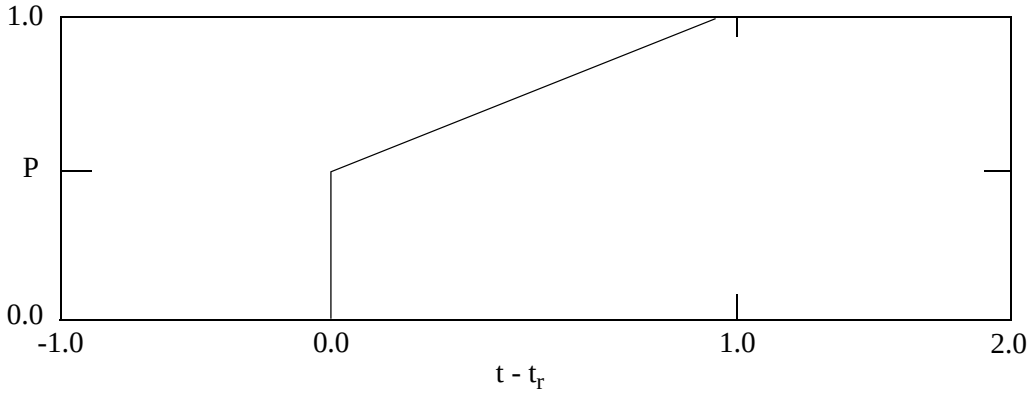
6 General Tables and Component Tables

General tables provide data for several models, including heat structures, valves, reactor kinetics, and control systems. The general table input provides for the following tables: power versus time, temperature versus time, heat flux versus time, heat transfer coefficient versus time, heat transfer coefficient versus temperature, reactivity versus time, and normalized valve area versus normalized stem position. An input item identifies each table so that proper unit conversion and input checking can be done. For example, specifying a temperature table when a power table is required is detected as an error. Because these tables are often experimental data, or scaling may be needed for parametric studies, the input provides for conversion and/or scaling factors for these tables. Input editing of these tables includes both the original and scaled data. General tables can be entered, deleted, or replaced at restart. The tables are linearly interpolated between table values, and the end-point values are used when the search arguments are beyond the range of entered data. The reactivity versus time table is often used for other cases (i.e., FUNCTION type control system, hydrodynamic pressurizer and feedwater components) since no British to SI units conversion is done for this table. If the reactivity versus time table is used, the input editing of the printed output will show the label ’reactivity’, whether it is used for reactivity or not.

Figure 6.0-1 shows input data for a power-type general table, and the graph shows its time history. The first entry of the first line indicates that it is a power versus time table, the second entry, indicates that the trip number for the table is trip number 605, the third entry indicates that the time values are to be used as entered, i.e., multiplied by 1.0, the last entry indicates that the power values that are input should be scaled by 50.0 + 6, i.e., 50 MW. The search logic is determined by the trip input. A nonzero trip number specifies the following logic: when the trip is false, the table is interpolated using a search argument of -1.0, resulting in a power of zero up to the trip time t_r; when the trip is true, the table is interpolated with search argument $t - t_r$, effectively shifting the origin of the table to time t_r. This is analytically equivalent to the application of a unit step function and delay. If a zero trip number is specified, current time is always the search argument. The remaining input defines the time history. The data are input as x-y pairs of time and power. The tabular data show two data points having the same time value, zero, but having different power values. This allows entry of step changes, as shown on the graph. The graph also illustrates that when search arguments are beyond the range of entered data, endpoint values are used rather than extrapolation.

Entry of a nonzero trip number in general tables is valid only when time is the independent variable.

Time-dependent volumes, time-dependent junctions, and pump angular velocity tables are examples of component tables. These tables provide for entry of a trip number and in the default mode use time as the independent variable. In this mode, the use of the trip time is identical to that described for time-dependent general tables. But the time-dependent volumes, junctions, and pump velocity tables also permit any time-advanced quantity to be specified as the independent variable. If a trip is specified and is false, the table is interpolated with -1.0×10^{308} as the search argument. If no trip is specified, or the trip is true, the specified time-advanced quantity is the search argument.



20201000	Power	605	1.0	50.0+6
20201001	-1.0, 0.0	0.0, 0.0		
20201002	0.0, 0.5	1.0, 1.0		

Figure 6.0-1 Input data for a power-type general table and graph.

A typical use of tables using a quantity other than time as the independent variable is the modeling of a high- or low-pressure reactor safety injection system. Rather than model the valve, pump, and motor for the system, a time-dependent junction is used to approximate the injection system. The pressure at the injection point is specified as the independent variable, and flow rate is the dependent variable. The table would define zero flow for the first zero pressure value, then appropriate flow rates for the second zero pressure and following pressure values. The last pressure value would be the cutoff pressure of the pump and have a corresponding zero flow. In normal reactor operation, the trip would be false, and the table interpolation would return zero flow. When the safety system is actuated, flow may still be zero if the reactor pressure exceeds the cutoff pressure. As the reactor pressure drops, flow would start; and the table could indicate increasing flow with decreasing pressure, possibly up to a maximum flow rate. The source of injection liquid is usually a time-dependent volume. This technique would not add pump work to the injected fluid. Some approximation of the pump work could be made by also specifying the injection point pressure as the independent variable of the time-dependent volume and entering appropriate thermodynamic conditions as dependent variables.

7 Initial and Boundary Conditions

All transient analysis problems require initial conditions from which to begin the transient simulation. Usually, the initial conditions will correspond to a steady-state, with the transient initiated from a change of some boundary condition. In general, the initial conditions required are a determinate set of the dependent variables of the problem. The hydrodynamic model requires four thermodynamic state variables in each volume and the velocities at each junction. Heat structures require the initial temperature at each node, control systems require the initial value of all control variables, and kinetics calculations require initial power and reactivity. All of these parameters are established through the code input and initialization process for a new problem. For a restart problem, the values are established from the previous calculation. For restart with renodalization or problem changes, the initialization will result from a combination of the two processes, and care must be exercised to ensure that the input values are compatible with those from the restart, especially if an initial steady-state is to be simulated.

Boundary conditions may be required for hydrodynamic models, heat structures, or control components if these parameters are governed by conditions outside of the problem boundaries. Examples of these could be mass and energy inflows or an externally specified control parameter.

Obtaining a desired simulation is very dependent upon proper specification of initial and boundary conditions. The purpose of this section is to summarize recommended approaches for these specifications.

7.1 Initial Conditions

All variables of the problem that are established by integration require initial values in order to begin a calculation or simulation. Problem variables related to the integration variables through quasi-steady relationships do not require initial conditions, since they can be established from the initial values required for the integration variables. An example is the pump head, which is related to the pump flow and speed, both of which are obtained by integration. Thus, the initial conditions for pump flow and speed must be specified.

7.1.1 Input Initial Values

Input initial values are required in order to begin a new problem regardless of whether a steady-state or a transient run is specified. These initial values are supplied by the user through input for each component. (Heat structures are an exception and can be initialized either by input or by steady-state initialization using the heat structure boundary conditions at time zero.)

The hydrodynamic volume components have seven options for specifying the volume initial conditions (see Section 2.4.1 for more detail). Four options are provided for pure vapor/liquid systems, and the remaining three options allow noncondensables. Boron concentration can be specified with all seven options by adding 10 to the control word, Word W1(I). Regardless of what option is used, the initialization computes initial values for all primary and secondary dependent variables. The primary variables are

pressure, void fraction, two phasic specific internal energies, noncondensable quality, and boron concentration. Secondary variables are static quality, two phasic densities, two phasic temperatures, etc.

The most common specification will be an equilibrium condition for the vapor/liquid system. The options 1-3 [i.e., control word W1(I) on single volume Card CCC0200, in Appendix A] are equilibrium specifications using temperature and static quality, pressure and static quality in the equilibrium condition, and pressure and temperature. The first two conditions are valid combinations for single- (at the saturation point) or two-phase saturated conditions. The third combination is valid for single-phase nonsaturated or saturated conditions. When noncondensable gas is specified, it is best to use conditions of 100% humidity or saturated noncondensable gas at the initial pressure and temperature of the system. The specification of dry noncondensable gas can cause numerical difficulties when mixing with liquid or vapor occurs. If noncondensable gas is the only system component and mixing with liquid or vapor does not occur, the specification of pure noncondensable gas will cause no problems.

Heat structure initial temperatures must be input. Depending upon the initialization option selected, these temperatures are either used as the initial temperatures or as the initial guess for an iterative solution for a steady-state temperature profile. The iteration solution will attempt to satisfy the boundary conditions and heat sources/sinks that have been specified through input. Some care is needed, since an indeterminate solution can result from specification of some boundary conditions (e.g., a two-sided conductor with specified heat fluxes). If the initial temperature of a heat structure is unknown, it is generally safer to use the steady-state option and supply as a first guess a uniform temperature distribution equal to the temperature of a hydrodynamic volume to which it is connected. In the case of a two-sided structure, either side may be selected. The steady-state solution algorithm will rapidly converge to a steady-state temperature distribution.

Initial conditions must be specified for each control component used, even if the option to compute the initial condition is selected. As stated above, only the integral functions should require initial conditions. However, since control components are initialized using a sequential single-pass solution scheme and since some control variables may be specified as arguments for other control variables, it is possible for some to be initially undefined. Hence, the initial value for all control variables must be specified. Also, the code does not check whether initial values are needed nor whether they are reasonable; thus, the user should always supply an accurate initial value.

The reactor kinetics model requires specification of an initial power and reactivity. Previous power history data may also be entered.

7.1.2 Steady-State Initialization

RELAP5-3D© contains an option to perform steady-state calculations (Card 100, Word 2 is STDY-ST, in Appendix A). This option uses the transient hydrodynamic, kinetics, and control system algorithms and a modified heat structure thermal transient algorithm to converge to a steady-state. The differences between the steady-state and transient options are that a lowered heat structure thermal inertia is used to accelerate the response of the thermal transient, and a testing scheme is used to check if

steady-state has been achieved. The testing scheme has been found to terminate calculations prematurely, so users usually disable the testing scheme through input in the time step cards. In code versions after version 2.2, activation of trips and CHF are bypassed by default in steady-state mode, and several solution control options are selected by default in steady-state mode. For these versions of the code, the default steady-state solution is obtained using the nearly-implicit hydrodynamic solution algorithm, the mass error time step control is bypassed, the steady state checking routine is bypassed, and the hydrodynamic and heat conduction solutions are coupled implicitly and use the same time step size. The user may override the steady-state default options to reactivate trips and CHF and use the solution controls found on the time step cards by using additional input data (Card 107, Words 1-3, in Appendix A). The results of the steady-state calculation are saved so that a restart can be made in the transient mode. In this case, all initial conditions for the transient are supplied from the steady-state calculation. It is also possible to restart in either the transient or steady-state mode from either a prior transient or steady-state run.

The user should be aware that use of the steady-state option provides a more optimum solution than simply running the problem as a transient and monitoring the results. This occurs because the code monitors results for the entire system, including the effects of calculational precision. Also, thermal inertia for the heat structures is generally quite large, so that for the transient option, the heat structure temperature distribution will not achieve steady-state in the time that a hydrodynamic steady-state can be achieved. Hence, use of the steady-state option will provide the user with a precise steady-state, including a precise heat structure steady-state.

It is still necessary to supply input specifying initial conditions for a steady-state run. However, the accuracy of the input data is less critical, since they are simply used as a starting point for convergence to a steady-state. The values used should be reasonable, however, since the closer they are to the actual steady-state, the shorter the calculation will be to achieve steady-state.

Once an initial steady-state is calculated, the user can save the restart-plot file and perform subsequent new steady-state runs using the previous steady-state results. This results in reduced calculational times for the subsequent runs and at the same time maintaining a complete set of steady-state initializations.

The steady-state initialization calculation is an open-loop calculation unless control functions are defined such that active control systems are used to obtain desired operating points. Active control is achieved using controlled variables such as pressure, flow rate, etc. The user must design and implement such control functions, and only a limited number of system parameters can be controlled independently. In this regard, the model behaves exactly as a real system, and, if a resistance to flow must be varied to achieve the desired steady-state, then a valve must be used with a controller. The use of a controller to achieve a desired steady-state can save considerable time compared to the process of open-loop control, in which a resistance or other parameter is varied from run to run until the desired steady-state is achieved.

In providing control systems and trips to drive the solution to steady-state, two rules of thumb must be considered, both of which revolve around the basic purpose of the steady-state run. The first rule is that if the run is to simulate the real behavior of a plant in achieving steady-state, then control systems and trips

simulating real plant controls or control procedures should be designed. However, the second rule of thumb is that if the run is simply to achieve a steady-state initialization of the system model, then controls not representative of the actual system may be designed that will drive the solution to steady-state in the fastest manner possible. The only restriction is that stability of the calculations must be maintained.

The default code contains a discrepancy when checking the steady-state by means of an energy balance. The default code should add the form loss (code calculated abrupt area change loss and user-supplied loss) dissipation to the phasic energies. This dissipation was removed in RELAP5/MOD2 because of temperature problems (i.e, overheating), and thus it is not present in RELAP5-3D©. The dissipation can be activated by the user in the input deck, however the user is cautioned that temperature problems may occur.

There is also an option, Word 4, digits tt, on the time step control cards, 201 through 299, which allows the user to select part of the steady-state calculation to be used. The thermal inertia of the heat structures is lowered, but the testing scheme to check the derivatives of variables to determine a steady-state is not used. This is the default behavior for code versions after version 2.2. This gives a user the advantage of using the artificially accelerated thermal steady-state in the heat structures while allowing use of either a set end time or else the user's own choice for a variable to monitor for a steady-state through a simple control system.

7.2 Boundary Conditions

Boundary conditions are required in most transient calculations. In reality, boundary conditions take the form of the containment atmosphere, operator actions, or mass and energy sources that are not explicitly modeled as part of the system. Such boundary conditions are simulated by means of time-dependent volumes for specified sources or sinks of mass, time-dependent junctions for specified flows, or specified heat structure surface heat fluxes and energy sources. Specified variation of parameters in control components to simulate an operator action may also be used. The time variation of the boundary conditions is specified by input tables that can also be varied dynamically by using trips.

7.2.1 Mass Sources or Sinks

Hydrodynamic mass sources or sinks are simulated by the use of a time-dependent volume with a time-dependent junction. The thermodynamic state of the fluid is specified as a function of time by input or by a control variable. The thermodynamic state is needed for inflow because the densities and specific internal energies are needed in the donored flux terms in the density and energy equations. The time-dependent junction mass flow rates or velocities are also specified. This approach can be used to model either an inflow or an outflow condition; however, care is required in modeling outflows. A time-dependent junction is analogous to a positive displacement pump in that the flow is independent of the system pressure. In the case of outflow, it is possible to specify a greater outflow than inflow to a volume or even outflow that will exhaust the volume. In this case, a numerical failure will result when the equivalent of a negative density is calculated. For this reason, modeling outflows using a time-dependent junction is not recommended.

It should be noted that for the case of a TMDPVOL (time-dependent volume) and TMDPJUN (time-dependent junction), the pressure upstream of the TMDPJUN in the TMDPVOL is neither changed nor checked by the code to be consistent with that derived from the pressure drop obtained in the momentum equation if the velocity option is used. The pressure upstream is the value that the user enters for the TMDPVOL. This is because this type of boundary condition is analogous to a positive displacement pump where the inflow rate is independent of the system pressure (see Volume II, Section 2 of the manual). An additional fact that should be considered when using a time-dependent junction as a boundary is that pump work is required for system inflow if the system pressure is greater than the time-dependent volume pressure. In particular, any energy dissipation associated with a real pumping process is not simulated.

