
COBRA/TRAC - A Thermal-Hydraulics Code for Transient Analysis of Nuclear Reactor Vessels and Primary Coolant Systems

Users' Manual

Prepared by M. J. Thurgood, J. M. Cuta, A. S. Koontz,
J. M. Kelly

Pacific Northwest Laboratory
Operated by
Battelle Memorial Institute

Prepared for
U.S. Nuclear Regulatory
Commission

NOTICE

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, or any of their employees, makes any warranty, expressed or implied, or assumes any legal liability of responsibility for any third party's use, or the results of such use, of any information, apparatus, product or process disclosed in this report, or represents that its use by such third party would not infringe privately owned rights.

Availability of Reference Materials Cited in NRC Publications

Most documents cited in NRC publications will be available from one of the following sources:

1. The NRC Public Document Room, 1717 H Street, N.W.
Washington, DC 20555
2. The NRC/GPO Sales Program, U.S. Nuclear Regulatory Commission,
Washington, DC 20555
3. The National Technical Information Service, Springfield, VA 22161

Although the listing that follows represents the majority of documents cited in NRC publications, it is not intended to be exhaustive.

Referenced documents available for inspection and copying for a fee from the NRC Public Document Room include NRC correspondence and internal NRC memoranda; NRC Office of Inspection and Enforcement bulletins, circulars, information notices, inspection and investigation notices; Licensee Event Reports; vendor reports and correspondence; Commission papers; and applicant and licensee documents and correspondence.

The following documents in the NUREG series are available for purchase from the NRC/GPO Sales Program: formal NRC staff and contractor reports, NRC-sponsored conference proceedings, and NRC booklets and brochures. Also available are Regulatory Guides, NRC regulations in the *Code of Federal Regulations*, and *Nuclear Regulatory Commission Issuances*.

Documents available from the National Technical Information Service include NUREG series reports and technical reports prepared by other federal agencies and reports prepared by the Atomic Energy Commission, forerunner agency to the Nuclear Regulatory Commission.

Documents available from public and special technical libraries include all open literature items, such as books, journal and periodical articles, and transactions. *Federal Register* notices, federal and state legislation, and congressional reports can usually be obtained from these libraries.

Documents such as theses, dissertations, foreign reports and translations, and non-NRC conference proceedings are available for purchase from the organization sponsoring the publication cited.

Single copies of NRC draft reports are available free upon written request to the Division of Technical Information and Document Control, U.S. Nuclear Regulatory Commission, Washington, DC 20555.

Copies of industry codes and standards used in a substantive manner in the NRC regulatory process are maintained at the NRC Library, 7920 Norfolk Avenue, Bethesda, Maryland, and are available there for reference use by the public. Codes and standards are usually copyrighted and may be purchased from the originating organization or, if they are American National Standards, from the American National Standards Institute, 1430 Broadway, New York, NY 10018.

NUREG/CR-3046
PNL-4385
Vol. 3
R4

COBRA/TRAC - A Thermal-Hydraulics Code for Transient Analysis of Nuclear Reactor Vessels and Primary Coolant Systems

Users' Manual

Manuscript Completed: November 1982
Date Published: March 1983

Prepared by
M. J. Thurgood, J. M. Cuta, A. S. Koontz,
J. M. Kelly

Pacific Northwest Laboratory
Richland, WA 99352

Prepared for
Division of Accident Evaluation
Office of Nuclear Regulatory Research
U.S. Nuclear Regulatory Commission
Washington, D.C. 20555
NRC FIN B2391

ABSTRACT

The COBRA/TRAC computer program has been developed to predict the thermal-hydraulic response of nuclear reactor primary coolant systems to small and large break loss-of-coolant accidents and other anticipated transients. The code solves the compressible three-dimensional, two-fluid, three-field equations for two-phase flow in the reactor vessel. The three fields are the vapor field, the continuous liquid field, and the liquid drop field. A five-equation drift flux model is used to model fluid flow in the primary system piping, pressurizer, pumps, and accumulators. The heat generation rate of the core is specified by input and no reactor kinetics calculations are included in the solution. This volume is the Users' Manual. It contains the input instructions for COBRA/TRAC and its auxiliary programs, SPECSET and GRAFIX. It also contains a users' guide to the code and is intended to aid the new user in becoming familiar with the capabilities and modeling conventions of the code.

CONTENTS

ACKNOWLEDGEMENTS	xi
1.0 INTRODUCTION	1.1
2.0 COBRA/TRAC INPUT INSTRUCTIONS.....	2.1
2.1 MAIN PROBLEM CONTROL DATA.....	2.4
2.2 ONE-DIMENSIONAL COMPONENT DATA	2.9
2.2.1 ACCUMulator Component Data	2.10
2.2.2 BREAK Component Data	2.12
2.2.3 FILL Component Data	2.15
2.2.4 PIPE Component Data	2.17
2.2.5 Pressurizer (PRIZER) Component Data.....	2.21
2.2.6 PUMP Component Data	2.24
2.2.7 Steam Generator (STGEN) Component Data.....	2.32
2.2.8 TEE Component Data	2.38
2.2.9 VALVE Component Data	2.45
2.3 VESSEL (3-D) COMPONENT DATA	2.51
2.4 TIME DOMAIN DATA	2.102
2.5 SPECSET REDIMENSIONING PROGRAM	2.105
2.6 GRAFIX POSTPROCESSOR PROGRAM	2.119
3.0 USER'S GUIDE TO COBRA/TRAC	3.1
3.1 COBRA/TRAC VESSEL COMPONENT GEOMETRY MODEL.....	3.1
3.1.1 Group 2: Channel Input Data.....	3.17
3.1.2 Group 3: Transverse Channel Connection Data.....	3.22
3.1.3 Group 4: Section Data.....	3.31
3.1.4 Groups 5 and 6: Axial Variations in Channels and Gaps.....	3.36
3.1.5 Group 7: Vertical Loss Coefficients.....	3.42
3.1.6 Group 12: Turbulent and Viscous Stress Data.....	3.44
3.2 VESSEL COMPONENT ROD AND UNHEATED CONDUCTOR MODEL.....	3.46
3.2.1 Group 8: Description of Rods and Unheated Conductors.....	3.46
3.2.2 Group 9: Geometry Description for Conduction Model.....	3.52
3.2.3 Group 10: Material Properties Table.....	3.62

3.2.4	Group 11: Axial Power Profiles and Forcing Functions for Total Power and Gap Conductance.....	3.63
3.3	BOUNDARY AND INITIAL CONDITIONS FOR THE VESSEL COMPONENT...	3.68
3.3.1	Group 1: Initial Conditions.....	3.70
3.3.2	Group 13: Vessel Component Mesh Cell Boundary Conditions.....	3.72
3.3.3	Vessel Connections to One-Dimensional Components.....	3.75
3.4	SPECIFICATION FOR OUTPUT OF RESULTS FOR VESSEL COMPONENT...	3.77
3.4.1	Group 14: Line-Printer Output Options for the Vessel Component.....	3.77
3.4.2	Graphics Options.....	3.78
3.5	ONE-DIMENSIONAL COMPONENTS.....	3.86
3.5.1	Accumulator.....	3.87
3.5.2	Break and Fill.....	3.88
3.5.3	Pipe Module.....	3.91
3.5.4	Pressurizer.....	3.94
3.5.5	Pump Module.....	3.97
3.5.6	Steam Generator.....	3.108
3.5.7	Tee Module.....	3.112
3.5.8	Valve Module.....	3.114
3.6	STEADY STATE CALCULATIONS.....	3.116
3.7	USING THE DUMP/RESTART OPTION.....	3.118
3.7.1	Simple Restart.....	3.119
3.7.2	Restart with Changes to Components.....	3.121
3.7.3	Restart with New Components.....	3.122
4.0	SAMPLE PROBLEM 4.1	
4.1	PKL SYSTEM DESCRIPTION FOR TEST K9.....	4.2
4.2	COBRA/TRAC MODEL OF PKL SYSTEM.....	4.13
4.2.1	Loop Piping Components.....	4.13
4.2.2	Steam Generator Components.....	4.22
4.2.3	VESSEL COMPONENT.....	4.23
REFERENCES	R.1

FIGURES

3.1	Basic Mesh Cell.....	3.3
3.2	Mesh Cell for Vertical Velocities.....	3.3
3.3	Mesh Cell for Transverse Velocities.....	3.4
3.4	Basic VESSEL Component Channel.....	3.4
3.5a	Core Region	3.6
3.5b	Downcomer Region.....	3.8
3.5c	Core Bypass Region.....	3.9
3.5d	Lower Plenum Region.....	3.10
3.5e	Upper Plenum Region.....	3.11
3.5f	Upper Head Region.....	3.12
3.6	Core Section	3.13
3.7	Complete Vessel Noding Diagram for PWR with UHI.....	3.15
3.8	Channel Node Numbering Convention.....	3.19
3.9	Channel Connections at Section Boundaries Allowed by the Channel Splitting Logic.....	3.19
3.10	Typical Configurations for Convection of Transverse Momentum between Sections.....	3.21
3.11	Vertical Momentum Mesh Cell at Section Boundary.....	3.24
3.12	Global Coordinate Systems.....	3.26
3.13	Normal Component of Transverse Momentum.....	3.27
3.14	Convection of Transverse Momentum by an Orthogonal Transverse Velocity.....	3.30
3.15	Allowable Vertical Connections Between Channels at Section Boundaries.....	3.33
3.16	Common Channel Splitting Errors.....	3.34
3.17	Example of Cell Numbering Scheme for Simultaneous Solution Groups.....	3.37
3.18	Examples of Vertical Variations in Continuity and Momentum Area, and Wetted Perimeter of a Channel.....	3.39
3.19	Examples of Channels with Local Form Losses.....	3.43
3.20	Examples of Heat-Generating VESSEL Structures.....	3.48
3.21	Examples of Unheated VESSEL Structures.....	3.49
3.22	Fuel Pellet Radial Noding for the Conduction Model.....	3.54
3.23	Radial Noding in a Typical Electrical Heater Rod.....	3.61