7.2.2 Pressure Boundary

A pressure boundary condition is modeled using a time-dependent volume in which the pressure and thermodynamic state variables are specified as a function of time through input by tables or by a control variable. The time-dependent volume is connected to the system through a normal junction; thus, inflow or outflow will result, depending upon the pressure difference. Several precautions are needed when specifying a pressure boundary, since flow invariably accompanies such a boundary. First, the time-dependent volume conditions must represent the state of fluid that would normally enter the system for an inflow condition. Second, there are implied boundary conditions for a time-dependent volume in addition to the specified values. Third, only the static energy of an incoming flow is fixed by a time-dependent volume. The total energy will include the inflow kinetic energy that increases with increasing velocity.

The additional boundary conditions represented by a time-dependent volume concern the virtual viscosity terms inherent in the numerical formulation of the momentum equation (see Section 3 in Volume I for a detailed discussion). For this purpose, the derivative of velocity across the time-dependent volume is zero, and the length and volume are assumed to be zero (regardless of the specified input). The fact that the energy of inflow increases with velocity can lead to a nonphysical result, since the stagnation pressure also increases, and for a fixed system pressure, an unmitigated increase in inflow velocity can result. This effect can be avoided by making the cross-sectional area of the time-dependent volume large compared to the junction so that the volume velocity of the time-dependent volume is small, and thus the total energy of the inflow is constant. When a large area ratio exists between the time-dependent volume and the junction connecting it to the system, a reservoir or plenum is simulated. As a general rule, all pressure boundary conditions having either inflow or outflow should be modeled as plenums for stability and realism. In particular, when an outflow is choked, the critical flow model more closely approximates the conditions at a large expansion (i.e., little or no diffusion occurs). Thus, this assumption is consistent with the choked flow model and is, therefore, recommended.

Volume I of the manual indicates that linear and cubic interpolation between neighboring volume properties is used for junction properties (e.g., densities, volume fractions) that are not donored. These junction properties are used in the momentum equations. For the case of a regular junction (i.e., non-time-dependent junction) next to a time-dependent volume (i.e., the back pressure case), the

momentum equations are modified to set the junction properties equal to the properties from the volume connected to the junction that is not the time-dependent volume. This results in the correct situation where changing the outlet time-dependent volume properties will not change the calculation results by changing the momentum equations junction properties. Thus, the code does not allow overspecification of the boundary conditions for these cases.

8 Problem Control

8.1 Problem Types and Options

RELAP5-3D© provides for four problem types--NEW, RESTART, PLOT, and STRIP. The first two are concerned with simulating hydrodynamic systems; NEW starts a simulation from input data describing the entire system; RESTART restarts a previously executed NEW or RESTART problem. PLOT and STRIP are output-type runs using the restart-plot file written by NEW or RESTART problems. NEW and RESTART problems require an additional option to be selected, STDY-ST or TRANSNT.

A RESTART problem may restart from any restart record. A note indicating the restart number and record number is printed at the end of the major edit whenever a restart record is written. The restart number is equal to the number of attempted advancements and is the number to be used on Card 103 to identify the desired restart record. The record number is simply a count of the number of restart records written, with the restart record at time equal zero having record number zero. Quantities written in the restart-plot records by default are noted in the input data description. User-specified input can add additional quantities to the restart-plot records.

PLOT and STRIP are output-type runs. PLOT generates plots from data stored on the restart-plot file. The PLOT capability is not now operational but is still documented. The PLOT capability may be dropped from the code since NPA[8.1-1] and XMGR5,[8.1-2] an INL extension of XMGR,[8.1-3] allow very general and high quality plots of RELAP5-3D© results and associated information. STRIP writes selected information from a restart-plot file onto a new file. The new file consists of records containing time and the user-selected variables in the order selected by the user. Data to be plotted or stripped are limited to that written in the plot records on the restart-plot file.

8.1.1 References

8.1-1. D. M. Snider, K. L. Wagner, W. H. Grush, and K. R. Jones, *Nuclear Plant Analyzer, Volumes 1-4*, NUREG/CR-6291, INEL-94/0123, Idaho National Engineering Laboratory, December 1994.

8.1-2. K. R. Jones and J. E. Fisher, *XMGR5 Users Manual*, INEL/EXT-97-00346, Idaho National Engineering Laboratory, March 1997.

8.1-3. P. J. Turner, *ACE/gr User's Manual*, SDS3, 91-11, Beaverton, OR, 1992.

8.2 Time Step Control

Input data for time step control consist of one or more cards containing an end time, minimum time step, maximum (or requested) time step, control option, minor edit plot/frequency, major edit frequency, and restart frequency. The end time must increase with increasing card numbers. The information on the first card is used until the problem time exceeds the end time on the first card; then, the next card is used,

and so on. In restart problems, these cards may remain or may be totally replaced. Cards are skipped if necessary until the problem time at restart is properly positioned with regard to the end time values.

The minimum and maximum (or requested) time step is discussed further in Volume V, Section 3.1.3.1 and Section 4.2.

The control option is a packed five digit (ssdtt) word containing a major edit select option (ss), a debug output option (d), and the time step control (tt). The major edit select option (ss) allows sections of major edits for the hydrodynamic volumes and junctions, heat structures, and statistics to be skipped. The debug output option (d) forces any combination of plot, minor edits, or major edit output to be written at each successful advancement rather than at just the completion of advancement over a requested time step. The time step control option (tt) allows the user to change the time step control logic. All options can be changed with each time step control card.

Specifically, digit tt allows the user to select several time step control options. The number input for tt is transformed into a six bit binary number (i.e., a number in the base 2). Each bit in this binary number controls an option affecting time step control. For example, inputting a value of 15 generates the six bit binary number 001111. This sets the first four bits to one and the fifth and sixth bits to zero (i.e., the rightmost bit is the first bit, the next rightmost bit is the second bit, and so on). The effect of setting the bits to one or zero is described below.

Entering zero (no bits set) attempts to advance both the hydrodynamic and heat conduction advancements at the requested time step. However, the hydrodynamic time step will be reduced if necessary such that the Courant limit is satisfied. If out-of-range thermodynamic property conditions are encountered, the hydrodynamic advancement will be retried with reduced time steps. The problem will be terminated if the time step must be reduced beyond the minimum time step. Each time step reduction halves the previously attempted time step. At the beginning of an advancement for a requested time step, a step counter is set to one. Whenever a reduction occurs, the step counter is doubled. When a successful advancement occurs, the step counter is reduced by one. When the step counter is decremented to 0, the problem has been advanced over one requested time step. Doubling of the time step is allowed only when the step counter is even, and the step counter is halved when the time step is doubled. With no bits set, the time step is doubled whenever possible. At the completion of advancements over a requested time step, the next requested advancement is obtained and may be different from the previous requested time step if data from the next time step control card are used. If necessary, the new requested time step is reduced by halving until the new actual time step is < 1.5 times the last successful time step.

Setting bit one to a value of one includes the features described for entering zero and, in addition, uses the halving and doubling procedures to maintain an estimate of hydrodynamic truncation error within program-defined limits. The estimate is based on the mass error computed by comparing densities derived from the mass conservation equations and the equations of state. If an acceptable error is not reached and the next reduction would lead to a time step below the minimum time step, the advancement is accepted. The first 100 such occurrences are noted in the output.

If the second bit is set to one, the heat structure time step will be the same as the hydrodynamic time step. The time step control for the hydrodynamics is determined by the status of the first bit as described above, and both the heat conduction and hydrodynamic advancements are repeated when a time step reduction occurs.

If the third bit is set to one, the heat conduction-transfer and the hydrodynamics are advanced implicitly. When the third bit is set indicating implicit coupling of heat conduction-transfer, the second bit indicating that the two advancements use the same time step must also be set. (Input checking does not now enforce this, but a future code change will include this checking.) If the third bit is not set, the heat conduction-transfer and hydrodynamic advancements are serially coupled. That is, the heat conduction transfer is advanced first using old hydrodynamic information, and the hydrodynamics is then advanced using new heat transfer information. The time step control for hydrodynamics is determined by the status of the first bit, as described above.

If the fourth bit is set to one, the hydrodynamics will use the nearly-implicit hydrodynamic numerical scheme. The time step can be as large as 20 times the Courant limit for the TRANSNT option and 40 times the Courant limit for the STDY-ST option. The time step control for hydrodynamics is determined by the status of the first bit, as described above.

If the fifth bit is set to one, termination of the steady-state advancement will not be controlled by the algorithm that measures the closeness to steady state (see section 8.3 in Volume I); if this bit is set to zero, steady state will be terminated by the algorithm when it detects steady state has been reached.

If the sixth bit is set to one, an on-line algorithm is used to choose between the semi- and nearly-implicit solution schemes of time integration is used to advance the hydrodynamics. The semi-implicit scheme will be used when the time step is below the Courant limit, and the nearly-implicit scheme will be used when a large time step can be taken.

The table below illustrates how the value input for tt results in option selection.

Table 8.2-1 Time step selection options.

tt input	Binary equivalent	Description	Comment
0	000000	No options selected	Not recommended
1	000001	Same as tt = 0 except time step size can be controlled by mass error	Care must be taken to prevent large time steps from introducing truncation error
3	000011	Same as tt = 1 plus heat transfer and hydrodynamics advanced with same time step	Recommended for all transient analysis

Table 8.2-1 Time step selection options.

tt input	Binary equivalent	Description	Comment
7	000111	Same as tt = 3 plus heat transfer and hydrodynamics are implicitly coupled	Suitable for all transient analysis
15	001111	Same as tt = 7 except nearly-implicit scheme is used rather that semi-implicit scheme	Suitable for steady-state analysis and slow transients
31	011111	Same as tt = 15 except bypasses check on approach to steady state	Used only in conjunction with a STDY_ST run

Note that combinations of the effects of setting of the individual bits are achieved by setting bits in combination. For example, entering three (setting bits two and one equal to 1) results in the combined effects described above for bits two and one. Older versions of RELAP5-3D© would convert 2 to 3 to maintain compatibility; this is no longer done.

Entering zero is not recommended except for special program testing situations. If bit one is set to one and zero is set for bits 2 and 3, care must be taken in selection of the requested time step. Individually, the hydrodynamic and heat conduction advancements are stable; the hydrodynamic time step is controlled to ensure stability, the heat conduction solution with constant thermal properties is stable for all time steps, and the change of thermal properties with temperature has not been a problem. The serial coupling of the hydrodynamic volumes and heat structures through heat structure boundary conditions can be unstable, and excessive truncation error with large time steps can occur. This has been observed in test problems. Entering three usually eliminates the problem, and nearly all verification and validation (assessment) of the code has been done using option three. Using option seven, which includes the implicit coupling of heat conduction and hydrodynamics should lead to an improved advancement. Users are encouraged to use option seven, but with the caution that this is a recent addition to RELAP5-3D© and is still under validation (assessment). When using the implicit coupling, the heat conduction time step must be the same as the hydrodynamic time step. This requirement is currently not enforced by the coding. Use of option 15 is suitable for steady-state runs and slow transients. Several improvements have been made to the nearly-implicit advancement, but use of that option is still under validation (assessment).

The minor edit, major edit, and restart frequencies are based on the requested time-step size. A frequency n means that the action is taken when a period of time equal to n requested time steps has elapsed. The edits and the restart record are written at time zero and at the specified frequencies up to the time limit on the time step control card. The maximum time step is reduced if needed, and the edits and restart record are forced at the end time value. Actions at the possibly new specified frequencies begin with the first advancement with a new time step control card. A restart forces a major and minor edit to be written, and a major edit forces a minor edit to be written. Plot information is written to the internal plot and restart-plot files whenever a minor edit is written. Note that minor edits are produced only if minor edit

requests are entered; an internal plot file is written only if internal plot requests are entered; and plot and restart data are written on the restart-plot file only if the file is requested.

An option (d) used for program testing can force a plot print, minor edit, major edit, or combinations of these to be written at each advancement. Care should be used, since considerable output can be generated.

Major edits forced by the program testing option or the last major edit of the problem terminated by approach to the job CPU limit may not coincide with the requested time step. When this occurs, a warning message is printed that states that not all quantities are advanced to the same time points.

8.3 Printed Output

A program version identification is printed at the beginning of printed output and the first page following the list of input data.

8.3.1 Input Editing

Printed output for a problem begins with a listing of the input; each line of input is preceded by a sequence number. The sequence number is not the same as the card number. Notification messages are listed when data card replacement or deletion occurs. Punctuation errors, such as an alphabetic character in numeric fields, multiple signs, periods, etc., are noted by an error message, and a ^ (caret) is printed under the card image indicating the column position of the error. The printing of this section of the output can be controlled by the user by use if input cards 4 or 5 (see Section 2.2 of Appendix A of this volume) to control the size of the printed output file.

Input processing consists of three phases. The first phase simply reads and stores all the input data for a problem such that the data can later be retrieved by card number. Error checking is limited to punctuation checking, and erroneous data flagged during this phase nearly always causes additional diagnostics in later phases. The second phase does the initial processing of data. Input data are moved and expanded into dynamic arrays sized for the problem being solved, and default options are applied. Processing and error checking is local to the data being processed. That is, when processing a single-junction component, no checking is performed regarding the existence of connected volumes. Similarly, hydrodynamic volumes connected to heat structure surfaces are not checked during processing of heat structure boundary data. At the end of this phase, all data cards should have been used. Unused cards are considered errors and are listed. Asterisks following the card number indicate that the card number was bad, that an error was noted in the card image listing, and that the number is the sequence number rather than the card number. The third phase completes input processing and performs requested initialization. Once the second phase has been completed, data specifying linkages between various blocks of data can now be processed and checked. Examples of error checking are junction connections made to nonexisting volumes, elevation loop closure, heat structure surfaces connected to nonexisting hydrodynamic volumes, specified thermal properties, and power data not entered. Solution of steady-state heat conduction for initial temperature distribution in heat structures is an example of initialization.

The flow map used for a particular volume is printed out during the input editing of the hydrodynamic volumes. **Table 8.3-1** shows the flow map and the corresponding number printed out under the label Flow Map.

Table 8.3-1 Flow map identifiers.

Flow map	Number (input edit)
Vertical	1
Horizontal	2
Annular	3
High Mixing	4
ECC Mixer	5

Depending on the type of data, input is edited in only one of the last two edits or in both of them. Error diagnostics can be issued during either phase, even if no editing for the erroneous data is done in a phase. When an error is detected, possible corrective actions are disregarding the data, which usually leads to other diagnostics, inserting benign data, or marking data as being entered but useless for further processing. These actions are taken so that input processing continues despite severe errors (other than on problem type and options). Regardless of errors, all data are given preliminary checking. Severe errors can limit cross-checking. Correcting input errors diagnosed in a submittal may lead to other diagnostics in a subsequent submittal, as elimination of errors allow more detailed checking. Except for exceeding requested computer time, disk limits, and printed output limits, any abnormal termination is considered a programming error and even exceeding computer time limits is prevented during transient execution. The final message of input processing indicates successful input processing or that the problem is being terminated as the result of input errors.

8.3.2 Major Edits

Major edits are an editing of most of the key quantities being advanced in time. The amount of output depends on the input deck and output options chosen by the code user. Output includes a time-step summary, trip information, reactor kinetics information, one to four sections of hydrodynamic volume information, hydrodynamic volume time-step control information, one or two sections of hydrodynamic junction information, metal-water reaction information, heat structure/heat transfer information, heat structure temperatures, reflood information, reflood surface temperatures, cladding rupture information, surface radiation information, control variable information, and generator information. Major edits are quite lengthy, and care should be used in selecting print frequencies. Some sections of major edits can be bypassed through input data on time-step control cards. In addition to controlling the content to the major edits through the time step control options as discussed in the previous section, the content of the major edits may be controlled by the user through the use of input cards 2 through 5 (see Section 2.2 of Appendix

A of this volume) to reduce the size of the printed output. An example of a major edit is shown in **Figure 8.3-1**.

Each section of information is discussed below in the order that each appears in a major edit. In particular, what the abbreviated labels stand for as well as how they relate to variables used in Volume 1 of this manual are indicated.

8.3.2.1 Time Step Summary. As shown in **Figure 8.3-1**, the first section of a major edit prints the problem time and statistics concerning time step control. ATTEMPTED ADV. is the total number of successful and repeated advancements. REPEATED ADV. is the number of advancements that were not accepted and were retried with a halved time step size (i.e., due to mass error, thermodynamic property error, etc.) or were repeated with the same time step size (i.e., due to velocity flip-flop, noncondensable appearance, water packing). SUCCESSFUL ADV. is the number of accepted advancements. REQUESTED ADV. is the number of advancements with the specified requested maximum time step. These are presented in two columns. The TOT. column is over the entire problem; the EDIT column contains the number since the previous major edit. MIN. DT, MAX. DT, and AVG. DT are the minimum, maximum, and average time step used since the last major edit. REQ. DT is the requested maximum time step used since the last major edit. This quantity may not be the requested time step entered on the card if the major edit is for the final time value on the card. LAST. DT is the time step used in the last advancement. CRNT. DT is the time step limit based on the Courant stability criterion for the last advancement. ERR. EST is the estimate of the truncation mass error fraction at the last advancement. This is the maximum of the two types of computed mass error (ε_m and ε_{rms}) discussed in Volume I, Section 8. Entering 1, 3, 5, 7, 9, 11, 13, 15, 17, 19, 21, 23, 25, 27, 29, 31, 33, 35, 37, 39, 41, 43, 45, 47, 49, 51, 53, 55, 57, 59, 61, or 63 for the time step control option will reduce or double the time step to keep this quantity between the limits of 8.0×10^{-4} and 8.0×10^{-3} if the mass error criterion is controlling the time step. In **Figure 8.3-1**, the problem is running at the requested (maximum) time-step, and the ERR. EST is below the lower limit. CPU is the CPU time for the entire problem up to the time of the major edit. TOT. MS is the total mass currently contained in the hydrodynamic systems, and MS. RED is an estimate of the cumulative error in the total mass owing to truncation error. Either M.RATO, which is the ratio of the cumulative mass error to the total mass at the start of the transient, or M. RATN, which is the ratio of the cumulative mass error to the current total mass is listed. The output lists the ratio with the largest denominator, thus the smaller of the two ratios. TIME is the simulated time for the entire problem up to the time of the major edits.

8.3.2.2 Trip Information. At major edits, each defined trip number and the current TIMEOF quantity are printed. The TIMEOF quantity is -1.0 when the trip is false, and when ≥ 0, indicates that the trip is true and is the time the trip last switched to true. **Figure 8.3-1** includes an example of a trip edit.

8.3.2.3 Reactor Kinetics Information. At major edits, the total reactor power (labeled TOTAL POWER), immediate (prompt and delayed neutron) fission power (labeled FISSION POWER), decay (fission product and actinide) power (labeled GAMMA POWER), reactivity (labeled REACTIVITY), and reciprocal period (labeled REC. PERIOD) are printed. Either the total power, immediate (prompt and delayed neutron) fission power, or decay (fission product and actinide) power can be specified as the time-varying part of the heat source in heat structures. **Figure 8.3-1** illustrates a reactor kinetics edit; however, it is not intended to be physically realistic.

```
  RELAP5/3.1#2#2        Reactor Loss Of Coolant Analysis Program
edward's pipe problem base case with extras                                    27-Dec-94    17:20:07
MAJOR EDIT !!!time=   0.100000     sec
 attempted adv: tot.=    109   edit=     20   min.dt=   1.000000E-03 sec   last dt=   1.000000E-03 sec   ms.red=  -1.787187E-03 kg
  repeated adv: tot.=      3   edit=      0   max.dt=   1.000000E-03 sec   crnt.dt=   7.025645E-03 sec   tot.ms=    11.1019     kg
successful adv: tot.=    106   edit=     20   avg.dt=   1.000000E-03 sec   err.est=   3.891584E-06       m.rato=  -1.149045E-04
 requested adv: tot.=    100   edit=     20   req.dt=   1.000000E-03 sec       cpu=    2.00080     sec     time=   0.100000     sec
Trip number, trip time (sec)
    501  -1.000000         502  -1.000000         505  6.0000000E-03
    601  -1.000000         602  -1.000000         603  0.0000000E+00
     Total power      fission power    gamma power      reactivity       rec. period
       (Watts)          (Watts)          (Watts)          (dollars)        (sec-1)
     6.46074E+10      6.45997E+10      7.75141E+06       1.5008           101.02
System  1    *none*  mass=   11.102     kg    mass error= -1.78719E-03 kg    err.est.=  3.89158E-06
   Vol.no.  pressure     voidf        voidg        voidgo      tempf       tempg       sat. temp.      uf           ug       vol-flag
              (Pa)                                                (k)         (K)         (K)          (J/kg)       (J/kg)      tlpvbfe
edward's pipe
  3-010000  2.58633E+06  0.87323      0.12677      0.12564     500.483     498.867     498.905      9.74638E+05  2.60139E+06 000000
  3-020000  2.58627E+06  0.88929      0.11071      0.10962     500.538     498.872     498.904      9.74892E+05  2.60140E+06 000000
  3-030000  2.58627E+06  0.86369      0.13631      0.13531     500.353     498.870     498.904      9.74035E+05  2.60140E+06 000000
  3-040000  2.58658E+06  0.87074      0.12926      0.12806     500.378     498.894     498.910      9.74150E+05  2.60144E+06 000000
  3-050000  2.58749E+06  0.89085      0.10915      0.10795     500.524     498.939     498.929      9.74826E+05  2.60151E+06 000000
  3-060000  2.58914E+06  0.90717      9.28307E-02  9.18278E-02 500.647     498.997     498.963      9.75398E+05  2.60158E+06 000000
  3-070000  2.59131E+06  0.91570      8.43030E-02  8.34841E-02 500.717     499.055     499.008      9.75723E+05  2.60161E+06 000000
  3-080000  2.59326E+06  0.91516      8.48351E-02  8.41059E-02 500.714     499.091     499.048      9.75708E+05  2.60161E+06 000000
  3-090000  2.59310E+06  0.91185      8.81460E-02  8.72750E-02 500.694     499.033     499.045      9.75614E+05  2.60148E+06 000000
  3-100000  2.58992E+06  0.89705      0.10295      0.10177     500.584     498.885     498.979      9.75102E+05  2.60126E+06 000000
  3-110000  2.58203E+06  0.88244      0.11756      0.11629     500.472     498.674     498.816      9.74589E+05  2.60111E+06 000000
  3-120000  2.56699E+06  0.73794      0.26206      0.23511     498.981     498.471     498.503      9.67675E+05  2.60133E+06 000000
  3-130000  2.46936E+06  0.56045      0.43955      0.43627     496.660     496.433     496.438      9.57014E+05  2.60099E+06 000000
  3-140000  2.40135E+06  0.49097      0.50903      0.50478     495.333     494.953     494.962      9.50972E+05  2.60059E+06 000000
  3-150000  2.36217E+06  0.46318      0.53682      0.53349     494.703     494.063     494.097      9.48107E+05  2.60031E+06 000000
  3-160000  2.33684E+06  0.44284      0.55716      0.55561     494.183     493.443     493.532      9.45740E+05  2.60003E+06 000000
  3-170000  2.31703E+06  0.42559      0.57441      0.57370     493.704     492.949     493.087      9.43550E+05  2.59979E+06 000000
  3-180000  2.30117E+06  0.41320      0.58680      0.58622     493.310     492.561     492.728      9.41752E+05  2.59962E+06 000000
  3-190000  2.28849E+06  0.39956      0.60044      0.59981     492.991     492.248     492.440      9.40295E+05  2.59949E+06 000000
  3-200000  2.28174E+06  0.43684      0.56316      0.56283     492.797     492.163     492.286      9.39410E+05  2.59962E+06 000000
rhtbdy   tmdpvol
  5-010000  1.00000E+05  0.00000E+00   1.0000       1.0000      372.782     372.782     372.782      4.17407E+05  2.50606E+06 011000
   Vol.no.    rhof         rhog        rho-mix     rho-boron   vel-liquid  vel-vapor    sounde       quality   quality   quality
             (kg/m3)      (kg/m3)      (kg/m3)     (kg/m3)      (m/sec)     (m/sec)     (m/sec)      mix-cup   static   non-cond.
  3-010000   830.80       12.945       727.12      0.00000E+00  0.25810     0.12238      26.817      5.163E-03 2.257E-03 0.000E+00
  3-020000   830.73       12.945       740.19      0.00000E+00  0.37923     0.24891      26.638      5.538E-03 1.936E-03 0.000E+00
  3-030000   830.98       12.945       719.47      0.00000E+00  0.61811     0.47123      26.627      5.361E-03 2.452E-03 0.000E+00
  3-040000   830.94       12.945       725.21      0.00000E+00  0.87499     0.68065      26.526      5.582E-03 2.307E-03 0.000E+00
  3-050000   830.75       12.949       741.49      0.00000E+00   1.1313     0.98186      26.488      5.874E-03 1.906E-03 0.000E+00
  3-060000   830.58       12.956       754.68      0.00000E+00   1.3522      1.2668      26.433      5.912E-03 1.594E-03 0.000E+00
```

Figure 8.3-1 Example of major edit.

3-070000	830.49	12.966	761.57	0.00000E+00	1.5330	1.4546	26.351	5.776E-03	1.435E-03	0.000E+00
3-080000	830.49	12.976	761.14	0.00000E+00	1.6950	1.5898	26.239	5.583E-03	1.446E-03	0.000E+00
3-090000	830.52	12.978	758.46	0.00000E+00	1.8733	1.7781	26.252	5.590E-03	1.508E-03	0.000E+00
3-100000	830.67	12.966	746.48	0.00000E+00	2.1180	2.1369	26.401	5.734E-03	1.788E-03	0.000E+00
3-110000	830.82	12.929	734.67	0.00000E+00	2.4275	2.6664	26.828	6.314E-03	2.069E-03	0.000E+00
3-120000	832.82	12.849	617.94	0.00000E+00	5.9491	9.0256	26.775	6.677E-03	5.449E-03	0.000E+00
3-130000	835.85	12.363	473.88	0.00000E+00	11.926	13.998	32.083	1.000E-02	1.147E-02	0.000E+00
3-140000	837.56	12.027	417.33	0.00000E+00	16.202	17.195	35.657	1.467E-02	1.467E-02	0.000E+00
3-150000	838.36	11.834	394.66	0.00000E+00	19.321	20.204	37.688	1.754E-02	1.610E-02	0.000E+00
3-160000	839.02	11.711	378.08	0.00000E+00	21.396	22.623	38.938	1.919E-02	1.726E-02	0.000E+00
3-170000	839.64	11.615	364.01	0.00000E+00	22.839	24.531	39.881	2.057E-02	1.833E-02	0.000E+00
3-180000	840.14	11.538	353.92	0.00000E+00	23.920	26.099	40.554	2.180E-02	1.913E-02	0.000E+00
3-190000	840.55	11.476	342.74	0.00000E+00	24.935	27.307	41.438	2.276E-02	2.010E-02	0.000E+00
3-200000	840.79	11.440	373.74	0.00000E+00	24.500	29.176	38.238	2.346E-02	1.724E-02	0.000E+00
Vol.no.	rhof (kg/m3)	rhog (kg/m3)	rho-mix (kg/m3)	rho-boron (kg/m3)	vel-liquid (m/sec)	vel-vapor (m/sec)	sounde (m/sec)	quality mix-cup	quality static	quality non-cond.
5-010000	958.39	0.59041	0.59041	0.00000E+00	23.455	30.878	473.21	2.257E-02	1.00	0.000E+00

Vol.no.	Q.wall.tot (Watts)	Qwg.wall.gas (Watts)	Vapor.gen (kg/sec-m3)	gamma.boil (kg/sec-m3)	Hif.liq.int (Watts/m3-k)	Hig.vap.int (Watts/m3-k)	Mass-flux (kg/sec-m2)	Reynolds liquid	Reynolds vapor	Flow regi
3-010000	834.80	0.00000E+00	15.228	0.48854	1.70679E+07	1.44099E+06	93.722	60278.	462.43	hst
3-020000	606.63	0.00000E+00	15.638	0.35504	1.70830E+07	1.58281E+06	278.02	1.78759E+05	1761.8	hst
3-030000	595.04	0.00000E+00	14.506	0.34830	1.78500E+07	1.52113E+06	450.88	2.89812E+05	3469.4	hst
3-040000	621.34	0.00000E+00	17.690	0.36378	2.15460E+07	1.82284E+06	631.71	4.05983E+05	5387.5	hst
3-050000	633.11	0.00000E+00	16.963	0.37075	1.89565E+07	2.04745E+06	829.37	5.33072E+05	6977.4	hst
3-060000	635.68	0.00000E+00	13.633	0.37233	1.43152E+07	1.84202E+06	1011.4	6.50298E+05	7628.4	hst
3-070000	644.67	0.00000E+00	10.825	0.37763	1.10888E+07	1.81699E+06	1162.2	7.47516E+05	7686.1	hst
3-080000	671.42	0.00000E+00	10.210	0.39332	1.06929E+07	1.71363E+06	1290.4	8.30236E+05	8027.0	hst
3-090000	704.52	0.00000E+00	11.828	0.41277	1.26306E+07	2.08088E+06	1423.2	9.15610E+05	9187.5	hst
3-100000	791.54	0.00000E+00	17.081	0.46389	1.90636E+07	2.96775E+06	1593.7	1.02475E+06	12194.	hst
3-110000	901.27	0.00000E+00	17.188	0.52830	1.89090E+07	6.49353E+06	1798.1	1.15480E+06	17530.	hst
3-120000	2867.0	0.00000E+00	484.66	1.6981	1.81826E+09	9.10588E+07	4030.9	2.57650E+06	1.01646E+05	bby
3-130000	7513.2	0.00000E+00	265.72	4.4462	2.16831E+09	3.92132E+08	6519.7	4.11257E+06	2.85180E+05	bby
3-140000	9875.7	0.00000E+00	194.69	5.8180	9.46904E+08	3.34815E+08	7225.3	4.50771E+06	4.61445E+05	slg
3-150000	10613.	0.00000E+00	136.36	6.2235	4.02689E+08	9.68110E+07	7850.0	4.86687E+06	5.86370E+05	slg
3-160000	10994.	0.00000E+00	98.479	6.4244	2.65440E+08	2.74888E+07	8274.8	5.10903E+06	6.78599E+05	slg
3-170000	11173.	0.00000E+00	77.516	6.5122	2.16145E+08	1.22905E+07	8485.4	5.22052E+06	7.54779E+05	slg
3-180000	11144.	0.00000E+00	64.022	6.4808	1.85848E+08	7.84995E+06	8601.2	5.27597E+06	8.18613E+05	slg
3-190000	10976.	0.00000E+00	54.341	6.3708	1.63814E+08	5.89453E+06	8701.7	5.32495E+06	8.70984E+05	slg
3-200000	10621.	0.00000E+00	40.610	6.1564	1.27093E+08	7.48087E+06	8783.6	5.36694E+06	9.11898E+05	slg
5-010000	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	1.00000E+12	0.00000E+00	4406.9	1.17198E+06	6.30557E+05	mpr

vol.no.	voidla	voidlb	vollev (m)	vlev (m/sec)

Vol.no.	lrgst.mass err. edit	lrgst.mass err. total	reduce-quality edit	reduce-quality total	reduce-extrap. edit	reduce-extrap. total	reduce-mass edit	reduce-mass total	reduce-propty. edit	reduce-propty. total	min.courant edit	min.courant total	reduce-courant edit	reduce-courant total
3-010000	0	6	0	0	0	0	0	0	0	0	0	0	0	0
3-020000	0	0	0	0	0	0	0	0	0	0	0	0	0	0