3.24 Nuclear Fuel Rod Power Profile.....	3.65
3.25 Heat Input Over One Fluid Node.....	3.66
3.26 Heater Rod Crossing Section Boundaries.....	3.67
3.27 Power Forcing Function.....	3.69
3.28 Control Volume for Pressure Sink Boundary Condition.....	3.76
3.29 Core Inlet-Lower Core Support Plate Section.....	3.82
3.30 Upper Head	3.84
3.31 Accumulator Noding Diagram.....	3.89
3.32 Break Noding Diagram.....	3.90
3.33 Fill Noding Diagram.....	3.90
3.34 Pipe Noding Diagram.....	3.92
3.35 Pipe Noding Diagram for the Edwards Blowdown Experiment.....	3.93
3.36 Pressurizer Noding Diagram.....	3.96
3.37 Pump Noding Diagram.....	3.98
3.38 Single-Phase Homologous Head Curves.....	3.105
3.39 Fully Degraded Homologous Head Curves.....	3.105
3.40 Head Degradation Multiplier.....	3.106
3.41 Single-Phase Homologous Torque Curves.....	3.106
3.42 Fully Degraded Homologous Torque Curves.....	3.107
3.43 Torque Degradation Multiplier.....	3.107
3.44 Noding Diagrams for Once-Through and U-Tube Steam Generators.....	3.109
3.45 Tee Noding Diagram.....	3.113
3.46 Valve Noding Diagram.....	3.114
4.1 Diagram of PKL Test Facility as Configured for Test K9.....	4.3
4.2 Cross Section of PKL Core.....	4.8
4.3 Vertical Cross Section of PKL Pressure Vessel.....	4.9
4.4 Vertical Cross Section of PKL External Downcomer.....	4.10
4.5 Seven-Step Axial Power Profile for PKL Test K9.....	4.12
4.6 Diagram of COBRA/TRAC Model of PKL System.....	4.15
4.7 COBRA/TRAC Model of Intact Loop Hot Leg Piping (PIPE9).....	4.18
4.8 COBRA/TRAC Model of Intact Loop Cold Leg Piping (PIPE10).....	4.19
4.9 COBRA/TRAC Model of Broken Loop Hot Leg Piping (PIPE2).....	4.20

4.10 COBRA/TRAC Model of Broken Loop Cold Leg Piping (PIPE4 and PIPE7).....	4.21
4.11 COBRA/TRAC Model of PKL Pressure Vessel and Downcomer.....	4.35

TABLES

2.1	SPECSET Dimensioning Parameters.....	2.112
2.2	Sample Input File TAPE2 for SPECSET Program.....	2.118
3.1	Channel Boundary Condition Types.....	3.73
3.2	Definition of the Four Curve Segments Used to Describe Homologous Pump Curves	3.101
3.3	Pump Control Input Parameters.....	3.104
3.4	VALVE Activation Options.....	3.115
3.5	Input File for Simple Restart.....	3.120
3.6	Input File for a Restart with Changes to Components.....	3.123
3.7	Input File for a Restart with New Components Added.....	3.125
4.1	Geometry of PKL System.....	4.4
4.2	Geometry of PKL Steam Generators.....	4.7
4.3	Geometry of PKL Pressure Vessel.....	4.11
4.4	Material Properties for Heater Rods in PKL Test Facility.....	4.11
4.5	Initial Conditions of Test K9.....	4.14
4.6	Test Parameters and Boundary Conditions.....	4.17
4.7	COBRA/TRAC Input File for PKL Test K9.....	4.24

ACKNOWLEDGEMENTS

COBRA/TRAC is the result of the efforts of a number of people. We wish to acknowledge the main contributors and to express our appreciation to those who have offered their advice and suggestions.

The main contributors to the program are listed below.

Fluid Dynamics:	M. J. Thurgood, T. L. George, and T. E. Guidotti
Heat Transfer:	J. M. Kelly and R. J. Kohrt
Turbulence Model:	K. R. Crowell
Graphics and Programming:	A. S. Koontz
Simulations:	K. L. Basehore, S. H. Bian, J. M. Cuta, R. J. Kohrt, G. A. Sly, and C. A. Wilkins
One-Dimensional Components and Code Architecture:	Members of the TRAC Code Development Group at LANL

We wish to thank Dr. S. Fabric of the U.S. Nuclear Regulatory Commission for his patience, support, and suggestions during this large undertaking. We also wish to thank Drs. Tong, Shotkin, Han and Zuber of the U.S. Nuclear Regulatory Commission and members of the Advanced Code Review Group for their many helpful suggestions. We also express our gratitude to our manager, Dr. D. S. Trent, for his support, and Cathy Darby and Peggy Snyder for their lead roles in typing this report.

COBRA/TRAC - A THERMAL-HYDRAULICS CODE FOR TRANSIENT ANALYSIS
OF NUCLEAR REACTOR VESSELS AND PRIMARY COOLANT SYSTEMS
VOLUME 3: USERS' MANUAL

1.0 INTRODUCTION

The COBRA/TRAC computer program has been developed by merging the COBRA-TF^(a) and TRAC-PD2 (Ref. 1) codes. The COBRA/TRAC code is used to predict the thermal-hydraulic response of nuclear reactor primary coolant systems to small- and large-break loss of coolant accidents and other anticipated transients. The COBRA-TF computer code provides a two-fluid, three-field representation of two-phase flow with continuous liquid, continuous vapor and entrained liquid droplets. Each field is treated in three dimensions and is compressible. The conservation equations for each of the three fields and for heat transfer from and within the solid structures in contact with the fluid are solved using a semi-implicit finite-difference numerical technique on an Eulerian mesh. COBRA-TF features extremely flexible noding for both the hydrodynamic mesh and the heat transfer solution. This flexibility provides the capability to model the wide variety of geometries encountered in vertical components of nuclear reactor primary systems.

TRAC-PD2 is a systems code designed to model the behavior of the entire reactor primary system. It features special models for each component in the system, including accumulators, pumps, valves, pipes, pressurizers, steam generators and the reactor vessel. With the exception of the reactor vessel, the thermal-hydraulic response of these components is treated with a five-equation drift flux representation of two-phase flow. The vessel component of TRAC-PD2 is somewhat restricted in the geometries that can be modeled and

(a) COBRA-TF is a two-fluid thermal hydraulics code developed at PNL under sponsorship of the NRC, Reactor Safety Research Branch. Volumes 1 and 2 of this report are the first complete documentation of the COBRA-TF code.

cannot treat the entrainment of liquid drops from the continuous liquid phase directly. The TRAC-PD2 vessel module has been removed, and COBRA-TF has been implemented as the new vessel component in TRAC-PD2. The resulting code is COBRA/TRAC.

The COBRA/TRAC code has the extended capabilities in the vessel component module provided by the three-field representation of two-phase flow and the flexible noding of COBRA-TF, plus the ability to model the entire primary system with the one-dimensional component modules of TRAC-PD2.

The documentation of the COBRA/TRAC program consists of five volumes. Volume 1 contains a description of the basic three-field conservation equations and constitutive models used in the vessel component (COBRA-TF). Volume 2 describes the finite-difference equations for the vessel component and the numerical techniques used to solve these equations. The coupling between the TRAC-PD2 equations and the COBRA-TF vessel equations is also described. Volume 3 is the User's Manual. It contains the input instructions for COBRA/TRAC and its auxiliary programs, SPECSET and GRAFIX. It also includes a user's guide to the code and is intended to aid the new user in becoming familiar with the capabilities and modeling conventions of the code. Volume 4 is the Applications Manual containing the results of simulations assessing code performance. These include a variety of two-phase flow experiments conducted to simulate important phenomena anticipated during postulated accidents and transients. Volume 5 is the Programmer's Manual. It describes the programming characteristics of the code, including the overlay structure and machine-dependent features.

The COBRA/TRAC code is an integrated entity that solves the thermal-hydraulic equations for the reactor primary system as a whole, but from the standpoint of the user, the separate origins of COBRA-TF and TRAC-PD2 are readily apparent. The input for the VESSEL component--which is essentially the input for COBRA-TF--differs significantly in style and format from the input for the one-dimensional components, such as PIPES, TEES and PUMPS. The VESSEL component can be used to model any vertical component of the system in one, two or three dimensions. The one-dimensional components can be used to model parts of the system that are essentially pipes, some with special

characteristics, such as TEEs, PUMPs, VALVEs, ACCUMulATORS, PRessurIZERS, and Steam GENERATORS. The components connect one to another by means of flow junctions specified in the input. The system may consist of closed loops, as does a normal reactor primary system, controlled by pump speeds, pressurizers and trip setpoints. Special boundary conditions can be applied with BREAKs or FILLs, to model pipe breaks, open relief valves, broken feedwater lines and similar disruptions of the system.

The input for the COBRA/TRAC code is necessarily complex since it is designed to model extremely complex systems in a relatively sophisticated manner. To effectively utilize this code, the user must have an understanding of how the code works, what it does with the input data, and how to interpret the results.

Section 2.0 contains the input instructions for each of the components in COBRA/TRAC. Every line of input is listed for each component with the format it must be entered in and a brief explanation of the meaning and usage of each input variable. The one-dimensional components show a unity of structure and format that will be immediately apparent to the user and will make learning the input for these components comparatively easy. The one-dimensional components are essentially pipes, each with a few special lines of input to make them behave as their namesakes. The input for the VESSEL component, however, reflects the quite different modeling philosophy of COBRA-TF and is easily as long as the input instructions for all the other components combined.

The user's guide to Section 3.0 is devoted almost exclusively to explaining the modeling conventions and input requirements for the VESSEL component. This section discusses the input for the vessel hydrodynamic mesh, the rod input for the conduction model, and the various features for setting vessel boundary conditions. The DUMP/RESTART capability is discussed in detail, and there is also a recommended procedure for obtaining steady-state solutions with the code. Section 3.0 also contains a brief explanation of each of the one-dimensional components, consisting mainly of information garnered from the TRAC-P1A User's Manual (Ref. 2).