Figure 8.3-1 Example of major edit. (Continued)

3-030000	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3-040000	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3-050000	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3-060000	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3-070000	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3-080000	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3-090000	1	1	0	0	0	0	0	0	0	0	0	0	0	0
3-100000	2	4	0	0	0	0	0	0	0	0	0	0	0	0
3-110000	5	6	0	0	0	0	0	0	0	0	0	0	0	0
3-120000	9	9	0	0	0	0	0	0	0	0	0	0	0	0
3-130000	3	13	0	0	0	0	0	0	0	0	0	0	0	0
3-140000	0	11	0	0	0	0	0	0	0	0	0	0	0	0
3-150000	0	9	0	0	0	0	0	0	0	0	0	0	0	0
3-160000	0	7	0	0	0	0	0	0	0	0	0	0	0	0
3-170000	0	8	0	0	0	0	0	0	0	0	0	0	0	0
3-180000	0	11	0	0	0	0	0	0	0	0	0	0	0	0
3-190000	0	8	0	0	0	0	0	0	0	0	0	0	0	0
3-200000	0	13	0	0	0	2	0	0	0	2	20	105	0	0
5-010000	0	0	0	0	0	0	0	0	0	0	0	0	0	0

System 1 *none*

Jun.no.	from vol.	to vol.	liq.j.vel. (m/sec)	vap.j.vel. (m/sec)	mass flow (kg/sec)	jun.area (m2)	throat ratio	jun-flag jefvcahs	flow regi	no. advs. last	edit	choked total
edward's pipe												
3-010000	3-010002	3-020001	0.25810	0.12238	0.85590	4.56037E-03	1.0000	0000000	hst	0	0	0
3-020000	3-020002	3-030001	0.49819	0.39379	1.6830	4.56037E-03	1.0000	0000000	hst	0	0	0
3-030000	3-030002	3-040001	0.74154	0.53413	2.4341	4.56037E-03	1.0000	0000000	hst	0	0	0
3-040000	3-040002	3-050001	1.0074	0.83515	3.3347	4.56037E-03	1.0000	0000000	hst	0	0	0
3-050000	3-050002	3-060001	1.2524	1.1555	4.2397	4.56037E-03	1.0000	0000000	hst	0	0	0
3-060000	3-060002	3-070001	1.4502	1.3975	4.9960	4.56037E-03	1.0000	0000000	hst	0	0	0
3-070000	3-070002	3-080001	1.6152	1.5175	5.6139	4.56037E-03	1.0000	0000000	hst	0	0	0
3-080000	3-080002	3-090001	1.7750	1.6616	6.1653	4.56037E-03	1.0000	0000000	hst	0	0	0
3-090000	3-090002	3-100001	1.9719	1.8903	6.8265	4.56037E-03	1.0000	0000000	hst	0	0	0
3-100000	3-100002	3-110001	2.2664	2.3482	7.7258	4.56037E-03	1.0000	0000000	hst	0	0	0
3-110000	3-110002	3-120001	2.5913	2.9460	8.6965	4.56037E-03	1.0000	0000000	hst	0	0	0
3-120000	3-120002	3-130001	9.9548	11.770	29.068	4.56037E-03	1.0000	0000000	bby	0	0	0
3-130000	3-130002	3-140001	14.513	15.378	31.561	4.56037E-03	1.0000	0000000	bby	0	0	0
3-140000	3-140002	3-150001	18.126	18.809	34.802	4.56037E-03	1.0000	0000000	slg	0	0	0
3-150000	3-150002	3-160001	20.588	21.549	37.336	4.56037E-03	1.0000	0000000	slg	0	0	0
3-160000	3-160002	3-170001	22.242	23.669	38.519	4.56037E-03	1.0000	0000000	slg	0	0	0
3-170000	3-170002	3-180001	23.460	25.374	39.064	4.56037E-03	1.0000	0000000	slg	0	0	0
3-180000	3-180002	3-190001	24.394	26.814	39.499	4.56037E-03	1.0000	0000000	slg	0	0	0
3-190000	3-190002	3-200001	25.494	27.791	39.979	4.56037E-03	1.0000	0000000	slg	0	0	0
rhtbdy sngljun												
4-000000	3-200002	5-010001	26.960	35.493	40.222	3.96752E-03	1.0000	0000000	slg	1	20	98

Figure 8.3-1 Example of major edit. (Continued)

Jun.no.	voidfj	voidgj	fij (N-s2/m5)	fwalfj	fwalgj	fjunf	fjunr	formfj	formgj	no. last	advs. edit	ccfl total
3-010000	0.87323	0.12677	12.181	6.057E-02	0.498	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-020000	0.88929	0.11071	44.597	4.627E-02	0.254	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-030000	0.86369	0.13631	18.655	4.403E-02	0.254	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-040000	0.87074	0.12926	31.701	4.077E-02	0.228	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-050000	0.89085	0.10915	59.239	3.862E-02	0.233	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-060000	0.90717	9.28307E-02	115.43	3.725E-02	0.243	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-070000	0.91570	8.43030E-02	71.569	3.624E-02	0.249	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-080000	0.91516	8.48351E-02	6.0437	3.544E-02	0.236	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-090000	0.91185	8.81460E-02	3.5683	3.483E-02	0.212	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-100000	0.89705	0.10295	4.7712	3.378E-02	0.177	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-110000	0.88244	0.11756	18.112	4.765E-02	0.114	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-120000	0.73794	0.26206	51326.	1.966E-02	2.760E-02	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-130000	0.56045	0.43955	2.00582E+05	1.720E-02	2.013E-02	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-140000	0.49097	0.50903	2.00261E+05	1.658E-02	9.475E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-150000	0.46318	0.53682	67008.	1.631E-02	2.725E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-160000	0.44284	0.55716	23980.	1.602E-02	1.223E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-170000	0.42559	0.57441	9935.6	1.581E-02	6.896E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-180000	0.41320	0.58680	4688.7	1.569E-02	4.393E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
3-190000	0.39956	0.60044	2620.7	1.555E-02	8.236E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0
4-000000	0.43684	0.56316	2944.0	6.521E-03	4.580E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0	0	0

RELAP5/3.1#2#2 Reactor Loss Of Coolant Analysis Program

edward's pipe problem base case with extras 27-Dec-94 17:20:07

HEAT STRUCTURE OUTPUT +++time= 0.100000 sec

str.no.	side	bdry.vol. number	surface temp. (K)	heat-trf. convection (Watt)	heat-flux convection (Watt/m2)	critical heat-flux (Watt/m2)	CHF mul	ht mode	heat-trf. coef.conv (Watt/m2-k)	int.-heat source (Watt)	conv+rad -source (Watt)	vol.ave. temp. (K)
30-001	left	3-010000	501.543	834.80	17027.	3.72688E+06	0.72	4	7212.4	0.00000E+00	834.80	502.05
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-002	left	3-020000	501.803	606.63	12373.	5.06775E+06	0.73	4	5916.0	0.00000E+00	606.63	502.11
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-003	left	3-030000	501.849	595.04	12137.	5.20362E+06	0.71	4	6233.7	0.00000E+00	595.04	502.13
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-004	left	3-040000	501.853	621.34	12673.	5.36164E+06	0.72	4	7114.5	0.00000E+00	621.34	502.13
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-005	left	3-050000	501.857	633.11	12913.	5.55411E+06	0.74	4	8225.1	0.00000E+00	633.11	502.13
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-006	left	3-060000	501.856	635.68	12966.	5.70648E+06	0.75	4	9259.3	0.00000E+00	635.68	502.13
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-007	left	3-070000	501.847	644.67	13149.	5.66860E+06	0.76	4	10149.	0.00000E+00	644.67	502.13
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			
30-008	left	3-080000	501.829	671.42	13695.	5.59057E+06	0.76	4	10879.	0.00000E+00	671.42	502.13
	right	0-000000	502.183	0.00000E+00	0.00000E+00	0.00000E+00	0.00	0	0.00000E+00			

Figure 8.3-1 Example of major edit. (Continued)

```
 30-009  left    3-090000  501.810   704.52       14370.      5.49912E+06 0.75   4     11586.      0.00000E+00   704.52     502.13
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-010  left    3-100000  501.775   791.54       16145.      5.45335E+06 0.74   4     12478.      0.00000E+00   791.54     502.12
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-011  left    3-110000  501.737   901.27       18383.      5.59273E+06 0.73   4     13538.      0.00000E+00   901.27     502.12
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-012  left    3-120000  501.611   2867.0       58477.      5.44584E+06 0.83   4     24834.      0.00000E+00   2867.0     502.11
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-013  left    3-130000  500.913   7513.2      1.53245E+05  4.27520E+06 0.84   4     36433.      0.00000E+00   7513.2     502.06
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-014  left    3-140000  500.003   9875.7      2.01433E+05  4.15672E+06 0.85   4     43779.      0.00000E+00   9875.7     501.98
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-015  left    3-150000  499.165   10613.      2.16480E+05  4.10480E+06 0.85   4     49141.      0.00000E+00   10613.     501.88
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-016  left    3-160000  498.469   10994.      2.24237E+05  4.10462E+06 0.86   4     52609.      0.00000E+00   10994.     501.78
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-017  left    3-170000  497.862   11173.      2.27894E+05  4.10481E+06 0.86   4     54902.      0.00000E+00   11173.     501.67
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-018  left    3-180000  497.329   11144.      2.27297E+05  4.10517E+06 0.86   4     56630.      0.00000E+00   11144.     501.55
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-019  left    3-190000  496.851   10976.      2.23867E+05  4.10547E+06 0.86   4     58069.      0.00000E+00   10976.     501.43
         right   0-000000  502.183  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
 30-020  left    3-200000  496.460   10621.      2.16638E+05  4.10587E+06 0.87   4     59203.      0.00000E+00   10621.     501.31
         right   0-000000  502.182  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00
200-001  left    0-000000    9.220  0.00000E+00  0.00000E+00  0.00000E+00 0.00   0    0.00000E+00  0.00000E+00   13510.       5.86
         right   0-000000    0.000   13510.      1.10741E+05  0.00000E+00 0.00   0    0.00000E+00
                                                                  Sum of int.-heat-source=   0.00000E+00 (watt)
 At time=    0.100000     sec       str.no.   mesh point temperatures     (K)
 30-001    501.54      501.74      501.90      502.02      502.09      502.14      502.17      502.18      502.18      502.18
           502.18
 30-002    501.80      501.94      502.04      502.10      502.14      502.16      502.18      502.18      502.18      502.18
           502.18
 30-003    501.85      501.98      502.07      502.12      502.15      502.17      502.18      502.18      502.18      502.18
           502.18
 30-004    501.85      501.99      502.07      502.13      502.16      502.17      502.18      502.18      502.18      502.18
           502.18
 30-005    501.86      501.99      502.08      502.13      502.16      502.17      502.18      502.18      502.18      502.18
           502.18
 30-006    501.86      501.99      502.08      502.13      502.16      502.17      502.18      502.18      502.18      502.18
           502.18
 30-007    501.85      501.99      502.08      502.13      502.16      502.17      502.18      502.18      502.18      502.18
           502.18
 30-008    501.83      501.98      502.08      502.13      502.16      502.17      502.18      502.18      502.18      502.18
           502.18
 30-009    501.81      501.97      502.07      502.13      502.16      502.17      502.18      502.18      502.18      502.18
           502.18
 30-010    501.78      501.95      502.06      502.13      502.16      502.17      502.18      502.18      502.18      502.18
```

Figure 8.3-1 Example of major edit. (Continued)

```
                502.18
  30-011        501.74      501.93      502.05      502.12      502.16      502.17      502.18      502.18      502.18      502.18
                502.18
  30-012        501.61      501.91      502.05      502.12      502.16      502.17      502.18      502.18      502.18      502.18
                502.18
  30-013        500.91      501.79      502.03      502.12      502.16      502.17      502.18      502.18      502.18      502.18
                502.18
  30-014        500.00      501.47      501.95      502.10      502.15      502.17      502.18      502.18      502.18      502.18
                502.18
  30-015        499.17      501.04      501.79      502.05      502.14      502.17      502.18      502.18      502.18      502.18
                502.18
  30-016        498.47      500.61      501.59      501.98      502.12      502.16      502.18      502.18      502.18      502.18
                502.18
  30-017        497.86      500.16      501.35      501.88      502.08      502.15      502.17      502.18      502.18      502.18
                502.18
  30-018        497.33      499.72      501.08      501.74      502.02      502.13      502.17      502.18      502.18      502.18
                502.18
  30-019        496.85      499.29      500.79      501.58      501.95      502.10      502.16      502.17      502.18      502.18
                502.18
  30-020        496.46      498.88      500.48      501.39      501.85      502.05      502.14      502.17      502.18      502.18
                502.18
 200-001        9.2198      9.1063      8.7686      8.2149      7.4590      6.5194      5.4193      4.1857      2.8491      1.4423
               0.00000E+00

                    Control variable edit; at time=    0.100000      sec
   1     ctl1         sum          2.586302E+06          4     ctl4         mult         1.140872E+06
   5     ctl5         mult          228.657             10     ctl10        div          4.382614E-07
  11     ctl10        div           1.00296             12     ctl12        diffreni      1.00000
  13     ctl13        integral     5.000000E-03         14     ctl14        integral     0.100000
  15     ctl15        diffreni     1.000000E-01         99     ctl99        diffrend      9.75990
 201     ctl201       function      3.00000            202     ctl202       stdfnctn     0.199667
 203     ctl203       tripunit     0.000000E+00        204     ctl204       tripdlay     -2.00000
 205     ctl205       poweri       0.200000            206     ctl206       powerr       5.000000E-03
 207     ctl207       powerx       0.300000            300     ctl300       delay        0.000000E+00
 301     ctl301       prop-int      2.15000            302     ctl302       lag          0.367876
 303     ctl303       lead-lag     0.683938            304     ctl304       constant     0.387000
 401     con1         constant     0.000000E+00        402     con2         constant     0.100000
 403     pumpctl      pumpctl     -1.000000E-03        404     steamctl     steamctl      2.00167
 405     feedctl      feedctl     -2.500000E-04

---Restart no.     109 written, block no.     2---
```

Figure 8.3-1 Example of major edit. (Continued)

8.3.2.4 Detector Information. This is output from the nuclear detector model that is not shown in **Figure 8.3-1**. The printout lists each detector number and the corresponding response for each detector number.

8.3.2.5 Hydrodynamic Volume Information--First Section. Systems are labeled SYSTEM, followed to the right by the system number (1, 2, 3, etc.) and the name of the system (optional; *none* if no name is input on Cards 120 through 129). To the right of this are the labels MASS, MASS ERROR, and ERR. EST. for this system, followed immediately by the actual value and unit. These three quantities correspond to the TOT. MS, MS. ERR, and ERR. EST listed in the Time Step Summary, except that these are only for the particular system whereas the Time Step Summary quantities are the sum for all the systems. The overall system mass error (ε_{ms} in Volume I, Section 8) is included in ERR.EST for the Time Step Summary but not for the individual systems. As **Figure 8.3-1** illustrates, quantities are grouped by component within each system. Each component is first labeled with the component name (supplied by the user) and the component type. Underneath this are the values for each volume within the component.

The first items printed in this section are the abbreviated labels and units for the quantities to be printed out. The first label is VOL. NO., which is the component number (CCC) and the six-digit volume subfield number (XX0000 for 1-D components and XYYZZ0 for 3-D components) within the component. These numbers are separated by a hyphen (-). Next is PRESSURE, which is the pressure (P_L^{n+1}) used in the hydrodynamic equation of Volume 1 of this manual. Next are VOIDF, VOIDG, and VOIDGO, which are the new liquid volume fraction, the new vapor/gas volume fraction, and the previous time step vapor/gas volume fraction $(\alpha_{f,L}^{n+1}, \alpha_{g,L}^{n+1}$, and $\alpha_{g,L}^{n})$ used in the equations. The previous time step vapor/gas volume fraction is significant because it helped determine the wall and interfacial terms on the current edit. Next are TEMPF, TEMPG, and SAT. TEMP., which are the liquid temperature $(T_{f,L}^{n+1})$, the vapor/gas temperature $(T_{g,L}^{n+1})$, and the saturation temperature $[T_L^{s,n+1}(P_s)]$ based on the vapor partial pressure used in the equations. For single-phase, the temperature of the missing phase is set to the saturation temperature. After this are UF and UG, which are the liquid specific internal energy $(U_{f,L}^{n+1})$ and the vapor/gas specific internal energy $(U_{g,L}^{n+1})$ used in the equations. Finally, the label VOL. FLAG is listed, which is the volume control flag (tlpvbfe) input by the user for hydrodynamic volume components. Following the labels, the title supplied by the user and type of component are given, followed by the actual values of the quantities for each volume.

Additional information is printed in the first hydrodynamic volume section that is unique to certain components. In **Figure 8.3-2**, additional information for a pump, compressor, turbine, and accumulator are given. For a pump, five additional quantities are printed. In the normal operating mode, these are the pump rotational velocity (RPM), pump head (HEAD), sum of the pump hydraulic torque and the pump frictional torque (TORQUE), pump octant number (OCTANT), and torque generated from the pump motor (MTR. TORQUE). These terms are discussed in Volume 1. For a compressor, five additional quantities are printed. In the normal operating mode, these are the compressor rotational velocity (RPM), compressor head (HEAD), sum of the compressor hydraulic torque and the compressor frictional torque (TORQUE), compressor thermodynamic efficiency (EFF), and the torque generated from the compressor motor (MTR.

TORQUE). These items are discussed in Volume I. For an accumulator, four additional quantities are printed. These are the volume of liquid in the tank-standpipe-surge line (LIQ. VOLUME), the mass of liquid in the tank-standpipe-surge line (MASS), the liquid level of liquid contained in the tank-standpipe-surge line (LEVEL), and the mean tank wall metal temperature (WALL TEMP). These terms are discussed in Volume 1 and in Section 2 of this volume. For a turbine, four additional quantities are printed. In the normal operating mode, these are the power extracted from the turbine (POWER), the torque extracted from the turbine (TORQUE), the turbine rotational speed (SPEED), and the efficiency factor used to represent nonideal internal processes (EFFICIENCY). These terms are also discussed in Volume 1 and in Section 2 of this volume.

```
ipump    pump
rpm = 125.22      (rad/sec)     head = 0.59115E+06 (Pa)   torque = - 0.10090E+06      (N-m)
octant = 2                                                  mtr. torque =  0.10090E+06  (N-m)

mcprss  cprssr
rpm = 376.99      (rad/sec)    head = 1.11923E+07 (Pa)   torque = - 1.22571E+05      (N-m)
eff = 0.93063                                             mtr. torque = 0.0000         (N-m)

snglaccm   accum
liq. volume = 24.404    m3, mass = 24167     kg, level = 19.028     m,  wall temp = 322.18 K

stage3   turbine
power = 1.75174E+08 (Watt)  torque = 0.30571E+06  (N-m) speed = 573.00 (rad/sec)
                                                        efficiency = 0.62945
```

Figure 8.3-2 Example of additional output for pumps, compressors, accumulators, and turbines.

8.3.2.6 Hydrodynamic Volume Information--Second Section. This information appears in every major edit if noncondensable species were specified in the input. This is not shown in **Figure 8.3-1**. In this section, no system or component label information is printed. The volume number (labeled VOL. NO.) and five to nine other quantities are printed on each line. These are printed out in numerical order within each system. The quantities are PART. PRESS., the partial pressure of vapor $(P_{s,L}^{n+1})$; DIRECT HTC, the direct heat transfer coefficient $(H_{gf,L}^{n})$; SATT-TOT, the saturation temperature $[T_L^{s,n+1}(P)]$ based on the total pressure; NONCOND. VAPOR MASS, noncondensable mass $(M_{n,L}^{n+1})$; and the mass fraction of each of the noncondensable species $(X_{ni,L}^{n+1})$, labeled by the element name and NCOND. QUAL. The noncondensable qualities X_{ni} sum to 1.0 in each volume.

8.3.2.7 Hydrodynamic Volume Information--Third Section. This section of output is optional and can be skipped by setting bit three in the ss digits of Word 4 (W4) on the time step control cards (Cards 201 through 299). This section is printed in **Figure 8.3-1**. In this section, no system information and no component label information is printed. Furthermore, no additional component quantities are printed out. Instead, just the volume number (VOL. NO.) and ten other quantities are printed out on each line. These are printed out in numerical order within each system. The quantities are RHOF, liquid density $(\rho_{f,L}^{n+1})$; RHOG, vapor/gas density $(\rho_{g,L}^{n+1})$; RHO-MIX, void-averaged mixture density (ρ_L^{n+1}) ; RHO-BORON, spatial boron density $(\rho_{B,L}^{n+1})$ [this is the volume liquid fraction (α_f) times the liquid density (ρ_f) times the

boron concentration (C_b). Boron concentration is used for hydrodynamic input, and spatial boron density is used for major edits.]; VEL-LIQUID, liquid volume-average velocity $(v_{f,L}^{n+1})$; VEL-VAPOR, vapor/gas volume-average velocity $(v_{g,L}^{n+1})$; SOUNDE, homogeneous frozen isentropic sonic velocity for single-phase or homogeneous equilibrium isentropic sonic velocity for two-phase $(a_{HE,L}^{n+1})$; QUALITY MIX-CUP, mixing cup or equilibrium quality used in wall heat transfer $(X_{e,L}^{n+1})$; QUALITY STATIC, static quality (X_L^{n+1}) ; and QUALITY NON-COND., noncondensable quality $(X_{n,L}^{n+1})$.

8.3.2.8 Hydrodynamic Volume Information--Fourth Section. This section prints whenever the third section prints. This section is printed in **Figure 8.3-1**. Following the volume number is TOT.HT.INP., the total wall heat transfer rate to the liquid and vapor/gas $(Q_L^n \bullet V_L)$; VAP.HT.INP., wall heat transfer rate to the vapor/gas $(Q_{wg,L}^n \bullet V_L)$; VAPOR-GEN., total mass transfer rate per unit volume at the vapor/liquid interface in the bulk fluid for vapor generation/condensation and in the boundary layer near the wall for vapor generation/condensation $(\Gamma_{g,L}^{n+1})$; WALL FLASHING, the mass transfer rate per unit volume at the vapor/liquid interface in the boundary layer near the wall for vapor generation $(\Gamma_{w,L}^{n+1})$; LIQ.INT.HTC, liquid-to-saturation interfacial heat transfer coefficient times area per unit volume $(H_{if,L}^n)$; VAP.INT.HTC, vapor/gas-to-saturation interfacial heat transfer coefficient times area per unit volume $(H_{ig,L}^n)$; MASS-FLUX, volume average mass flux (G); REYNOLDS LIQUID, liquid Reynolds number; REYNOLDS VAPOR, vapor/gas Reynolds number; and finally, FLOW REGI, flow regime. See Section 2.1 of this volume for the meaning of the flow regime label.

8.3.2.9 Hydrodynamic Volume Time Step Control Information. This section is also optional and can be skipped by setting bit four in the ss digits of Word 4 (W4) on the time step control cards (Cards 201 through 299). This section is printed in **Figure 8.3-1**. As with the previous section, no system or component label information is printed, no additional component quantities are printed, and all quantities are printed in volume numerical order within each system. All quantities are presented in two columns. The EDIT column contains the number since the previous major edit; the TOTAL column is over the entire problem.

The numbers under LRGST. MASS ERR give the number of times a volume had the largest mass error. The numbers under MIN. COURANT give the number of times a volume had the smallest time step based on the Courant stability limit. One volume under each of the headings is incremented by one for each successful advancement. The columns under REDUCE indicate volumes that have caused time step reductions. The MASS and PROPTY columns are for reductions resulting from mass error and out-of-range thermodynamic fluid properties. The MASS column is for time step size reduction resulting from local mass error; it does not include reductions resulting from overall (global) mass error. The QUALITY column is for reductions resulting from problems with void fraction (α_g), noncondensable quality (X_n), and mixture density from the phasic continuity equations (ρ_m). Advancements that result in α_g and X_n being slightly less than 0.0 or slightly greater than 1.0 are allowed, and the variable is reset to

0.0 or 1.0. Advancements that result in values much less than 0.0 or much greater than 1.0 are considered an error, and the time step is repeated. The cutoff points are based on a functional relationship. This relation is tied to the mass error upper limit (8×10^{-3}). Advancements that result in ρ_m being ≤ 0 are also counted in the QUALITY column. The final cause of a QUALITY column reduction relates to the one-phase to two-phase (appearance) case discussed in Volume 1 of this manual. If too much of one phase appears (more than a typical thermal boundary layer thickness), an error is assumed to have occurred, the time step is halved and repeated, and the QUALITY column counter is incremented. The EXTRAP column is for reductions when extrapolation into a metastable thermodynamic state causes problems (see Section 8 of Volume I for a discussion of metastable thermodynamic conditions). These problems are vapor/gas density (ρ_g) ≤ 0.0, vapor/gas temperature (T_g) ≤ 274 K, liquid density (ρ_f) ≤ 0.0, liquid temperature (T_f) $\geq$ saturation temperature (T^s) + 50 K, and vapor/gas temperature (T_g) $\leq$ saturation temperature (T^s) - 50 K. The COURANT column is for reductions resulting from the material Courant limit check. When the semi-implicit numerical scheme is used, the time step is reduced to the material Courant limit. When the nearly-implicit numerical scheme is used, the time step is reduced to 20 times the material Courant limit for the TRANSNT option and to 40 times the material Courant limit for the STDY-ST option.

Columns under the first four REDUCE headings are incremented only after a successful advancement following one or more successive reductions. Quantities are incremented only for those volumes that caused the last reduction. More than one column and row quantity can be incremented in a time step. Because of this characteristic, quantities in the first four REDUCE headings do not necessarily equal the REPEATED ADV quantity in the Time Step Summary at the top of a major edit. Since the REDUCE-COURANT column is for a reduction that occurs before the advancement takes place, it does not cause the time step to be repeated and, thus, does not increase the REPEATED ADV quantity.

New items have been recently added to this section that are not shown in **Figure 8.3-1**. Columns under the REPEAT headings are incremented if an advancement is repeated for several different reasons. If noncondensable gas first appears in a volume during an advancement, the quantity under the REPEAT-AIR APP is incremented and the time step is repeated with the same time step size. If water packing is detected in a volume, the quantity under the heading REPEAT-PACKING is incremented and the time step is repeated with the same time step size. In either of these situations, the time advancement and the time step is repeated with the same time step size. In either of these situations, the time advancement algorithm is modified to accommodate the appearance of noncondensable gas or water packing. The modifications to the hydrodynamic advancement algorithm are described in Volume I in the sections entitled Special Techniques and Water Packing Mitigation Scheme. In addition, if in the time step in which noncondensable gas first appears in a volume, the pressure change in that cell is too large, the time step is repeated with a smaller time step size to reduce the pressure change in that cell. The same advancement may be repeated several times with smaller and smaller time steps until the pressure change criterion is satisfied. Like the quantities under the REDUCE headings, the quantity under the REPEAT - DEL PRES heading is incremented only after the pressure change criterion is satisfied.

8.3.2.10 Hydrodynamic Junction Information--First Section. This section of output is not optional and always appears in a major edit. This section is printed in **Figure 8.3-1**. As with the first section of the

hydrodynamic volume information, quantities are grouped by system. For each system, the label SYSTEM, the system number (1, 2, 3, etc.), and the system name (optional) are printed on the first line. The first printed quantity for each junction is the junction number. [Labeled JUN. NO., it denotes the component number (CCC) and the six-digit junction subfield number (XX0000 for 1-D components and XYYZZF for 3-D components) within the component.] These numbers are separated by a hyphen (-). The next two quantities are the volume numbers for the *from* and *to* volumes associated with the junction (labeled FROM VOL. and TO VOL.). For 3-D components, the face number F is added to the volume number. A minus sign will be printed in front of the *from* volume number if it is not the outlet end of the volume. Similarly, a minus sign will be printed in front of the *to* volume number if it is not the inlet end of the volume. Next are LIQ.J.VEL. and VAP.J.VEL, liquid junction velocity and vapor/gas junction velocity ($v_{f,j}^{n+1}$ and $v_{g,j}^{n+1}$) . These velocities correspond to the junction area A_j, which is described in this section and Section 2.3.1. In single-phase, the velocities are equal. This is followed by MASS FLOW, mass flow rate $[(\dot{\alpha}_{f,j}^{n}\dot{\rho}_{f,j}^{n}v_{f,j}^{n+1} + \dot{\alpha}_{g,j}^{n}\dot{\rho}_{g,j}^{n}v_{g,j}^{n+1})A_j]$. The next two quantities are JUN. AREA, junction area (A_j) and THROAT RATIO, throat ratio $\frac{A_T}{A_j}$, where A_T is the physical junction area at the throat (see Section 2.3.1 for more detail). For the smooth area option, A_j is the physical junction area (full open area if a valve). When using the smooth area option, the throat ratio is set to 1.0, except for valves where it may be less than 1.0, For the abrupt area option, A_j is the minimum area of the two connecting volumes. The throat ratio is the ratio of the physical junction area to the defined junction area A_j. When using the abrupt area option, this quantity may be less than 1.0 for orifices and valves. The velocities are based on the junction area A_j. The next quantity is the junction control flag (JUNCTION FLAGS), which is the eight-digit packed number jefvcahs that the user inputs for each junction. The next quantity is the junction flow regime, FLOW REGI; see Section 2.3.1 of this volume of the manual for the meaning of the flow regime label. The last three columns are a choking summary (NO.ADVS.CHOKED). The subheading LAST indicates whether the choking model was applied on the last time step (set to 1 if it was, set to 0 if it was not). The subheading EDIT lists the number of times the choking model was applied since the last major edit; the subheading TOTAL lists the number of times the choking model was applied for the entire problem. As with the first section of the hydrodynamic volume information, quantities within each system are grouped by component, with the component name and type printed above the quantities.

8.3.2.11 Hydrodynamic Junction Information--Second Section. This section of output is optional and can be skipped by setting bit two in the ss digits of Word 4 (W4) on the time step control cards (Cards 201 through 299). This section is printed in **Figure 8.3-1**. As with the second section of the hydrodynamic volume information, no system information is printed, no component label information is printed, no additional component quantities are printed, and all quantities are printed in numerical order within each system. The junction number (JUN. NO.) and twelve other quantities are next printed out on each line. These are printed out in numerical order within each system. The quantities are VOIDFJ, liquid junction volume fraction ($\dot{\alpha}_{f,j}^{n+1}$); VOIDGJ, vapor/gas junction volume fraction ($\dot{\alpha}_{g,j}^{n+1}$); FIJ, interphase friction coefficient C_i^n; FWALFJ and FWALGJ, dimensionless liquid and vapor/gas wall friction coefficients $[\left(2 \bullet FWF_j^n \bullet \frac{\Delta x_j}{|v_{f,j}^n|}\right)$ and $\left(2 \bullet FWG_j^n \bullet \frac{\Delta x_j}{|v_{g,j}^n|}\right)$ in most cases, where the dimensionless phase wall friction coefficients, FWALFJ and FWALGJ, represent a dimensionless friction factor times a length divided by a

hydraulic diameter]; user-specified dimensionless forward and reverse flow energy loss coefficients, FJUNF and FJUNR [$\left(2 \bullet \frac{HLOSSF_j^n}{\left|v_{f,j}^n\right|}\right)$ and $\left(2 \bullet \frac{HLOSSG_j^n}{\left|v_{g,j}^n\right|}\right)$ in most cases, where FJUNF is used in HLOSSF and HLOSSG for forward flow and FJUNR is used in HLOSSF and HLOSSG for reverse flow]; and the dimensionless abrupt area change liquid and vapor/gas loss coefficients, FORMFJ and FORMGJ [$\left(2 \bullet \frac{HLOSSF_j^n}{\left|v_{f,j}^n\right|}\right)$ and $\left(2 \bullet \frac{HLOSSG_j^n}{\left|v_{g,j}^n\right|}\right)$ in most cases, where FORMFJ is used in HLOSSF and FORMGJ is used in HLOSSG]. The previous six quantities were all made dimensionless so that the relative importance of each in the momentum equations could be determined from the major edits. The last three quantities are a countercurrent flow limitation (CCFL) model summary (NO. ADVS. CCFL). The subheading LAST indicates whether the CCFL model was applied on the last time step (set to 1 if it was or set to 0 if it was not); the subheading EDIT lists the number of times the CCFL model was applied since the last major edit; the subheading TOTAL lists the number of times the CCFL model was applied for the entire problem.