2.0 COBRA/TRAC INPUT INSTRUCTIONS

Input instructions for COBRA/TRAC are provided in this section. The input for this code is not simple. Users unfamiliar with COBRA/TRAC should refer to Section 3.0 for a more complete explanation of the input requirements before attempting to set up the data for a specific problem.

COBRA/TRAC input is divided into five major sections, each of which provides input for a particular purpose.

The first section provides data for the overall description of the problem, such as the number of components, number of trips and component identification numbers. Problem control parameters, such as convergence criteria and iteration limits, are entered in this section.

The second section provides data for the occurrence of trips. Trips control such events as switching pumps on or off and opening and closing valves under specific conditions.

In the third section, geometry and initial conditions for the one-dimensional components--ACCUMulators, BREAKs, PIPEs, PRessurIZERs, PUMPs, STeam GENERators, TEEs, and VALVEs--are defined.

The fourth section defines the VESSEL component input data. The geometry and initial and operating conditions for the vessel are specified in this section.

The fifth section defines the length of the calculation with the problem time domain data. Time intervals for the output file, restart file and graphics dump are also specified.

The data arrays for the first three sections, including the input for the one-dimensional components, are entered in the LOAD format. The LOAD format is an automated data entry procedure that can significantly reduce the volume of data that must be supplied, especially for the one-dimensional components. It permits:

- 1) automatic repetition of elements in an array
- 2) array filling with specified constants

- 3) interpolation between array elements to fill a specified number of intervening locations
- 4) skipping from an incomplete record to the next record to continue array filling.

The LOAD format has the following restrictions:

- 1) The end of data for each array must be identified with an E.
- 2) The array must be exactly filled (no over or underfilling allowed.)
- 3) Interpolation of integer data is not allowed.

The LOAD format allows for entry of up to five (5) array elements on a record. Each array element occupies 14 spaces on the record in the following format:

<u>Position</u>	<u>Format</u>	<u>Description</u>
1	A1	Operation control. Valid entries are: E End of array. F Fill the array starting at the current index with the specified constant. I Interpolate between this data constant and the succeeding data constant. M Multiple repetition of data constants; (repeats constant 10*I2 times--I2 described below). R Repetition of data constant I2 times; (I2 described below) S Skip to next card. Blank Enter the data constant in the array at the current index.
2-3	I2	Enter a counter for data constant repetition or interpolation. If there is an M in column 1, enter the number of tens of times the data constant will be repeated. (For example, if the data constant is to be repeated 20 times, enter 2 for I2.)

If there is an R in column 1, enter the number of times the data constant is to be repeated.

If there is an I in column 1, enter the number of values to be interpolated between the data constant and the data constant in the next 14-wide field.

If there is an E, F or S in column 1, I2 is left blank.

4-14 E11.2 Enter the data constant (real or integer number).
 or I11

An entire array can be filled with just one set of 14 columns on one input record. For example, the input record

F _____ 25.0E

will fill the entire array with the value of 25.0. (The user does not specify the size of the array since the LOAD format logic determines that automatically.)

2.1 MAIN PROBLEM CONTROL DATA, read by subroutine INPUT

INPUT.1: NUMTCR, ITRAC, ICOBRA FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NUMTCR	Enter the number of title cards to be read. (At least one title card must be entered.)
15-28	ITRAC	Enter input units option for the one-dimensional components: 0 = Use metric units (default). 1 = Use English units; code will convert all data to metric.
29-42	ICOBRA	Enter vessel input units option: 0 = Use English units (default). 1 = Use metric units; code will convert the data to English units.

NOTE: The working units in the code for the one-dimensional components are metric; the working units for the vessel component are English. Although the code will perform the units conversion if necessary, it is recommended that the one-dimensional component data be entered in metric units and the three-dimensional vessel data be entered in English units.

Title Card(s):

INPUT.2: (TITLE(I),I=1,20) FORMAT(20A4)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-80	TITLE(I)	Enter alphanumeric information describing this simulation.

Repeat card INPUT.2 NUMTCR times.

Restart Data:

INPUT.3: DSTEP, TIMET FORMAT(I14,E14.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	DSTEP	Enter the time step number of the dump to be used for restarting. Enter zero (0) if this is not a restart.
15-28	TIMET	Enter the restart time for the problem. Enter zero (0.0) if this is not a restart.

Component Data:

INPUT.4: NCOMP, NJUN, IPAK FORMAT(3I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCOMP	Enter the number of components.
15-28	NJUN	Enter the number of junctions.
29-42	IPAK	Enter the one-dimensional component water-packing option. Valid entries are: 0 = off 1 = on

Iteration Control:

INPUT.5: EPSO,OITMAX,IITMAX FORMAT(E14.0,2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	EPSO	Enter the outer iteration convergence criterion; suggested value = 0.001.
15-28	OITMAX	Enter the maximum number of outer iterations; suggested value = 5.
29-42	IITMAX	Enter the maximum number of vessel iterations; suggested value = 40.

Trip Control Data:

INPUT.6: NTRX,NDMPTR FORMAT(2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NTRX	Enter the number of trips.
15-28	NDMPTR	Enter the number of trips for which a dump is to be written when the trip is set.

Order Array:

The ORDER array contains a list of the identification numbers of the components in the system. Exactly NCOMP entries must be provided. The code rearranges the entries in the ORDER array to reflect the sequence in which the system of components will be solved, but the user may enter the numbers in any order he finds convenient.

INPUT.7: (ORDER(I),I=1,NCOMP) FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14,...	ORDER(I)	Enter the the NCOMP component numbers.

Enter NCOMP component numbers with a maximum of five (5) entries per card. The ORDER array is entered via the LOAD format previously described. The final entry in the array should be followed by an E.

Trips that Initiate Dumps:

INPUT.8: (IDMPTR(I),I=1,NDMPTR) FORMAT(5I14) Read only if NDMPTR > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14,...	IDMPTR(I)	Enter identification number of the trip that initiates dumps to the restart file.

Repeat this card until NDMPTR values have been entered with up to five (5) values per card. If NDMPTR is zero, this card is omitted.

TRIP SPECIFICATION, read by subroutine RDTRIP

Trips are used to simulate the actions of power plant protection systems during a transient or as reactions to abnormal conditions in the system. Trips control such things as valve openings and pump startup or shutdown. A problem may reference many trips, but each trip acts on only one component.

Trips are defined by a signal index, a signal set point, and a signal time delay. The signal index specifies the variable that will trigger the trip (such as pressure, temperature or transient time). The signal setpoint defines an upper or lower limit for the variable that will initiate the trip. The signal time delay simulates the time required for the operating system to respond to the trip signal.

RDTRIP.1a: ITID,ISID,TSP,TDT FORMAT(2I14,2E14.0) Read only if NTRX > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	ITID	Enter a unique trip identification number. If ITID > 1000 the time step size will be set to DTMIN.
15-28	ISID	Enter the trip signal index. Enter a negative value if the trip setpoint is to be the <u>lower</u> limit. Enter a positive value if the trip setpoint is to be the <u>upper</u> limit. Valid entries are: 0 = transient time ±1 = pressure ±2 = liquid temperature ±3 = vapor temperature ±4 = vapor fraction ±5 = wall temperature ±7 = accumulator or pressurizer water level ±8 = mixture velocity ±9 = relative velocity

29-42 TSP Enter the trip setpoint (the value of the trip signal variable that will initiate the trip).
 For example:
 If ISID = 2, enter the liquid temperature that will initiate the trip.

43-56 TDT Enter the trip delay time, in seconds.

RDTRIP.1b: ID1, ID2, ID3, ID4 FORMAT(4I14) Read only if NTRX > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	ID1	Enter the number of the component initiating this trip. Enter 0 if trip is on transient time (ISID=0). Note: Only one-dimensional component numbers are valid. Trips cannot be set on VESSEL variables.
15-28	ID2	Enter the component qualifier: 0 if ID1 does not specify a steam generator component number. 1 to trip according to conditions on the primary side of the steam generator. 2 to trip according to conditions on the secondary side of the steam generator.
29-42	ID3	Enter the node number of the array element to watch, if ISID = 1,2,3,4,5,8 or 9. If ISID = 0, enter zero. If ISID = 7, enter 1.
43-56	ID4	Enter a second (optional) node number of an array element to watch, if ISID = 1,2,3,4,5,8 or 9. Enter zero if ISID = 7 or 0.
Note:		If the trip is on reactor time (ISID = 0) ID1, ID2, ID3 and ID4 are not used. This card can be entered as a blank line.

RDTRIP.1 is read for NTRX trips. Note that this input consists of 2 lines of data for each trip.

2.2 ONE-DIMENSIONAL COMPONENT DATA

A set of data for each of the components in the problem is normally read from an input file. On a restart run, only those components that must be redefined are entered via the input file. The remaining components are obtained from the restart dump file. New components may be added to the system on a restart by adding them to the input file. This enables the user to change the physical characteristics of the system during the transient.

Component data consists of geometry data, information connecting the component to adjacent components, and data describing the fluid state in the component (such as void fraction, temperature and pressure). The following subsections list the input required for each one-dimensional component.

2.2.1 ACCUMulator Component Data, read by subroutine RACCUM

ACCUM.1 TYPE,NUM,ID,CTITLE FORMAT(A5,9X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	TYPE	Enter ACCUM
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number.
43-72	CTITLE	Enter alphanumeric identifier for this component.

NOTE: To minimize confusion, it is often convenient to set NUM and ID to the same value for each component.

ACCUM.2 NCELLS,JUN2 FORMAT(2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELLS	Enter the number of fluid cells.
15-28	JUN2	Enter the junction number to connect the accumulator to its adjacent component. (This is the accumulator discharge.)

ACCUM.3 through ACCUM.14 are entered using the LOAD format and define the accumulator data arrays.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>ACCUM.3</u>	DX array	Enter NCELLS values for cell lengths, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>ACCUM.4</u>	VOL array	Enter NCELLS values for cell volumes, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>ACCUM.5</u>	FA array	Enter (NCELLS + 1) values for cell flow areas, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>ACCUM.6</u>	FRIC array	Enter (NCELLS + 1) values for additive loss coefficients.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>ACCUM.7</u>	GRAV array	Enter (NCELLS + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1. (NOTE: The cell numbering convention for the accumulator component places the discharge at node (NCELLS + 1).)
<u>ACCUM.8</u>	HD array	Enter (NCELLS + 1) values for hydraulic diameters, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>ACCUM.9</u>	NFF array	Enter (NCELLS + 1) values for friction factor correlation options. Valid entries are: <ul style="list-style-type: none"> 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm (Use a negative entry for automatic form loss computation due to area change at node N.)
<u>ACCUM.10</u>	ALP array	Enter NCELLS values for void fraction.
<u>ACCUM.11</u>	VM array	Enter (NCELLS + 1) values for mixture velocity, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>ACCUM.12</u>	TL array	Enter NCELLS values for liquid temperature, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>ACCUM.13</u>	TV array	Enter NCELLS values for vapor temperature, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>ACCUM.14</u>	P array	Enter NCELLS values for pressure, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).

2.2.2 BREAK Component Data, read by subroutine RBREAK

BREAK.1 TYPE,NUM,ID,CTITLE FORMAT(A5,9X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	TYPE	Enter BREAK
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary component identification number (may be the same as NUM).
43-72	CTITLE	Enter alphanumeric identifier for this component.

BREAK.2 JUN1,IBROP,NBTB,ISAT FORMAT(4I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	JUN1	Enter the junction number where this break is located.
15-28	IBROP	Enter the break table option. Valid entries are: 0 = no break tables 1 = pressure table 2 = temperature and pressure tables 3 = temperature, pressure and void fraction tables
29-42	NBTB	Enter the number of points (pairs of values) in each of the break tables if IBROP > 0. Enter zero (0) if IBROP is zero.
43-56	ISAT	Enter a value to control the use of the break tables if IBROP > 0. Valid entries are: 0 = use TIN or the break table value for liquid and vapor temperatures 1 = use TIN or the break table value for liquid temperatures; use T _{sat} for vapor temperature

2 = use TIN or the break table for vapor temperature; use T_{sat} for liquid temperature

3 = use T_{sat} for both liquid and vapor temperatures

Note: If ISAT = 0, 1, or 2 and no temperature table is available (IBROP = 0 or 1), then TIN and T_{sat} will be used as specified by ISAT.

BREAK.3 DXIN,VOLIN,ALPIN,TIN,PIN FORMAT(5E14.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	DXIN	Enter the length of the break cell, in meters (if ITRAC = 0) or feet (if ITRAC = 1). (Usually DXIN should equal the length of the neighboring cell in the adjacent component.)
15-28	VOLIN	Enter the volume of the break cell, in m^3 (if ITRAC = 0) or ft^3 (if ITRAC = 1).
29-42	ALPIN	Enter the break cell void fraction.
43-56	TIN	Enter the mixture temperature at the break, in $^{\circ}K$ (if ITRAC = 0) or $^{\circ}F$ (if ITRAC = 1).
57-70	PIN	Enter the break pressure, in N/m^2 (if ITRAC = 0) or in psia (if ITRAC = 1).

BREAK.4 through BREAK.6 are entered using the LOAD format.

BREAK.4 (TDUM(I),PDUM(I),I=1,NBTB) FORMAT(5E14.0) Read only if IBROP > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14,...	TDUM(I)	Enter the time for element I of the pressure versus time table. Units on time must be seconds.

15-28 PDUM(I) Enter the pressure for element I of the pressure versus time table. Units on pressure must be N/m^2 if ITRAC = 0 or psia if ITRAC = 1.

BREAK.5 (TDUM(I),TPDUM(I),I=1,NBTB) FORMAT(5E14.0) Read only if IBROP > 1.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14,...	TDUM(I)	Enter the time for element I of the temperature versus time table. Units on time must be seconds.
15-28,...	TPDUM(I)	Enter the temperature for element I of the temperature versus time table. Units on temperature must be °K if ITRAC = 0 or °F if ITRAC = 1.

BREAK.6 (TDUM(I),VDUM(I),I=1,NBTB) FORMAT(5E14.0) Read only if IBROP = 3.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14,...	TDUM(I)	Enter the time for element I of the void fraction versus time table. Units on time must be in seconds.
15-28,...	VDUM(I)	Enter the void fraction for element I of the void fraction versus time table.

2.2.3 FILL Component Data, read by subroutine RFILL

FILL.1 TYPE,NUM,ID,CTITLE FORMAT(A4,10X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-4	TYPE	Enter FILL
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (may be the same as NUM).
43-72	CTITLE	Enter alphanumeric identifier for this component.

FILL.2 JUN1,IFTY,IFTR,NFTX FORMAT(4I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	JUN1	Enter the junction number where the fill is located.
15-28	IFTY	Enter the FILL type option. Valid entries are: 1 = constant velocity 2 = velocity vs time 3 = velocity vs pressure 4 = constant velocity until a trip occurs, then velocity vs time 5 = constant velocity until a trip occurs, then velocity vs pressure
29-42	IFTR	Enter a trip identification number corresponding to a value specified for ITID on trip specification card RDTRIP.1a. Enter zero (0) if this fill does not reference a trip.
43-56	NFTX	Enter the number of points (pairs of values) in fill tables. Enter zero if no fill tables are needed.

FILL.3 DXIN,VOLIN,ALPIN,VIN,TIN FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	DXIN	Enter the length of the FILL cell, in meters (if ITRAC = 0) or feet (if ITRAC = 1). (Generally the same as the neighboring cell in the adjacent component.)
15-28	VOLIN	Enter the fill cell volume, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
29-42	ALPIN	Enter the fill void fraction.
43-56	VIN	Enter the entrant mixture velocity, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
57-70	TIN	Enter the entrant mixture temperature, in °K (if ITRAC = 0) or °F (if ITRAC = 1).

FILL.4 PIN FORMAT(E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	PIN	Enter the fill pressure, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).

FILL.5 (VELDUM(I),TPDUM(I),I=1,NFTX) FORMAT(5E14.0) Read only if NFTX > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14,...	TPDUM(I)	If IFTY = 2 or 4, enter the time for element I in the velocity versus time table. Units on time must be seconds. If IFTY = 3 or 5, enter the pressure for element I in the velocity versus pressure table. Units on pressure are N/m ² (if ITRAC=0) or psia (if ITRAC=1).
15-28,...	VELDUM(I)	Enter the velocity for element I in the velocity versus time or pressure table. Units on velocity are m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).

2.2.4 PIPE Component Data, read by subroutine RPIPE

PIPE.1 TYPE,NUM,ID,CTITLE FORMAT(A4,10X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-4	TYPE	Enter PIPE
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (may be the same as NUM).
43-72	CTITLE	Enter an alphanumeric identifier for this component.

PIPE.2 NCELLS,NODES,JUN1,JUN2,MAT FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELLS	Enter the number of fluid cells in the pipe.
15-28	NODES	Enter the number of radial heat transfer nodes in the pipe wall. (Zero (0) implies that there is no wall heat transfer.)
29-42	JUN1	Enter the junction number for the connection to cell 1 of the pipe.
43-56	JUN2	Enter the junction number for the connection to cell NCELL of the pipe.
57-70	MAT	Enter material type index for the pipe wall. Valid entries are: 6 = SS 304 7 = SS 316 8 = SS 347 9 = Carbon steel A 508 10 = Incone1 718

PIPE.3 ICHF,IHYDRO

FORMAT(2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	ICHF	Enter the CHF calculation option. Valid entries are: 0 = no CHF calculation 1 = Zuber/Biasi for rod CHF (recommended for inclined or vertical tubes) 2 = Biasi (recommended for unheated and/or horizontal pipes) 3 = Bowring
15-28	IHYDRO	Enter a hydrodynamics option. Valid entries are: 0 = partially implicit 1 = fully implicit

PIPE.4 RADIN,TH,HOUTL,HOUTV,TOU TL

FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	RADIN	Enter the inner radius of the pipe wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
15-28	TH	Enter the pipe wall thickness, in meters (if ITRAC = 0) or in feet (if ITRAC = 1).
29-42	HOUTL	Enter the heat transfer coefficient between the outer boundary of the pipe wall and the liquid. Units are $W/m^2\text{ }^\circ\text{C}$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ\text{F}$ (if ITRAC = 1).
43-56	HOUTV	Enter the heat transfer coefficient between the outer boundary of the pipe wall and the vapor. Units are $W/m^2\text{ }^\circ\text{C}$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ\text{F}$ (if ITRAC = 1).
57-70	TOU TL	Enter the liquid temperature outside of the pipe wall, in $^\circ\text{K}$ (if ITRAC = 0) or $^\circ\text{F}$ (if ITRAC = 1).

PIPE.5 TOUTV

FORMAT(E14.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	TOUTV	Enter the vapor temperature outside of the pipe wall, in °K (if ITRAC = 0) or °F (if ITRAC = 1).

PIPE.6 through PIPE.18 are entered using the LOAD format and define the pipe data arrays.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>PIPE.6</u>	DX array	Enter NCELLS values for cell lengths, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>PIPE.7</u>	VOL array	Enter NCELLS values for cell volumes, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>PIPE.8</u>	FA array	Enter (NCELLS + 1) values for cell flow areas, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>PIPE.9</u>	FRIC array	Enter (NCELLS + 1) values for additive loss coefficients.
<u>PIPE.10</u>	GRAV array	Enter (NCELLS + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1.
<u>PIPE.11</u>	HD array	Enter (NCELLS + 1) values for hydraulic diameters, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>PIPE.12</u>	NFF array	Enter (NCELLS + 1) values for friction factor correlation options. Valid entries are: ±0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm (Use negative entry for automatic form loss computation in node N.)