8.3.2.12 Hydrodynamic Junction Time Step Control Information. This section prints whenever the hydrodynamic volume time step control information (Section 8.3.2.9 of this volume of the manual) prints. This section has been recently added and is not shown in **Figure 8.3-1**. Columns under the REPEAT headings are incremented if an advancement is repeated for several different reasons. If water packing is detected in a volume and the liquid momentum equation in a junction is changed as a result, the quantity under the heading REPEAT-PACKING is incremented, and the time step is repeated with the same time step size. If a velocity flip-flop has occurred in a junction, the quantity under the heading REPEAT-FLIP FLOP is incremented, and the time step is repeated with the same time step size.

8.3.2.13 Heat Structure-Heat Transfer Information. This section of output is not optional and always appears in a major edit when heat structures are present. Quantities in this section are printed in numerical order. The first printed quantity for each heat structure is the individual heat structure number STR. NO., denoting the heat structure-geometry number (CCCG) and the three-digit individual heat structure subfield number (0NN). These numbers are separated by a hyphen (-). Following this, nine quantities are printed out for both sides of the heat structure. First, the surface indicator is printed for both sides (SIDE, printed as either LEFT or RIGHT). Next, the volume number for the hydrodynamic volume connected on each side is printed (BDRY. VOL. NUMBER, 0-000000 is printed if no volume is present). Then the surface temperature is printed for both sides (SURFACE TEMP.). After this is the heat transfer rate out of the heat structure for both sides (HEAT-TRF. CONVECTION). A positive heat transfer rate represents heat flow out of the heat structure. This is followed by two fluxes for both sides, the heat flux and the critical heat flux (HEAT-FLUX CONVECTION and CRITICAL HEAT-FLUX). A positive heat flux represents heat flow out of the heat structure. After these, the critical heat flux multiplier, the mode of heat transfer, and the heat transfer coefficient are printed for both sides (CHF-MUL, HT MODE, and HEAT-TRF. COEF.CONV.). The multiplier is the coefficient multiplied times the critical heat flux found in the CHF table to obtain the final value printed here. Section 3.2 describes the meaning of the modes. The heat transfer coefficient is discussed in Volume IV. Finally, three quantities are printed for the individual heat structure. These are the heat generated within the structure (INT.-HEAT SOURCE), the net heat transfer rate out of the structure [i.e., convection plus radiation/conduction enclosure minus generation (CONV+RAD-SOURCE)], and the volume-average temperature for the structure (HT-ST VOL AVE

TEMP). **Figure 8.3-1** shows an example of this section of the major edit. Following this section, the sum of the sources is given.

8.3.2.14 Heat Structure Temperature. This section of output is optional and can be skipped by setting bit one in the ss digits of Word 4 (W4) on the time step control cards (Cards 201 through 299). As in the first heat structure section, the individual heat structure number (STR. NO) is printed in the first column. Then, all the mesh point temperatures (MESH POINT TEMPERATURES) for the individual heat structure are printed, starting with the left side and proceeding toward the right side (read from left to right across the page). In **Figure 8.3-1**, 11 mesh point temperatures are printed out.

8.3.2.15 Reflood Information. This section of output is not optional and always appears in a major edit when heat structures are present and the reflood model is turned on. Once the model is turned on, it stays on, and this section continues to be printed out. **Figure 8.3-3** shows an example of this section preceded by the normal heat structure printouts. The section begins with the label REFLOOD EDIT and the time. The first quantity printed is the heat structure-geometry number (CCCG, labeled GEOM. NO.). Following this are two columns providing information about the number of axial nodes (AXIAL NODES NUMBER). The first of these columns is the assigned maximum number of axial nodes (MAXIMUM). This number is computed at input time, and it is the theoretical maximum [(number of heat structures with this geometry) • (maximum number of axial intervals) + 1] when the user requests 2, 4, or 8 maximum number of axial intervals. Owing to storage limitations, this number is calculated by a formula that reduces the number below the theoretical maximum for 16, 32, 64, or 128 maximum number of axial intervals. For the example in **Figure 8.3-3**, the user requested 16, so the theoretical maximum is 321, which is larger than the assigned maximum of 153. The next column is the actual number of axial nodes used for the last time advancement (EDIT), and, in this case, it is 59. If the EDIT column is ever larger than the MAXIMUM column, the code will abort. The next four quantities are used in deciding on the number of nodes needed to define the boiling curve. The first three are the wall temperature at incipience of boiling, INC. BOIL. TEMP. (T_{IB}); the wall temperature at critical heat flux, CRITICAL TEMP. (T_{CHF}); and the wall rewetting or quench temperature, REWETTING TEMP. (T_Q). These numbers are set to 5 degrees below and 40 and 250 degrees above the saturation temperature, respectively. The final number is the location of the critical temperature, CRIT. TEMP. POSITION. This location is the distance from the start of the first heat structure. This last output does not appear to be working, but the quench position can be found by examining the next two sections. Next is the axial position of all 59 nodes, followed by the left- and right-side surface temperatures at these axial positions.

This axial section of output is optional, and it is skipped when the heat structure temperatures are skipped. As with the previous section on reflood information, this section is not printed until the reflood model is turned on, and then it continues to be printed out. An example of this section is also shown in **Figure 8.3-3**. The temperatures are printed from left to right, beginning with the first heat structure. In this example of 20 heat structures, 59 axial mesh point surface temperatures are printed.

8.3.2.16 Cladding Oxidation and Rupture Information. If the user has activated the metal-water reaction model by using a 1CCCG003 card, the cladding inside and outside oxide penetration depth (i.e., thickness) is printed prior to the heat structure output. **Figure 8.3-4** gives an example where there are two stacks of eight heat structures. The first stack (21) and the second stack (31) at each elevation show some outside cladding oxidation. The second stack (31) at elevation 5 shows some inside cladding oxidation.

```
                    Reflood edit at time=    6.556816E-02 sec
geom.no.  axial nodes number  inc. boil.   critical     rewetting    crit. temp.
          maximum    edit     temp.        temp.        temp.        position
                              (K)          (K)          (K)          (m)
  61          153      59      510.14       555.14       765.14      0.00000E+00

Present position of axial mesh (0.0 is bottom)  (m)
           0.00000E+00 1.14300E-02 2.28600E-02 3.42900E-02 4.57200E-02 5.71500E-02 6.85800E-02 8.00100E-02 9.14400E-02 0.10287
           0.11430     0.12573     0.13716     0.14859     0.16002     0.17145     0.18288     0.27432     0.36576     0.45720
           0.54864     0.64008     0.73152     0.82296     0.91440      1.0058      1.0973      1.1887      1.2802      1.3716
            1.4630      1.5545      1.6459      1.7374      1.8288      1.9202      2.0117      2.1031      2.1946      2.2860
            2.3774      2.4689      2.5603      2.6518      2.7432      2.8346      2.9261      2.9718      3.0175      3.0632
            3.1090      3.1547      3.2004      3.2461      3.2918      3.3833      3.4747      3.5662      3.6576

geom.no.  surface axial mesh point temperatures (K)
and side
  61 left   469.29      473.46      477.65      481.84      486.02      490.21      494.40      498.59      502.78      506.97
            511.16      515.34      519.53      523.72      527.91      532.10      536.29      569.80      613.17      656.55
            694.73      732.92      784.23      835.55      883.84      932.11      946.97      961.83      984.99      1008.2
            1033.0      1057.8      1082.7      1107.6      1108.3      1109.0      1097.2      1085.4      1079.5      1073.6
            1032.9      992.32      952.23      912.13      871.17      830.20      780.18      755.16      730.15      713.58
            697.00      680.43      663.85      650.65      637.46      611.06      601.23      591.40      581.57
  61 right  471.45      475.61      479.78      483.95      488.12      492.29      496.46      500.63      504.80      508.97
            513.14      517.31      521.48      525.65      529.82      533.99      538.05      571.30      614.73      658.07
            696.02      733.91      784.68      835.36      882.87      930.31      944.58      958.83      981.16      1003.5
            1027.2      1050.9      1074.5      1098.1      1098.1      1098.0      1085.5      1073.0      1065.7      1058.4
            1016.3      974.46      942.90      910.13      861.79      815.16      768.24      745.17      721.40      705.53
            690.41      674.96      659.10      646.32      633.89      608.50      598.81      589.07      579.50
```

Figure 8.3-3 Example of reflood major edit.

This is because this elevation has ruptured, as can be seen in the next section in **Figure 8.3-4**. The pressure shown is the pressure inside the gap**.**

8.3.2.17 Radiation Enclosure Model Information. **Figure 8.3-5** shows an example of output from the radiation enclosure model. This radiation enclosure of six heat structures was tripped on at 0.25 seconds and was never tripped off. The radiation enclosure heat flux is out of five structures and into number six. The radiation enclosure heat flux is determined from Volume I, Section 4.13, Equation (4.13-5) in the manual. In the radiation enclosure energy exchange calculation, -0.28 W is unaccounted for. The column labeled 'radiation heat flux' has an area associated with it, which is the surface area of the heat structure for the specified boundary (left or right). The radiation heat flux is multiplied by this area to get the quantity in the column labeled 'radiation energy'.

8.3.2.18 Conduction Enclosure Model Information. This is output from the conduction enclosure model that is not shown in **Figure 8.3-5**. The printout is very similar to the radiation enclosure model printout. The conduction enclosure heat flux is determined from Volume I, Section 4.14, Equation (4.14-1) in the manual.

8.3.2.19 Control Variable Information. This section of output is not optional and always appears in a major edit when control systems are present. **Figure 8.3-1** shows examples of such printout, which begins with the label CONTROL VARIABLE EDIT. Four items are printed for each variable, with two sets of information printed per line. The four items are the control variable number (NNN), the alphanumeric name of the control variable, the control component type, and the value of the control variable at the end of the last advancement.

8.3.2.20 Generator Information. This section of output is not optional and always appears when a generator control component is present. This is not shown in **Figure 8.3-1**. As discussed in Volume 1, the generator component is an optional feature of the shaft component. As a result, the first column under the GENERATOR label in the major edit is the control variable number (NNN) of the corresponding shaft component. To the right of this, under normal operating conditions, is the torque exerted by the generator (TORQUE). Under normal conditions, the torque will be negative, since it is required to turn the generator. The next quantity printed, under normal conditions, is the power applied by the generator (INPUT POWER). Again, under normal conditions, the power will be negative.

8.3.2.21 Nodal Kinetics Information. This section of output describes the reactor power in more detail if the user has selected the nodal kinetics power option in his input deck. This is not shown in **Figure 8.3-1**. The printed output begins with the reactor power from fission, power from the decay of fission products, and power from the from the decay of actinides. This information is followed by a printout of the positions of all of the control rods. This is followed by a printout of the power and thermal hydraulic conditions in the kinetics zones. A printout of the axial and radial relative power shapes follows the printout of the zone conditions. The final subsection of the portion of the nodal kinetics printout lists the neutron cross sections in each of the kinetics nodes. The printing of the control rod data, the zone thermal hydraulic data and the neutron cross section data can be controlled by the user through input cards 4 and 5 (see Section 2.2 of Appendix A of the volume) to reduce the size of the printed output file.

Str.no.	side	penetration depth (m)
21-001	inside	0.00000E+00
21-001	outside	3.85866E-09
21-002	inside	0.00000E+00
21-002	outside	3.65601E-08
21-003	inside	0.00000E+00
21-003	outside	1.52705E-07
21-004	inside	0.00000E+00
21-004	outside	3.07263E-07
21-005	inside	0.00000E+00
21-005	outside	3.36703E-07
21-006	inside	0.00000E+00
21-006	outside	2.10397E-07
21-007	inside	0.00000E+00
21-007	outside	7.34253E-08
21-008	inside	0.00000E+00
21-008	outside	1.35699E-08
31-001	inside	0.00000E+00
31-001	outside	4.44638E-09
31-002	inside	0.00000E+00
31-002	outside	5.52415E-08
31-003	inside	0.00000E+00
31-003	outside	2.15746E-07
31-004	inside	0.00000E+00
31-004	outside	3.91791E-07
31-005	inside	1.83263E-07
31-005	outside	2.52239E-07
31-006	inside	0.00000E+00
31-006	outside	2.89031E-07
31-007	inside	0.00000E+00
31-007	outside	9.74868E-08
31-008	inside	0.00000E+00
31-008	outside	1.49986E-08

Total hydrogen generated 3.82445E-04 (kg)

Str.no.	gas gap m,	clad radius m,	ruptured	pressure (Pa)
21-001	1.07668E-04	6.15165E-03	no	1.25221E+07
21-002	1.04248E-04	6.15778E-03	no	
21-003	1.02303E-04	6.16311E-03	no	
21-004	1.01429E-04	6.16637E-03	no	
21-005	1.01504E-04	6.16685E-03	no	
21-006	1.02673E-04	6.16458E-03	no	
21-007	1.05300E-04	6.16027E-03	no	
21-008	1.09669E-04	6.15496E-03	no	
31-001	1.04044E-04	6.14859E-03	no	4.13800E+06
31-002	1.00966E-04	6.15537E-03	no	
31-003	9.81420E-05	6.16033E-03	no	
31-004	9.61598E-05	6.16288E-03	no	
31-005	7.39275E-04	6.80334E-03	yes	
31-006	9.92277E-05	6.16150E-03	no	
31-007	1.01248E-04	6.15719E-03	no	
31-008	1.05442E-04	6.15135E-03	no	

Figure 8.3-4 Example of cladding oxidation and rupture major edit.

```
Radiation set 1. last time when radiation calculation became active was    0.25000
                    last time when radiation calculation became inactive was    0.00000E+00
Num   str.no. side    radiation      radiation
                       heat flux      energy
                       (Watt/m2)      (Watt)
  1   2111-  1 right    8159.0         1507.2
  2   2222-  1 right    8534.2         2627.5
  3   2333-  1 right    7788.8         1918.4
  4   2444-  1 right    6320.7         1264.9
  5   2555-  1 right    677.39         31.282
  6   2666-  1 left    -34278.        -7349.6
     The sum of the radiation energy=   -0.28
```

Figure 8.3-5 Example of radiation major edit.

8.3.3 Minor Edits

Minor edits are condensed edits of user-specified quantities. The frequency of minor edits is user-specified and may be different from the major edit frequency. **Figure 8.3-6** shows one page of minor edits. The selected quantities are held until 50 time values are stored. The minor edit information is then printed, 50 time values on a page, nine of the selected quantities per page, with time printed in the leftmost column on each page. Minor edits can print selected quantities at frequent intervals using much less paper than major edits. Section 4 of Appendix A of this volume indicates how to request minor edits and what the user-specified quantities represent. Minor edits can be controlled by the user by use of input cards 4 or 5 (see Section 2.2 of Appendix A of this volume) to control the size of the printed output file.