<u>PIPE.13</u>	QPPP array	Enter NCELLS values for volumetric heat source in the pipe wall, in W/m^3 (if ITRAC = 0) or $Btu/hr-ft^3$ (if ITRAC = 1).
<u>PIPE.14</u>	ALP array	Enter NCELLS values for void fraction.
<u>PIPE.15</u>	VM array	Enter (NCELLS + 1) values for mixture velocity, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>PIPE.16</u>	TL array	Enter NCELLS values for liquid temperature, in $^{\circ}K$ (if ITRAC = 0) or $^{\circ}F$ (if ITRAC = 1).
<u>PIPE.17</u>	P array	Enter NCELLS values for pressure in N/m^2 (if ITRAC = 0) or psia (if ITRAC = 1).
<u>PIPE.18</u>	TW array	Enter NODES*NCELLS values for wall temperature in $^{\circ}K$ (if ITRAC = 0) or $^{\circ}F$ (if ITRAC = 1). Note: If NODES = 0, omit card PIPE.18.

2.2.5 Pressurizer (PRIZER) Component Data, read by subroutine RPRIZR

PRIZER.1 TYPE,NUM,ID,CTITLE FORMAT(A6,8Y,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-6	TYPE	Enter PRIZER
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (this number may be the same as NUM).
43-72	CTITLE	Enter an alphanumeric identifier for this component.

PRIZER.2 NCELLS, JUN2 FORMAT(2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELLS	Enter the number of fluid cells in the pressurizer.
15-28	JUN2	Enter the junction number for the connection to cell NCELLS of the pressurizer. (This is the pressurizer discharge.)

PRIZER.3 QHEAT,PSET,DPMAX,ZHTR FORMAT(4E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	QHEAT	Enter the total heater power, in watts (if ITRAC = 0) or Btu/hr (if ITRAC = 1).
15-28	PSET	Enter the pressure setpoint for heater/sprayer controller, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).
29-42	DPMAX	Enter pressure differential at which the heater/sprayer has maximum power. Units are N/m ² (if ITRAC = 0) or psid (if ITRAC = 1).
43-56	ZHTR	Enter water level for heater cutoff, in meters (if ITRAC = 0) or feet (if ITRAC = 1).

PRIZER.4 through PRIZER.15 are entered using the LOAD format and define the pressurizer array data.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>PRIZER.4</u>	DX array	Enter NCELLS values for cell lengths, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>PRIZER.5</u>	VOL array	Enter NCELLS values for cell volumes, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>PRIZER.6</u>	FA array	Enter (NCELLS + 1) values for cell flow areas, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>PRIZER.7</u>	FRIC array	Enter (NCELLS + 1) values for additive loss coefficients.
<u>PRIZER.8</u>	GRAV array	Enter (NCELLS + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to N + 1. (Note: The cell numbering convention for pressurizer components places the discharge at node (NCELLS + 1).)
<u>PRIZER.9</u>	HD array	Enter (NCELLS + 1) values for hydraulic diameters, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>PRIZER.10</u>	NFF array	Enter (NCELLS + 1) values for friction factor correlation options. Valid entries are: 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm (Use a negative entry for automatic form loss computation at node N.)
<u>PRIZER.11</u>	ALP array	Enter NCELLS values for void fraction.
<u>PRIZER.12</u>	VM array	Enter (NCELLS + 1) values for mixture velocity, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>PRIZER.13</u>	TL array	Enter NCELLS values for liquid temperature, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>PRIZER.14</u>	TV array	Enter NCELLS values for vapor temperature, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>PRIZER.15</u>	P array	Enter NCELLS values for pressure, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).

2.2.6 PUMP Component Data, read by subroutine RPUMP

PUMP.1 TYPE,NUM,ID,CTITLE FORMAT(A4,10X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	TYPE	Enter PUMP
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (may be the same as NUM).
43-72	CTITLE	Enter an alphanumeric identifier for this component.

PUMP.2 NCELLS,NODES,JUN1,JUN2,MAT FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELLS	Enter the number of fluid cells in the pump, (minimum of 2). (Note: The momentum source for the pump is always added at node 2, regardless of the number of cells in the pump.)
15-28	NODES	Enter the number of radial heat transfer nodes in the pump wall. Zero (0) implies that there is no wall heat transfer.
29-42	JUN1	Enter the junction number connecting to cell 1 of the pump.
43-56	JUN2	Enter the junction number connecting to cell NCELLS of the pump.
57-70	MAT	Enter the material index for the pump wall.

Valid entries are:

- 6 = SS 304
- 7 = SS 316
- 8 = SS 347
- 9 = Carbon steel A 508
- 10 = Inconel 718

PUMP.3 ICHF,IHYDRO,IPMPTY,IRP,IPM FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	ICHF	Enter the CHF calculation option. Valid entries are: 0 = no CHF calculation 1 = Zuber/Biasi for rod CHF (recommended for inclined or vertical tubes) 2 = Biasi CHF correlation (recommended for unheated and/or horizontal pipes) 3 = Bowring CHF correlation
15-28	IHYDRO	Enter the hydrodynamics option. Valid entries are: 0 = partially implicit 1 = fully implicit
29-42	IPMPTY	Enter pump type. Valid entries are: 1 = pump speed variation will be specified by input 2 = pump speed is calculated after the occurrence of a trip
43-56	IRP	Enter a reverse speed option. Valid entries are: 0 = no reverse rotation allowed 1 = reverse rotation allowed
57-70	IPM	Enter two-phase option. Valid entries are: 0 = use single phase only 1 = use two phase

PUMP.4 IPMPTR,NPMPTX FORMAT(2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	IPMPTR	Enter the index number of trip (corresponding to a value of ITID entered on RDTRIP.1a) that affects this pump.

Enter zero (0) if this pump does not reference a trip.

15-28 NPMPTX

Enter the number of points (pairs of values) in the pump speed table.

PUMP.5 RADIN,TH,HOUTL,HOUTV,TOUTL FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	RADIN	Enter the inner radius of the pump wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
15-28	TH	Enter the thickness of the pump wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
29-42	HOUTL	Enter the heat transfer coefficient between the outer boundary of the pump wall and the liquid. Units are $W/m^2 \text{ } ^\circ C$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ F$ (if ITRAC = 1).
43-56	HOUTV	Enter the heat transfer coefficient between the outer boundary of the pump wall and the vapor. Units are $W/m^2 \text{ } ^\circ C$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ F$ (if ITRAC = 1).
57-70	TOUTL	Enter the liquid temperature outside of the pump wall, in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 1).

PUMP.6 TOUTV FORMAT(E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	TOUTV	Enter the vapor temperature outside of the pump wall, in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 1).

PUMP.7 RHEAD,RTORK,RFLOW,RRHO,ROMEGA FORMAT(5E14.6)

Columns	Variable	Description
1-14	RHEAD	Enter the pump rated head, ($\Delta P/\rho$), in N-m/kg (if ITRAC = 0) or ft-lbf/lbm (if ITRAC = 1).
15-28	RTORK	Enter the pump rated torque, in N-m (if ITRAC = 0) or ft-lbf (if ITRAC = 1).
29-42	RFLOW	Enter the pump rated flow rate, in m ³ /sec (if ITRAC = 0) or ft ³ /sec (if ITRAC = 1).
43-56	RRHO	Enter the pump rated density, in kg/m ³ (if ITRAC = 0) or lbm/ft ³ (if ITRAC = 1).
57-70	ROMEGA	Enter the pump rated speed, in radians/sec.

PUMP.8 EFFMI,TFR1,TFR2,OMEGAN FORMAT(4E14.6)

Columns	Variable	Description
1-14	EFFMI	Enter the effective moment of inertia, in kg-m ² (if ITRAC = 0) or lbm-ft ² (if ITRAC = 1).
15-28	TFR1	Enter the constant torque due to friction, in N-m (if ITRAC = 0) or ft-lbf (if ITRAC = 1).
29-42	TFR2	Enter the bearing and windage torque constant, in N-m (if ITRAC = 0) or ft-lbf (if ITRAC = 1).
43-56	OMEGAN	Enter the initial pump speed, in radians/sec.

PUMP.9 OPTION FORMAT(I14)

Columns	Variable	Description
1-14	OPTION	Enter a pump curve option. Valid entries are: 0 = user-specified pump curves 1 = use the default pump curves (Semiscale type).

PUMP.10 through PUMP.31 specify the pump curves and are read by subroutine RDDIM. This input is omitted if OPTION = 1.

PUMP.10 (NDATA(I),I=1,5) FORMAT(5I14) Read only if OPTION = 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NDATA(1)	Enter the number of points (pairs of values) in curve HSP1 of the single-phase head curves.
15-28	NDATA(2)	Enter the number of points in curve HSP2 of the single-phase head curves.
29-42	NDATA(3)	Enter the number of points in curve HSP3 of the single-phase head curves.
43-56	NDATA(4)	Enter the number of points in curve HSP4 of the single-phase head curves.
57-70	NDATA(5)	Enter the number of points in curve HTP1 of the fully degraded head curves.

PUMP.11 (NDATA(I),I=6,10) FORMAT(5I14) Read only if OPTION = 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NDATA(6)	Enter the number of points in curve HTP2 of the fully degraded head curves.
15-28	NDATA(7)	Enter the number of points in curve HTP3 of the fully degraded head curves.
29-42	NDATA(8)	Enter the number of points in curve HTP4 of the fully degraded head curves.
43-56	NDATA(9)	Enter the number of points in curve TSP1 of the single-phase torque curves.
57-70	NDATA(10)	Enter the number of points in curve TSP2 of the single-phase torque curves.

PUMP.12 (NDATA(I),I=11,15) FORMAT(5I14) Read only if OPTION = 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NDATA(11)	Enter the number of points in curve TSP3 of the single-phase torque curves.

15-28	NDATA(12)	Enter the number of points in curve TSP4 of the single-phase torque curves.
29-42	NDATA(13)	Enter the number of points in curve TTP1 of the fully degraded head curves.
43-56	NDATA(14)	Enter the number of points in curve TTP2 of the fully degraded head curves.
57-70	NDATA(15)	Enter the number of points in curve TTP3 of the fully degraded head curves.