8.3.4 Diagnostic Edit

During a transient (TRANSNT on Card 100) or steady-state (STDY-ST on Card 100) problem, additional tables of variables can be printed out by inputting Words 4 and 5 on Card 105, or the tables often will be printed out when a failure occurs. These tables will be discussed in this section. This printout contains key variables from the hydrodynamic and heat transfer subroutines. The main variable in the code that activates this output is the variable HELP. Normally, HELP = 0, and no diagnostic printout occurs. The various ways that this diagnostic edit can occur will be presented, along with the value of the variable HELP. Some examples of the type of printout that occurs in the diagnostic edit will also be presented.

One way a diagnostic edit occurs is when it is forced out for more than one time step. This can be done by inputting Words 4 and 5 on Card 105 which sets HELP = 3, which will force out the hydrodynamic diagnostic edit. This, in turn, will set IWRITE = 1 in the heat transfer subroutines, forcing out the heat transfer diagnostic edit. The diagnostic edit will continue to appear for successive time steps until the count number reaches W5. Then, the calculation will stop. This method is often used by the development staff in debugging the code. An example of a diagnostic edit for one time step when HELP = 3 is presented in Appendix B.

Another way a diagnostic edit can occur is to set HELP = 2 with a debugger in any of the hydrodynamic subroutines. This will force out the diagnostic edit for the remainder of the hydrodynamic subroutines in this time step. Then, the time step will be repeated with HELP set to -2 and IWRITE set to 1

time (sec)	p 3010000 (Pa)	p 3050000 (Pa)	p 3100000 (Pa)	p 3150000 (Pa)	p 3200000 (Pa)	quals 3010000	quals 3200000	velfj 3010000 (m/sec)	velgj 3010000 (m/sec)
0.000000E+00	7.00000E+06	7.00000E+06	7.00000E+06	7.00000E+06	7.00000E+06	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
2.000000E-03	6.09595E+06	5.59016E+06	3.90506E+06	2.63640E+06	2.62530E+06	0.00000E+00	3.15487E-03	0.21781	0.21781
4.000000E-03	1.37441E+06	1.49439E+06	2.04011E+06	2.53500E+06	2.51717E+06	0.00000E+00	9.50141E-03	0.35402	0.35402
6.000000E-03	1.96118E+06	2.38019E+06	2.55980E+06	2.64681E+06	2.45216E+06	1.03229E-04	1.24858E-02	0.54599	-0.61598
8.000000E-03	2.61603E+06	2.61606E+06	2.63588E+06	2.65547E+06	2.46592E+06	1.67083E-04	1.21614E-02	0.12627	-1.7106
1.000000E-02	2.67931E+06	2.65608E+06	2.65518E+06	2.65501E+06	2.49875E+06	2.04469E-04	1.06812E-02	0.21780	-1.2075
1.200000E-02	2.67327E+06	2.65938E+06	2.65889E+06	2.65107E+06	2.52564E+06	2.45838E-04	9.28386E-03	0.26593	-0.80086
1.400000E-02	2.66856E+06	2.66000E+06	2.65810E+06	2.64618E+06	2.53806E+06	2.92285E-04	8.53122E-03	0.29181	-0.49368
1.600000E-02	2.66440E+06	2.65952E+06	2.65666E+06	2.64487E+06	2.53504E+06	3.41520E-04	8.47477E-03	0.30592	-0.28538
1.800000E-02	2.66036E+06	2.65806E+06	2.65475E+06	2.64555E+06	2.51880E+06	3.92138E-04	8.98618E-03	0.31254	-0.15440
2.000000E-02	2.65624E+06	2.65578E+06	2.65247E+06	2.64469E+06	2.49330E+06	4.43197E-04	9.91746E-03	0.31422	-7.47591E-02
3.000000E-02	2.63579E+06	2.63869E+06	2.63721E+06	2.62933E+06	2.37695E+06	6.90530E-04	1.53013E-02	0.29283	3.46642E-02
4.000000E-02	2.61627E+06	2.61820E+06	2.61488E+06	2.60294E+06	2.35334E+06	9.15933E-04	1.56999E-02	0.26775	5.00349E-02
5.000000E-02	2.59669E+06	2.59811E+06	2.60207E+06	2.59739E+06	2.32639E+06	1.12553E-03	1.61790E-02	0.25628	7.19093E-02
6.000000E-02	2.58577E+06	2.58423E+06	2.58937E+06	2.60217E+06	2.31729E+06	1.33803E-03	1.63810E-02	0.26613	0.11720
7.000000E-02	2.58946E+06	2.58786E+06	2.57898E+06	2.49350E+06	2.30182E+06	1.57807E-03	1.67368E-02	0.27204	0.14126
8.000000E-02	2.59205E+06	2.59227E+06	2.58084E+06	2.45390E+06	2.29184E+06	1.80915E-03	1.68859E-02	0.25649	0.10747
9.000000E-02	2.58958E+06	2.59177E+06	2.58565E+06	2.39619E+06	2.29192E+06	2.03421E-03	1.70636E-02	0.25519	0.10989
0.100000	2.58633E+06	2.58749E+06	2.58992E+06	2.36217E+06	2.28174E+06	2.25691E-03	1.72378E-02	0.25810	0.12238
0.110000	2.58230E+06	2.58331E+06	2.59420E+06	2.34174E+06	2.27952E+06	2.48113E-03	1.73292E-02	0.25970	0.13130
0.120000	2.59000E+06	2.58786E+06	2.60043E+06	2.32430E+06	2.27297E+06	2.70341E-03	1.75719E-02	0.23041	6.28296E-02
0.130000	2.58640E+06	2.58175E+06	2.48120E+06	2.31092E+06	2.26785E+06	2.91332E-03	1.78266E-02	0.24140	9.40326E-02
0.140000	2.58288E+06	2.57279E+06	2.43458E+06	2.30155E+06	2.26280E+06	3.12678E-03	1.80562E-02	0.23671	8.66769E-02
0.150000	2.57992E+06	2.56282E+06	2.39006E+06	2.29433E+06	2.25817E+06	3.34363E-03	1.82649E-02	0.22555	6.39216E-02
0.160000	2.57310E+06	2.55244E+06	2.35333E+06	2.28703E+06	2.25352E+06	3.56586E-03	1.84566E-02	0.22539	6.82506E-02
0.170000	2.56001E+06	2.53912E+06	2.33132E+06	2.27954E+06	2.24833E+06	3.80216E-03	1.86416E-02	0.22808	8.04724E-02
0.180000	2.55186E+06	2.52416E+06	2.31369E+06	2.27185E+06	2.24236E+06	4.05832E-03	1.88338E-02	0.22644	8.31920E-02
0.190000	2.54663E+06	2.43665E+06	2.29849E+06	2.26364E+06	2.23537E+06	4.33389E-03	1.90561E-02	0.25844	0.16503
0.200000	2.53246E+06	2.37423E+06	2.28615E+06	2.25508E+06	2.22724E+06	4.62236E-03	1.93277E-02	0.42554	0.55780
0.210000	2.51779E+06	2.33807E+06	2.27584E+06	2.24647E+06	2.21826E+06	5.11058E-03	1.96530E-02	0.83404	1.5376
0.220000	2.50015E+06	2.31672E+06	2.26673E+06	2.23758E+06	2.20886E+06	8.93717E-03	2.00231E-02	6.1628	11.401
0.230000	2.30867E+06	2.29389E+06	2.25941E+06	2.22924E+06	2.19934E+06	1.81870E-02	2.04248E-02	7.1194	7.4004
0.240000	2.16806E+06	2.22330E+06	2.24440E+06	2.22077E+06	2.19064E+06	2.65854E-02	2.08391E-02	4.2868	3.3204
0.250000	2.09789E+06	2.14246E+06	2.20520E+06	2.20649E+06	2.18064E+06	3.23567E-02	2.12484E-02	2.5818	1.3673
0.260000	2.06772E+06	2.08623E+06	2.14102E+06	2.17666E+06	2.16348E+06	3.65658E-02	2.17062E-02	1.7801	0.48090
0.270000	2.04889E+06	2.05279E+06	2.08302E+06	2.12231E+06	2.12665E+06	4.00966E-02	2.24343E-02	1.5093	0.35386
0.280000	2.02845E+06	2.02863E+06	2.04013E+06	2.05998E+06	2.05637E+06	4.34078E-02	2.38038E-02	1.4108	0.40779
0.290000	2.00588E+06	2.00501E+06	2.00481E+06	1.99889E+06	1.97047E+06	4.66588E-02	2.57386E-02	1.3539	0.48217
0.300000	1.97803E+06	1.97588E+06	1.96501E+06	1.93955E+06	1.89802E+06	4.98339E-02	2.77759E-02	1.3231	0.65219
0.310000	1.93495E+06	1.93150E+06	1.91683E+06	1.88863E+06	1.84206E+06	5.27627E-02	2.98041E-02	1.3268	1.0511
0.320000	1.87558E+06	1.87523E+06	1.87078E+06	1.84232E+06	1.79862E+06	5.53508E-02	3.19419E-02	1.2971	1.2496
0.330000	1.82688E+06	1.82559E+06	1.81483E+06	1.79327E+06	1.74986E+06	5.80595E-02	3.62839E-02	1.2369	1.2114

Figure 8.3-6 Example of minor edit.

in the heat transfer subroutines. As a result, the entire time step will be repeated with the diagnostic edit obtained for the hydrodynamic and heat transfer subroutines. After this, the code continues the calculation with HELP reset to 0, resulting in no further diagnostic edits.

The final way a diagnostic edit can occur is when a code failure occurs. This does not occur for every code failure, but it does occur for a large number of them. When this occurs, HELP will be set to 1 in most cases. When it is set to 1, the diagnostic edit will be forced out for the remainder of the time step. Then, the time step will be repeated with HELP set to -1 and IWRITE set to 1 in the heat transfer subroutines. As with the previous case, the entire time step will be repeated with diagnostic edit obtained for the hydrodynamic and heat transfer subroutines. For this case, however, the calculation terminates and a final major edit plus a minor edit are printed out.

There are two added printouts for this failure case (HELP = -1) that are an aid in tracing the code failure. Just preceding the diagnostic edit, information concerning the reason why the code failed is printed out. This information begins with eight asterisks (********). An example of this printout for the case of a thermodynamic property failure at the minimum time step is shown in the middle of **Figure 8.3-7**. Following this, the old time subroutine STATE diagnostic printout is forced out. The other message often printed out for this case (HELP = -1) can usually be buried somewhere within the diagnostic edit. For the example of a thermodynamic property error, information from the STATEP subroutine concerning the faulty volume is printed out (see middle of **Figure 8.3-8**). The information is the label THERMODYNAMIC PROPERTY FAILURE, the volume number (VOLNO), pressure (P), vapor/gas specific internal energy (UG), liquid specific internal energy (UF), noncondensable quality (QUALA), liquid volume fraction (VOIDF), and vapor/gas volume fraction (VOIDG). Further information on the specifics of the thermodynamic property failure such as in which phase the failure occurred is usually printed. This particular printout (using the semi-implicit hydrodynamic scheme) is located between the subroutine EQFINL and subroutine STATE diagnostic printouts. (No MASS ERROR diagnostic occurs for this failure.) The control of the diagnostic edits can be controlled by the user through the use of input cards 2 through 5 (see Section 2.2 of Appendix A of this volume) to control the size of the printed output file.

Failures that result in a diagnostic edit with HELP = -1 can be grouped into two cases. The first case occurs when the user is responsible. The thermodynamic property error mentioned above and shown in **Figure 8.3-7** and **Figure 8.3-8** can occur as a result of this. This can occur when the user inputs state properties that are undetected in input processing and thus get into the transient calculation. Thermodynamic property errors are the same as when either the REDUCE-PROPTY or REDUCE-EXTRAP flags are set in the major edit hydrodynamic volume time step control information block (see Section 8.3.2.9). Another example of a user-caused failure is when material property data are out of range. Two more user-caused failures can occur in the case of valves. If both motor valve trips become true at the same time, a failure will result. In addition, if the control system is set up incorrectly and this results in the servo valve stem position not being between 0 and 1, a failure will result. Another example is when a divide by 0 occurs in a control variable. The second case occurs as the result of a coding failure, which can be caused by a programming error or a model deficiency. Such a failure should be

```
 At time=    0.000000E+00 sec       str.no.   mesh point temperatures     (k)
61-001    502.48      502.48      502.48      502.48      502.48      502.48      502.48
61-002    569.54      569.54      569.54      569.54      569.54      569.54      569.54
61-003    656.32      656.32      656.32      656.32      656.32      656.32      656.32
61-004    732.58      732.58      732.58      732.58      732.58      732.58      732.58
61-005    835.17      835.17      835.17      835.17      835.17      835.17      835.17
61-006    931.68      931.68      931.68      931.68      931.68      931.68      931.68
61-007    961.37      961.37      961.37      961.37      961.37      961.37      961.37
61-008    1007.7      1007.7      1007.7      1007.7      1007.7      1007.7      1007.7
61-009    1057.3      1057.3      1057.3      1057.3      1057.3      1057.3      1057.3
61-010    1107.0      1107.0      1107.0      1107.0      1107.0      1107.0      1107.0
61-011    1108.4      1108.4      1108.4      1108.4      1108.4      1108.4      1108.4
61-012    1084.9      1084.9      1084.9      1084.9      1084.9      1084.9      1084.9
61-013    1073.1      1073.1      1073.1      1073.1      1073.1      1073.1      1073.1
61-014    991.89      991.89      991.89      991.89      991.89      991.89      991.89
61-015    911.68      911.68      911.68      911.68      911.68      911.68      911.68
61-016    829.82      829.82      829.82      829.82      829.82      829.82      829.82
61-017    729.82      729.82      729.82      729.82      729.82      729.82      729.82
61-018    663.62      663.62      663.62      663.62      663.62      663.62      663.62
61-019    610.82      610.82      610.82      610.82      610.82      610.82      610.82
61-020    591.15      591.15      591.15      591.15      591.15      591.15      591.15

---Restart no.       0 written, block no.     0---

******** Thermodynamic property error with minimum time step, transient being terminated.

******** Trouble, last advancement being repeated with  debug printout.

##########################################################################################################################
     state Diagnostic printout, timehy =  6.5567970E-02, dt =  1.9073486E-07, ncount =      1339, help = -1, lsuces =  2, fail = F

Volume mixture properties
==========================================================================================================================
      volno      v            p            voidf        quals        dotm         quala        boron        sounde       rho
                 vo           pps          voidg        quale        dotmo        sigma        borono       dsnddp       satt
==========================================================================================================================
      005010000  0.00000E+00  2.75790E+05   1.0000      0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00   1537.8       987.89
                 0.00000E+00  2.75790E+05  0.00000E+00 -0.15544      0.00000E+00  5.26935E-02  0.00000E+00  2.72067E+06   403.84
      006010000  2.83055E-03  1.54960E+06  9.46341E-03  0.20025       146.12      0.00000E+00  0.00000E+00   24.186       11.839
                 2.83055E-03  1.54960E+06  0.99054      9.54995E-03   146.12      3.78434E-02  0.00000E+00  4.97445E+06   472.99
      006020000  2.83055E-03  1.34989E+06  7.20929E-05  0.98079      -963.41      0.00000E+00  0.00000E+00   707.12       3.4851
                 2.83055E-03  1.34989E+06  0.99993       1.0000      -963.41      3.93030E-02  0.00000E+00  3.67281E+06   466.50
      006030000  2.83055E-03  1.26800E+06  2.67725E-04  0.93243       8.6735      0.00000E+00  0.00000E+00   704.85       3.4673
                 2.83055E-03  1.26800E+06  0.99973       1.0000       8.6735      3.99465E-02  0.00000E+00  3.65922E+06   463.62
      006040000  2.83055E-03  1.32421E+06  3.87655E-04  0.91149       14.999      0.00000E+00  0.00000E+00   694.77       3.8216
                 2.83055E-03  1.32421E+06  0.99961       1.0000       14.999      3.95017E-02  0.00000E+00  3.60120E+06   465.61
      006050000  2.83055E-03  1.39871E+06  4.81529E-04  0.90059       18.214      0.00000E+00  0.00000E+00   685.13       4.2117
                 2.83055E-03  1.39871E+06  0.99952       1.0000       18.214      3.89326E-02  0.00000E+00  3.54662E+06   468.15
      006060000  2.83055E-03  1.48181E+06  6.08762E-04  0.88749       19.338      0.00000E+00  0.00000E+00   673.86       4.6874
                 2.83055E-03  1.48181E+06  0.99939       1.0000       19.338      3.83229E-02  0.00000E+00  3.48886E+06   470.86
      006070000  2.83055E-03  1.57004E+06  7.46737E-04  0.87685       22.429      0.00000E+00  0.00000E+00   660.78       5.2366
                 2.83055E-03  1.57004E+06  0.99925       1.0000       22.429      3.77018E-02  0.00000E+00  3.42392E+06   473.61
      006080000  2.83055E-03  1.66580E+06  9.06057E-04  0.86712       39.730      0.00000E+00  0.00000E+00   647.34       5.8687
                 2.83055E-03  1.66580E+06  0.99909       1.0000       39.730      3.70553E-02  0.00000E+00  3.35791E+06   476.47
      006090000  2.83055E-03  1.77436E+06  1.10754E-03  0.85613       103.33      0.00000E+00  0.00000E+00   634.93       6.5977
```

Figure 8.3-7 Example of printout before the diagnostic edit when a failure occurs.

```
006180000    19  8.23918E-02  2.29640E-07  3.40446E-07  1.82739E+06  2.62915E+06  8.85948E+05  0.99870      0.00000E+00   9.8622
                 0.74161      0.00000E+00  0.00000E+00  1.82741E+06  2.62915E+06  8.85958E+05  0.99870      0.00000E+00   35.839
006190000    20  4.48173E-02  7.66357E-08  1.73934E-07  1.41060E+06  2.62187E+06  8.31420E+05  0.99907      0.00000E+00   7.6334
                 0.35177      0.00000E+00  0.00000E+00  1.41061E+06  2.62187E+06  8.31422E+05  0.99907      0.00000E+00   87.442
006200000    21  2.18821E-02  6.21175E-08  1.16083E-07  8.49402E+05  2.60382E+06  7.31203E+05  0.99948      0.00000E+00   4.6877
                 0.24154      0.00000E+00  0.00000E+00  8.49410E+05  2.60382E+06  7.31204E+05  0.99948      0.00000E+00   143.65
007010000    22  0.82303      9.74341E-06  1.19936E-05  2.75790E+05  2.65715E+06  5.49024E+05   1.0000      0.00000E+00   1.2690
                  29.009      0.00000E+00  0.00000E+00  2.75790E+05  2.65715E+06  5.49024E+05   1.0000      0.00000E+00  0.00000E+00

******** Thermodynamic property failure, volno=    6010000, p =   1.550289E+06, ug =   4.326621E+06, uf =   3.459355E+01
                                        quala =   0.000000E+00, voidf =   9.463429E-03, voidg =   9.905366E-01
******** Liquid phase property call had error.

##########################################################################################################################
     state Diagnostic printout, timehy =  6.5568161E-02, dt =  1.9073486E-07, ncount =      1340, help = -1, lsuces =  2, fail = F

Volume mixture properties
==========================================================================================================================
      volno      v            p            voidf        quals        dotm         quala        boron        sounde       rho
                 vo           pps          voidg        quale        dotmo        sigma        borono       dsnddp       satt
==========================================================================================================================
      005010000  0.00000E+00  2.75790E+05   1.0000      0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00   1537.8       987.89
                 0.00000E+00  2.75790E+05  0.00000E+00 -0.15544      0.00000E+00  5.26935E-02  0.00000E+00  2.72067E+06   403.84
      006010000  2.83055E-03  1.55029E+06  9.46343E-03  0.22472       146.08      0.00000E+00  0.00000E+00   86.615       10.557
                 2.83055E-03  1.55029E+06  0.99054      7.01770E-02   146.12      3.78386E-02  0.00000E+00  4.97394E+06   473.01
      006020000  2.83055E-03  1.35004E+06  7.25842E-05  0.98070      -924.80      0.00000E+00  0.00000E+00   707.13       3.4857
                 2.83055E-03  1.35004E+06  0.99993       1.0000      -963.41      3.93019E-02  0.00000E+00  3.67286E+06   466.50
      006030000  2.83055E-03  1.26804E+06  2.67783E-04  0.93242       8.6740      0.00000E+00  0.00000E+00   704.84       3.4675
                 2.83055E-03  1.26804E+06  0.99973       1.0000       8.6735      3.99462E-02  0.00000E+00  3.65918E+06   463.62
      006040000  2.83055E-03  1.32424E+06  3.87707E-04  0.91148       15.000      0.00000E+00  0.00000E+00   694.77       3.8217
                 2.83055E-03  1.32424E+06  0.99961       1.0000       14.999      3.95015E-02  0.00000E+00  3.60118E+06   465.61
      006050000  2.83055E-03  1.39873E+06  4.81578E-04  0.90059       18.212      0.00000E+00  0.00000E+00   685.12       4.2118
                 2.83055E-03  1.39873E+06  0.99952       1.0000       18.214      3.89324E-02  0.00000E+00  3.54660E+06   468.15
      006060000  2.83055E-03  1.48183E+06  6.08810E-04  0.88749       19.336      0.00000E+00  0.00000E+00   673.86       4.6875
                 2.83055E-03  1.48183E+06  0.99939       1.0000       19.338      3.83228E-02  0.00000E+00  3.48884E+06   470.86
      006070000  2.83055E-03  1.57006E+06  7.46785E-04  0.87685       22.424      0.00000E+00  0.00000E+00   660.78       5.2368
```

Figure 8.3-8 Example of printout buried in the diagnostic edit when a failure occurs.

reported to the development staff through the RELAP5-3D© User Services. Such errors often result in negative densities, bad viscosities, bad thermal conductivities, or thermodynamic property errors.

8.4 Plotted Output

The two methods normally used to obtain time plots of computed information are described below.

8.4.1 External Plots

The STRIP option (on Card 100) may be used to obtain ASCII data from the RSTPLT file. **Figure 8.4-1** shows an example of a strip input file. Data for all the parameters listed in the input file will appear on an ASCII file called STRIPF. These data must then be processed to put them into a format that is acceptable to the users' plotting software. XMGR[8.1-3] could be used to plot data from the STRIPF file. The INL usually uses XMGR5,[8.1-2] an INL extension to XMGR[8.1-3] that adds features to conveniently plot information from restart-plot files or STRIPF files.

```
=flecht-seaset separate effects reflood calculation: test 31504
*
100  strip fmtout
103  0
1001 httemp      006100407
1020 tempg       006040000
1030 voidg       006040000
1040 quale       006040000
1041 gammaw      006040000
1042 vapgen      006040000
1043 velg        006040000
1044 velf        006040000
1050 floreg      006040000
1060 htrnr       006100401
1061 hthtc       006100401
1062 htmode      006100401
.end
```

Figure 8.4-1 Strip input file.

If users plan on using external plots, they should make up their strip input file before generating the RSTPLT file because some parameters they may desire are on the RSTPLT file only if they are specifically requested with 2080XXXX cards.

8.4.2 Internal Plots

This code feature, designed at the INL for the CDC-176, has currently not been made compatible with the CRAY, workstations, and personal computers. The following discussion of this capability is printed so that it will be available after the capability is restored.

A plot package has been provided in RELAP5-3D© so that the user may produce graphs of calculational results. However, because each user may have a different use for the plots, many options are provided so that the user may design and vary the quality of plots as desired. In addition, since it is often necessary to compare the results to experiments or other calculations, a means to input plot comparison data tables has also been provided.

For convenience to the user, a check plot option is provided that will produce plots of input data, such as for time-dependent volumes and junctions, general tables, plot comparison data tables, valve area and flow coefficients, etc. This option can be used by the input of the *check plot* general plot request cards. The plots are constructed upon completion of the third phase of input data processing so that all information processed by the code will be included. Once the option is activated, it will remain in effect for all subsequent restarts and plot only jobs, including restarts with renodalization, until cancelled by the user with appropriate input.

It is assumed that each plot must be uniquely identified, and, hence, the run time, date, and code version is written in the plot margin oriented to appear on the edge that would be placed in a notebook binder. The plot heading and title are written at the top of the plot, and the axes labels and titles are written parallel to the left-hand and bottom axes. In addition, the curves plotted must lie within the axes' extremities and yet span as much of the axes as possible. The axes labeling subdivisions are also rounded to the first significant digit in order to produce simple labels.

Results can be plotted for any NEW or RESTART run. In addition, a PLOT run can be performed for which plots can be made of any variable stored in the plot record on the restart-plot file.

Plot input is analogous to the component input for NEW and RESTART problems in that once plot requests have been input, the resultant plot records and plot comparison data records are written to the restart-plot file. Hence, only input to delete, replace, insert, or add plot requests is required for successive RESTART or PLOT runs. In addition, undefined results are not plotted for components added or deleted by renodalization.

Some user inconvenience is apparent for input of plot comparison data tables because this input must be in an 80-character card image and must be part of the user input stream. If each data table is reasonably small, the user may manually produce the card images on a terminal. The tables may then be stored for future use with RELAP5-3D© runs or be made part of each problem input stream as desired. To produce plot comparison data tables from other restart-plot files, the RELAP5-3D© STRIP option may be used to retrieve results and build plot comparison data tables. If the data are contained on user tapes or disk files, the user can provide programs to build plot comparison data tables in the format required by RELAP5-3D©.

8.5 RELAP5-3D© Control Card Requirements

When run under the UNIX and LINUX operating system, the code includes processing of the command line that initiates RELAP5-3D© execution. This processing permits specification of some options and the names of the files such as input, output, and restart-plot files needed for execution (UNIX is an operating system available on Cray computers and most workstations.) The command line for execution under UNIX and LINUX is documented in the last section of the Input Data Requirements, Appendix A, of this volume of the manual.

8.6 Transient Termination

The transient advancement should not abort (terminate by operating system intervention) except for exceeding available disk space. Other program aborts such as floating point errors, illegal address, or segmentation faults are indications of programming errors and should be reported to the RELAP5-3D© development staff.

The user may optionally specify one or two trips to terminate a problem. Normal termination is from one of these trips or the advancement reaching the final time on the last time step control card. Minor and major edits are printed and a restart record is written at termination. Since trips can be redefined and new time step cards can be entered at restart, the problem can be restarted and continued.

Transient termination can also occur based on two tests on the CPU time remaining for the job. One test terminates if the remaining CPU time at the completion of a requested time step is less than an input quantity. The second test is similar, but the comparison is to a second input quantity and is made after every time advancement. The input quantity for the first test is larger than for the second test because the preferred termination is at the completion of a requested time step. In either case, the termination can be restarted.

Failure terminations can occur from several sources, including hydrodynamic solution outside the range of thermodynamic property subroutines, heat structure temperatures outside of thermal property tables or functions, and attempting to access an omitted pump curve. Attempting to restart at the point of failure or at an earlier time without some change in the problem input will only cause another failure. Problem changes at restart may allow the problem to be successfully restarted.

Additional information on terminating calculations is presented in Section 3 of Volume V.

8.7 Problem Changes at Restart

The most common use of the restart option is simply to continue a problem after a normal termination. If the problem terminated because it approached the CPU time limit, the problem can be restarted with no changes to information obtained from the restart file. If the problem stopped because the advancement time reached the end time on the last time step card, new time cards must be entered. If the

problem was terminated by a trip, the trip causing the termination must be redefined to allow the problem to continue. Thus, the code must provide for some input changes for even a basic restart capability.

The ability to modify the simulated system at restart is a desirable feature. The primary need for this feature is to provide for a transition from a steady-state condition to a transient condition. In many cases, simple trips can activate valves that initiate the transient. Where trips are not suitable, the capability to redefine the problem at restart can save effort in manually transcribing quantities from the output of one simulation to the input of another. One example of a problem change between steady-state and transient is the use of a liquid-filled, time-dependent volume in place of the vapor/gas region of a pressurizer during steady-state. The time-dependent volume provides the pressurizer pressure and supplies or absorbs liquid from the primary system as needed. The time-dependent volume is replaced by the vapor/gas volumes at initiation of the transient. This technique avoids modeling the control system that maintains liquid level and temperature during steady-state calculations when they are not needed in the transient.

Another reason for a problem change capability is to reduce the cost of simulating different courses of action at some point in the transient. An example is a need to determine the different system responses when a safety system continues to operate or fails late in the simulation. One solution is to run two complete problems. An alternative is to run one problem normally and restart that problem at the appropriate time with a problem change for the second case.

The problem change capability could also be used to renodalize a problem for a certain phase of a transient. This has not been necessary or desirable for problems run at the INL. For this reason, techniques to automate the redistribution of mass, energy, and momentum when the number of volumes changes have not been provided.

The current status of allowed problem changes at restart in RELAP5-3D© are summarized below. In all instances, the problem definition is that obtained from the restart file unless input data are entered for deletions, modifications, or additions. The problem defined after input changes must meet the same requirements as a new problem.

Time step control can be changed at restart. If time step cards are entered at restart, all previous time step cards are deleted. New cards need only define time step options from the point of restart to the end of the transient.

Minor edit, internal plot input data, and expanded edit/plot variables (2080XXXX) cards can be changed at restart. If any of the minor edit cards are entered, all previous cards are deleted. New cards must define all desired minor edit quantities. The internal plot request data and expanded edit/plot variables (2080XXXX) cards are handled in the same manner.

Trip cards can be entered at restart. The user can specify that all previous trips be deleted and can then define new trips. The user can also specify that the previously defined trips remain but that specific trips be deleted, be reset to false, be redefined, or that new trips be added.

Existing hydrodynamic components can be deleted or changed, and new hydrodynamic components can be added at restart. An especially useful feature is that the tables in time-dependent volumes and time-dependent junctions can be changed. If a hydrodynamic component is changed, all of the cards for the hydrodynamic component must be entered. If a hydrodynamic component is changed, all input that references this component (i.e., heat structures, reactor kinetics, radionuclide transport, etc.) may have to be changed and entered depending on the hydrodynamic component changes.

Interactive input data can be changed or added at restart.

Heat structures, radiation/conduction enclosures, general tables, and material properties can be deleted, changed, or added at restart. Heat structures can only be changed at the level of heat structure geometry. When the heat structure geometry is changed, all heat structures referencing that heat structure geometry are affected. Individual general tables and material properties can be added, deleted, or changed. The heat structures may be renodalized on restart (i.e., add a new heat structure, delete an existing heat structure, or replace an existing heat structure) if the radiation/conduction enclosure model is activated in the original run.

Control system components can be deleted, replaced, or added at restart.

Reactor kinetics data can be added, replaced, or deleted at restart. A complete set of reactor kinetics data must be input, i.e., individual sections of reactor kinetics data may not be specified as replacement data.

Radionuclide transport data can be added, replaced, or deleted at restart. A complete set of radionuclide transport data must be input, i.e., individual sections of radionuclide transport data may not be specified as replacement data.

In summary, all modeling features in RELAP5-3D© can be added, deleted, or replaced at restart.