PUMP.13 NDATA(16),NHDM,NTDM FORMAT(3I14) Read only if OPTION = 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NDATA(16)	Enter the number of points in curve TTP4 of the fully degraded head curves.
15-28	NHDM	Enter the number of points in the head degradation multiplier curve.
29-42	NTDM	Enter the number of points in the torque degradation multiplier curve.

PUMP.14 through PUMP.31 define the pump curves and are entered via the LOAD format. Refer to Section 3.5.5 for examples of these curves. These cards are entered only if OPTION = 0.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>PUMP.14</u>	HSP1 Curve	Enter NDATA(1) points, 2 values per point.
<u>PUMP.15</u>	HSP2 Curve	Enter NDATA(2) points, 2 values per point.
<u>PUMP.16</u>	HSP3 Curve	Enter NDATA(3) points, 2 values per point.
<u>PUMP.17</u>	HSP4 Curve	Enter NDATA(4) points, 2 values per point.
<u>PUMP.18</u>	HTP1 Curve	Enter NDATA(5) points, 2 values per point.
<u>PUMP.19</u>	HTP2 Curve	Enter NDATA(6) points, 2 values per point.
<u>PUMP.20</u>	HTP3 Curve	Enter NDATA(7) points, 2 values per point.
<u>PUMP.21</u>	HTP4 Curve	Enter NDATA(8) points, 2 values per point.
<u>PUMP.22</u>	TSP1 Curve	Enter NDATA(9) points, 2 values per point.
<u>PUMP.23</u>	TSP2 Curve	Enter NDATA(10) points, 2 values per point.
<u>PUMP.24</u>	TSP3 Curve	Enter NDATA(11) points, 2 values per point.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>PUMP.25</u>	TSP4 Curve	Enter NDATA(12) points, 2 values per point.
<u>PUMP.26</u>	TTP1 Curve	Enter NDATA(13) points, 2 values per point.
<u>PUMP.27</u>	TTP2 Curve	Enter NDATA(14) points, 2 values per point.
<u>PUMP.28</u>	TTP3 Curve	Enter NDATA(15) points, 2 values per point.
<u>PUMP.29</u>	TTP4 Curve	Enter NDATA(16) points, 2 values per point.
<u>PUMP.30</u>	HDM Curve	Enter NHDM points, 2 values per point.
<u>PUMP.31</u>	TDM Curve	Enter NTDM points, 2 values per point.

PUMP.32 through PUMP.45 are entered via the LOAD format and define the pump array data.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>PUMP.32</u>	SPTBL array	Enter NPMPX pairs of values for table of pump speed versus time since trip. (time-speed pairs).
<u>PUMP.33</u>	DX array	Enter NCELLS values for cell lengths, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>PUMP.34</u>	VOL array	Enter NCELLS values for cell volumes, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>PUMP.35</u>	FA array	Enter (NCELLS + 1) values for cell flow areas, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>PUMP.36</u>	FRIC array	Enter (NCELLS + 1) values for additive loss coefficients.
<u>PUMP.37</u>	GRAV array	Enter (NCELLS + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1.
<u>PUMP.38</u>	HD array	Enter (NCELLS + 1) values for hydraulic diameters, in meters (if ITRAC = 0) or feet (if ITRAC = 1).

<u>PUMP.39</u>	NFF array	<p>Enter (NCELLS + 1) values for friction factor correlation options. Valid entries are:</p> <ul style="list-style-type: none"> 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm <p>(Use negative entry for automatic form loss computation.)</p>
<u>PUMP.40</u>	QPPP array	<p>Enter NCELLS values for volumetric heat source in the pump wall. Units are W/m^3 (if ITRAC = 0) or $Btu/hr-ft^3$ (if ITRAC = 1).</p>
<u>PUMP.41</u>	ALP array	<p>Enter NCELLS values for void fraction.</p>
<u>PUMP.42</u>	VM array	<p>Enter (NCELLS + 1) values for mixture velocity, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).</p>
<u>PUMP.43</u>	TL array	<p>Enter NCELLS values for liquid temperature, in °K (if ITRAC = 0) or °F (if ITRAC = 1).</p>
<u>PUMP.44</u>	P array	<p>Enter NCELLS values for pressure, in N/m^2 (if ITRAC = 0) or psia (if ITRAC = 1).</p>
<u>PUMP.45</u>	TW array	<p>Enter NODES*NCELLS values for wall temperatures in °K (if ITRAC = 0) or °F (if ITRAC = 1). Omit this data if NODES = 0.</p>

2.2.7 Steam Generator (STGEN) Component Data, read by subroutine RSTGEN

Only the tube region of the steam generator is modeled with the one-dimensional steam generator component. The flow rate specified into the secondary side of the steam generator should correspond to the total recirculation flow rate and not just the feedwater flow rate. The complete steam generator can be modeled using the vessel component.

STGEN.1 TYPE,NUM,ID,CTITLE FORMAT(A5,9X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	TYPE	Enter STGEN
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (may be the same as NUM).
43-72	CTITLE	Enter an alphanumeric identifier for this component.

STGEN.2 NCELL1,NODES,JUN11,JUN12,MAT FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELL1	Enter the number of fluid cells on the primary side of the steam generator.
15-28	NODES	Enter the number of radial heat transfer nodes in the steam generator wall (minimum of one).
29-42	JUN11	Enter the junction number adjacent to cell 1 on the primary side.
43-56	JUN12	Enter the junction number adjacent to cell NCELL1 on the primary side.

57-70 MAT Enter the material index for the steam generator tubes. Valid entries are:

- 6 = SS 304
- 7 = SS 316
- 8 = SS 347
- 9 = Carbon steel A 508
- 10 = Inconel 718

STGEN.3 KIND, IHYDRO, ICHF1, ICHF2 FORMAT(4I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	KIND	Enter the steam generator type. Valid entries are: 1 = U-tube 2 = once-through
15-28	IHYDRO	Enter the hydrodynamics option. Valid entries are: 0 = partially implicit 1 = fully implicit
29-42	ICHF1	Enter the primary side CHF option. Valid entries are: 0 = no CHF calculation 1 = CHF calculation
43-56	ICHF2	Enter the secondary side CHF option. Valid entries are: 0 = no CHF calculation 1 = CHF calculation

Note: Boiling heat transfer is not calculated in the steam generator unless CHF calculations are specified with ICHF1 and ICHF2. For most calculations, it is recommended that ICHF1 = 0 and ICHF2 = 1.

STGEN.4 RADIN,TH FORMAT(2E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	RADIN	Enter the inner radius of the steam generator wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
15-28	TH	Enter the tube wall thickness, in meters (if ITRAC = 0) or feet (if ITRAC = 1).

STGEN.5 NCELL2,JUN21,JUN22 FORMAT(3I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELL2	Enter the number of cells on the secondary side of the steam generator.
15-28	JUN21	Enter the junction number adjacent to cell 1 on the secondary side of the steam generator.
29-42	JUN22	Enter the junction number adjacent to cell NCELL2 on the secondary side of the steam generator.

STGEN.6 through STGEN.32 are entered via the LOAD format and define the array variables for both the primary and secondary sides of the steam generator.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>STGEN.6</u>	DX1 array	Enter NCELL1 values for cell lengths for the primary side, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>STGEN.7</u>	VOL1 array	Enter NCELL1 values for cell volumes for the primary side, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>STGEN.8</u>	FA1 array	Enter (NCELL1 + 1) values for cell flow areas for the primary side, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>STGEN.9</u>	FRIC1 array	Enter (NCELL1 +1) values for additive loss coefficients for the primary side.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>STGEN.10</u>	GRAV1 array	Enter (NCELL1 + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1 for the primary side.
<u>STGEN.11</u>	HD1 array	Enter (NCELL1 + 1) values for hydraulic diameters for the primary side, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>STGEN.12</u>	NFF1 array	Enter (NCELL1 + 1) values for friction factor correlations options for the primary side. Valid entries are: <ul style="list-style-type: none"> 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm (Use negative entry for automatic form loss computation at node N.)
<u>STGEN.13</u>	WA1 array	Enter NCELL1 values for wall heat transfer area for the primary side, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>STGEN.14</u>	ALP1 array	Enter NCELL1 values for void fraction for the primary side.
<u>STGEN.15</u>	VM1 array	Enter (NCELL1 + 1) values for mixture velocity for the primary side, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>STGEN.16</u>	TL1 array	Enter NCELL1 values for liquid temperature for the primary side, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>STGEN.17</u>	TV1 array	Enter NCELL1 values for vapor temperature for the primary side, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>STGEN.18</u>	P1 array	Enter NCELL1 values for pressure for the primary side, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>STGEN.19</u>	DX2 array	Enter NCELL2 values for cell lengths for the secondary side, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>STGEN.20</u>	VOL2 array	Enter NCELL2 values for cell volumes for the secondary side, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>STGEN.21</u>	FA2 array	Enter (NCELL2 + 1) values for cell flow areas for the secondary side in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>STGEN.22</u>	FRIC2 array	Enter (NCELL2 + 1) values for additive loss coefficients for the secondary side.
<u>STGEN.23</u>	GRAV2 array	Enter (NCELL2 + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1 for the secondary side.
<u>STGEN.24</u>	HD2 array	Enter (NCELL2 + 1) values for hydraulic diameters for the secondary side, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>STGEN.25</u>	NFF2 array	Enter (NCELL2 + 1) values for friction factor correlation options for the secondary side. Valid entries are: <ul style="list-style-type: none"> 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm (Use negative entry for automatic form loss computation at node N.)

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>STGEN.26</u>	WA2 array	Enter NCELL2 values for wall heat transfer areas on the secondary side, in m^2 (if ITRAC = 0) or ft^2 (if ITRAC = 1). (See Section 3.5.6 for guidance on specifying wall area on secondary side, particularly for u-tube steam generators.)
<u>STGEN.27</u>	ALP2 array	Enter NCELL2 values for void fraction for the secondary side.
<u>STGEN.28</u>	VM2 array	Enter (NCELL2 + 1) values for mixture velocity for the secondary side, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>STGEN.29</u>	TL2 array	Enter NCELL2 values for liquid temperature for the secondary side, in $^{\circ}K$ (if ITRAC = 0) or $^{\circ}F$ (if ITRAC = 1).
<u>STGEN.30</u>	TV2 array	Enter NCELL2 values for vapor temperature for the secondary side, in $^{\circ}K$ (if ITRAC = 0) or $^{\circ}F$ (if ITRAC = 1).
<u>STGEN.31</u>	P2 array	Enter NCELL2 values for pressure for the secondary side, in N/m^2 (if ITRAC = 0) or psia (if ITRAC = 1).
<u>STGEN.32</u>	TW array	Enter NODES*NCELL1 values for initial tube wall temperatures in $^{\circ}K$ (if ITRAC = 0) or $^{\circ}F$ (if ITRAC = 1).

2.2.8 TEE Component Data, read by subroutine RTEE

TEE.1 TYPE,NUM, ID,CTITLE FORMAT(A3,11X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-3	TYPE	Enter TEE
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (may be the same as NUM).
43-72	CTITLE	Enter an alphanumeric identifier for this component.

TEE.2 JCELL, NODES, MATID, COST, ICHF FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	JCELL	Enter the TEE junction cell number (cannot be 1 or NCELL1).
15-28	NODES	Enter the number of radial heat transfer nodes in the wall. Zero (0) implies no wall heat transfer.
29-42	MATID	Enter the material type index for the wall. Valid entries are: 6 = SS 304 7 = SS 316 8 = SS 347 9 = Carbon steel A 508 10 = Inconel 718
43-56	COST	Enter the cosine of the angle between the primary tube and the secondary tube. The angle is measured between a line from cell 1 to JCELL on the primary tube and a line from JCELL to NCELL2 on the secondary tube. (See Figure 3.45.)
57-70	ICHF	Enter the CHF calculation option. Valid entries are:

- 0 = no CHF calculation
- 1 = Zuber/Biasi for rod CHF (recommended for inclined or vertical tubes).
- 2 = Biasi (recommended for unheated and/or horizontal tubes.)
- 3 = Bowring

TEE.3 IHYD1,NCELL1,JUN1,JUN2 FORMAT(4I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	IHYD1	Enter the hydrodynamics option in the primary tube. Valid entries are: 0 = partially implicit 1 = fully implicit
15-28	NCELL1	Enter the number of fluid cells in the primary tube (minimum of two).
29-42	JUN1	Enter the number of the junction adjacent to cell 1.
43-56	JUN2	Enter the number of the junction adjacent to cell NCELL1.

TEE.4 RADIN1,TH1,HOUTL1,HOUTV1,TOU1L1 FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	RADIN1	Enter the radius of the primary tube wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
15-28	TH1	Enter the thickness of the primary tube wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
29-42	HOUTL1	Enter the heat transfer coefficient between the outer surface of the primary tube wall and the surrounding liquid, in W/m ² °C (if ITRAC = 0) or Btu/ft ² -hr-°F (if ITRAC = 1).

43-56 HOUTV1 Enter the heat transfer coefficient between the outer surface of the primary tube wall and the surrounding vapor, in $W/m^2 \text{ } ^\circ C$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ F$ (if ITRAC = 1).

57-70 TOUTL1 Enter the temperature of the liquid outside of the primary tube wall, in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 0).

TEE.5 TOUTV1 FORMAT(E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	TOUTV1	Enter the temperature of the vapor outside of the primary tube wall, in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 1).

TEE.6 IHYD2,NCELL2,JUN3,ISEP FORMAT(4I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	IHYD2	Enter the hydrodynamics option for the side tube. Valid entries are: 0 = partially implicit 1 = fully implicit
15-28	NCELL2	Enter the number of fluid cells in side tube.
29-42	JUN3	Enter the junction number adjacent to the free end of the side tube (cell NCELL2).
43-56	ISEP	Enter flag for phase separation calculation. Valid entries are: 0 = no phase separation 1 = phase separation above void fraction of ALSEP (entered on TEE.8)

TEE.7 RADIN2,TH2,HOUTL2,HOUTV2,TOUTL2 FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	RADIN2	Enter the inner radius of the side wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
15-28	TH2	Enter the thickness of the side tube wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
29-42	HOUTL2	Enter the heat transfer coefficient between the outer surface of the side tube wall and the surrounding liquid, in $W/m^2 \text{ } ^\circ C$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ F$ (if ITRAC = 1).
43-56	HOUTV2	Enter the heat transfer coefficient between the outer surface of the side tube wall and the surrounding vapor in $W/m^2 \text{ } ^\circ C$ (if ITRAC = 0) or $Btu/ft^2\text{-hr-}^\circ F$ (if ITRAC = 1).
57-70	TOUTL2	Enter the temperature of the liquid surrounding the side tube, in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 1).

TEE.8 TOUTV2,ALSEP FORMAT(2E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	TOUTV2	Enter the temperature of the vapor surrounding the side tube, in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 1).
15-28	ALSEP	Enter void fraction for phase separation model. (Used only if ISEP = 1).

TEE. 9 through TEE.33 are entered via the LOAD format and define the arrays for the primary and side tubes of the tee.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>TEE.9</u>	DX array	Enter NCELL1 values for cell lengths in the primary tube, in meters (if ITRAC = 0) or feet (if ITRAC = 1).

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>TEE.10</u>	DX array	Enter NCELL2 values for cell lengths in the side tube, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>TEE.11</u>	VOL array	Enter NCELL1 values for cell volumes in the primary tube, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>TEE.12</u>	VOL array	Enter NCELL2 values for cell volumes in the side tube, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>TEE.13</u>	FA array	Enter (NCELL1 + 1) values for cell flow areas in the primary tube, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>TEE.14</u>	FA array	Enter (NCELL2 + 1) values for cell flow areas in the side tube, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>TEE.15</u>	FRIC array	Enter (NCELL1 + 1) values for additive loss coefficients in the primary tube.
<u>TEE.16</u>	FRIC array	Enter (NCELL2 + 1) values for additive loss coefficients in the side tube.
<u>TEE.17</u>	GRAV array	Enter (NCELL1 + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1 in the primary tube.
<u>TEE.18</u>	GRAV array	Enter (NCELL2 + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1 in the side tube.
<u>TEE.19</u>	HD array	Enter (NCELL1 + 1) values for hydraulic diameters in the primary tube, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>TEE.20</u>	HD array	Enter (NCELL2 + 1) values for hydraulic diameters in the side tube, in meters (if ITRAC = 0) or feet (if ITRAC = 1).

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>TEE.21</u>	NFF array	<p>Enter (NCELL1 + 1) values for friction factor correlation options in the primary tube. Valid entries are:</p> <ul style="list-style-type: none"> 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm <p>(Use negative entry for automatic form loss computation at node N.)</p>
<u>TEE.22</u>	NFF array	<p>Enter (NCELL2 + 1) values for friction factor correlation options in the side tube. Valid entries are:</p> <ul style="list-style-type: none"> 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm <p>(Use negative entry for automatic form loss computation at node N.)</p>
<u>TEE.23</u>	QPPP array	<p>Enter NCELL1 values for volumetric heat source in the primary tube wall. Units are W/m^3 (if ITRAC = 0) or $Btu/hr-ft^3$ (if ITRAC = 1).</p>
<u>TEE.24</u>	QPPP array	<p>Enter NCELL2 values for volumetric heat source in the side tube wall. Units are W/m^3 (if ITRAC = 0) or $Btu/hr-ft^3$ (if ITRAC = 1).</p>
<u>TEE.25</u>	ALP array	<p>Enter NCELL1 values for void fraction in the primary tube.</p>
<u>TEE.26</u>	ALP array	<p>Enter NCELL2 values for void fraction in the side tube.</p>

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>TEE.27</u>	VM array	Enter (NCELL1 + 1) values for mixture velocity in the primary tube, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>TEE.28</u>	VM array	Enter (NCELL2 + 1) values for mixture velocity in the side tube, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>TEE.29</u>	TL array	Enter NCELL1 values for liquid temperature in the primary tube, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>TEE.30</u>	TL array	Enter NCELL2 values for liquid temperature in the side tube, in °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>TEE.31</u>	P array	Enter NCELL1 values for pressure in the primary tube, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).
<u>TEE.32</u>	P array	Enter NCELL2 values for pressure in the side tube, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).
<u>TEE.33</u>	TW array	Enter NODES*NCELL1 values for tube wall temperature in °K (if ITRAC = 0) or °F (if ITRAC = 1).

2.2.9 VALVE Component Data, read by subroutine RVLVE

VALVE.1 TYPE,NUM,ID,CTITLE FORMAT(A5,9X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	TYPE	Enter VALVE
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (may be the same as NUM).
43-72	CTITLE	Enter an alphanumeric identifier for this component.

VALVE.2 NCELLS,NODES,JUN1,JUN2,MAT

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	NCELLS	Enter the number of fluid cells in the valve (minimum of two).
15-28	NODES	Enter the number of radial heat transfer nodes in the valve wall. Zero (0) implies no wall heat transfer.
29-42	JUN1	Enter the junction number adjacent to cell 1.
43-56	JUN2	Enter the junction number adjacent to cell NCELLS.
57-70	MAT	Enter the valve wall material index. Valid entries are: 6 = SS 304 7 = SS 316 8 = SS 347 9 = Carbon steel A 508 10 = Inconel 718

VALVE.3 ICHF, IHYDRO

FORMAT(2I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	ICHF	Enter the CHF calculation option. valid entries are: 0 = no CHF calculation 1 = Zuber/Biasi CHF (recommended for inclined or vertical tubes) 2 = Biasi (recommended for unheated and/or horizontal tubes) 3 = Bowring dryout correlation
15-28	IHYDRO	Enter the hydrodynamics option. Valid entries are: 0 = partially implicit 1 = fully implicit

VALVE.4 RADIN, TH, HOUTL, HOUTV, TOUTL

FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	RADIN	Enter the inner radius of the valve wall, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
15-28	TH	Enter the valve wall thickness, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
29-42	HOUTL	Enter the heat transfer coefficient between the outer surface of the valve wall and the surrounding liquid, in $W/m^2 \cdot ^\circ C$ (if ITRAC = 0) or $Btu/hr-ft^2 \cdot ^\circ F$ (if ITRAC = 1).
43-56	HOUTV	Enter the heat transfer coefficient between the outer surface of the valve wall and the surrounding vapor, in $W/m^2 \cdot ^\circ C$ (if ITRAC = 0) or $Btu/hr-ft^2 \cdot ^\circ F$ (if ITRAC = 1).
57-70	TOUTL	Enter the temperature of the surrounding liquid in $^\circ K$ (if ITRAC = 0) or $^\circ F$ (if ITRAC = 1).

VALVE.5 TOUTV FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	TOUTV	Enter the temperature of the surrounding vapor, in °K (if ITRAC = 0) or °F (if ITRAC = 1).

VALVE.6 IVTY,IVTR,NVTX,IVPG,IVPS FORMAT(5I14)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	IVTY	Enter the valve type. Valid entries are: 1 = valve normally open; occurrence of a trip closes it instantaneously 2 = valve normally closed; occurrence of a trip opens it instantaneously 3 = valve normally open; trip begins to close it according to a time-dependent table 4 = valve normally closed; trip begins to open it according to a time-dependent table 5 = valve is a check valve controlled by a static pressure gradient
15-28	IVTR	Enter the index number of the trip number controlling this valve, (corresponding to a value of ITID on card RDTRIP.1a).
29-42	NVTX	Enter the number of entries in the valve open/closed vs time table. (Used only if IVTY = 3 or 4.)

43-56 IVPG Enter the valve pressure gradient option (used only if IVTY = 5). Valid options are:
 = 1 if $\Delta P = P(\text{IVPS} - 1) - P(\text{IVPS})$
 = 2 if $\Delta P = P(\text{IVPS}) - P(\text{IVPS} - 1)$

Note:

If $\Delta P + \text{PVC} > 0$, the valve is open.
 If $\Delta P + \text{PVC} < 0$, the valve is closed.
 (PVC is defined on card VALVE.7.)

57-70 IVPS Enter the number of the cell the valve is located in; $2 < \text{IVPS} < \text{NCELLS}$.

VALVE.7 AVLVE,HVLVE,PVC FORMAT(3E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	AVLVE	Enter the area of the valve when completely open, in m^2 (if ITRAC = 0) or ft^2 (if ITRAC = 1).
15-28	HVLVE	Enter the hydraulic diameter of the valve when completely open, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
29-42	PVC	Enter the valve set point, in N/m^2 (if ITRAC = 0) or psi (if ITRAC = 1). If IVPG = 1, the valve will open when the pressure drop across the valve reaches $\Delta P + \text{PVC}$. If IVPG = 2, the valve will close when the pressure drop across the valve reaches $\Delta P + \text{PVC}$. Enter zero if IVTY \neq 5.

VALVE.8 through VALVE.21 are entered via the LOAD format and define the valve array cards.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>VALVE.8</u>	DX array	Enter NCELLS values for cell lengths, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>VALVE.9</u>	VOL array	Enter NCELLS values for cell volumes, in m ³ (if ITRAC = 0) or ft ³ (if ITRAC = 1).
<u>VALVE.10</u>	FA array	Enter (NCELLS + 1) values for cell edge flow areas, in m ² (if ITRAC = 0) or ft ² (if ITRAC = 1).
<u>VALVE.11</u>	FRIC array	Enter (NCELLS + 1) values for additive loss coefficients.
<u>VALVE.12</u>	GRAV array	Enter (NCELLS + 1) values for the cosine of the angle between a vertical vector and the vector from cell N to cell N + 1.
<u>VALVE.13</u>	HD array	Enter (NCELLS + 1) values for hydraulic diameters, in meters (if ITRAC = 0) or feet (if ITRAC = 1).
<u>VALVE.14</u>	NFF array	Enter (NCELLS + 1) values for friction factor correlation options. Valid entries are: 0 = constant friction factor, user input ±1 = homogeneous flow ±2 = Armand ±3 = CISE ±4 = annular flow ±5 = Chisolm (Use negative entry for automatic form loss computation at node N.)
<u>VALVE.15</u>	QPPP array	Enter NCELLS values for volumetric heat source in the valve wall in W/m ³ (if ITRAC = 0) or Btu/hr-ft ³ (if ITRAC = 1).
<u>VALVE.16</u>	VLTB array	Enter the valve open/closed vs time-since-trip array in pairs of (time and fraction open). Enter NVTX points. (Omit this card if NVTX = 0).
<u>VALVE.17</u>	ALP array	Enter NCELLS values for void fraction.

<u>Cards</u>	<u>Variable</u>	<u>Description</u>
<u>VALVE.18</u>	VM array	Enter (NCELLS + 1) values for mixture velocity, in m/sec (if ITRAC = 0) or ft/sec (if ITRAC = 1).
<u>VALVE.19</u>	TL array	Enter NCELLS values for liquid temperature, °K (if ITRAC = 0) or °F (if ITRAC = 1).
<u>VALVE.20</u>	P array	Enter NCELLS values for pressure, in N/m ² (if ITRAC = 0) or psia (if ITRAC = 1).
<u>VALVE 21.</u>	TW array	Enter NODES*NCELLS values for the value wall temperature in °K (if ITRAC = 0) or °F (if ITRAC = 1). If NODES = 0, do not enter the TW array data.

2.3 VESSEL (3-D) COMPONENT DATA

The vessel input data is organized in groups. Each group fulfills some general input requirement such as channel geometry, channel connections or boundary conditions. Any number of vessels or vertical components may be modeled with a single vessel component.

COBRA.1 TYPE,NUM,ID,CTITLE FORMAT(A6,8X,2I14,3A10)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-6	TYPE	Enter VESSEL
15-28	NUM	Enter a unique component number.
29-42	ID	Enter an arbitrary identification number (this number may be the same as NUM).
43-72	CTITLE	Enter alphanumeric identifier for this component.

VESSEL Calculation Control Data, read by subroutine COBRAI

COBRA.2 INIT,TEXT FORMAT(2I5,17A4)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
6-10	INIT	Enter the vessel initialization option. Valid entries are: 1 = initial start 4 = fill vessel arrays with data obtained from a restart file
11-80	TEXT	Enter alphanumeric information to identify the simulation. (This can be different from the title entered on INPUT.2.)

VESSEL Component Restart Data, read by subroutine RESTRT

COBRA.3 IDUM FORMAT(I5) Read only if INIT = 4.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	IDUM	Enter zero (0) to restart a previous calculation.

Card groups VESSEL 1 through VESSEL 14 are not read on a restart and should not be included in the restart input deck. Any card group may be eliminated from the initial input deck if data in that card group is not required for the problem to be set up.

VESSEL Group 1: Calculation Variables and Initial Conditions, read by subroutine SETIN

VESSEL1.1. NGROUP,NDX FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter one (1).
6-10	NDX	Enter the total number of vertical levels in the vessel (the sum of all levels in all sections).

VESSEL.1.2 Z FORMAT(F5.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	Z	Enter the total vertical height, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). (NOTE: ICOBRA is specified on INPUT.1; see Section 2.1.)

VESSEL1.3 PREF,HIN,GIN,AFLUX FORMAT(4F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	PREF	Enter the initial vessel operating pressure, in psi (if ICOBRA = 0) or N/m^2 (if ICOBRA = 1).
11-20	HIN	Enter the enthalpy for fluid initialization, in Btu/lbm (if ICOBRA = 0) or J/kg (if ICOBRA = 1).
21-30	GIN	Enter the initial mass flux, in lbm/ft^2 -sec (if ICOBRA = 0) or kg/m^2 -sec (if ICOBRA = 1).
31-40	AFLUX	Enter the average linear heat rate per active rod in kW/ft (if ICOBRA = 0) or kW/m (if ICOBRA = 1).

VESSEL Group 2: Channel Description, read by subroutine SETIN

VESSEL2.1 NGROUP,NCHANL,NCHANR FORMAT(3I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter two (2).
6-10	NCHANL	Enter the number of channels in the problem.
11-15	NCHANR	Enter the number of channels with thermal connections to rods or unheated conductors.

Cards VESSEL2.2 and VESSEL2.3 are read in pairs, NCHANL times.

VESSEL2.2 I,AN(I),PW(I),ABOT(I),ATOP(I),NAMGAP FORMAT(I5,4F5.0,I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter the channel identification number. (Note: Channel index numbers must be unique, but they do not have to be sequential. Skipping numbers is permitted, so long as exactly NCHANL channels are identified.)
6-10	AN(I)	Enter the nominal channel area, in in ² (if ICOBRA = 0) or m ² (if ICOBRA = 1). (Do not enter zero.)
11-15	PW(I)	Enter the channel wetted perimeter, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). (Do not enter zero.)
16-20	ABOT(I)	Enter the area of the bottom of the channel for use in the momentum equation. Units are in. ² (if ICOBRA = 0) or m ² (if ICOBRA = 1). If ABOT(I) is entered as zero (0.0), it is set to AN(I).
21-25	ATOP(I)	Enter the area on the top of the channel for use in the momentum equation. Units are in. ² (if ICOBRA = 0) or m ² (if ICOBRA = 1). If ATOP(I) is entered as zero (0), it is set to AN(I).

26-30 NAMGAP Enter the number of gaps for which the vertical velocity of channel I convects transverse momentum between sections. (Refer to Section 3.1.1 for further discussion.)

VESSEL2.3 (INODE(I,N),KGAPB(I,N),KGAPA(I,N),N=1,NAMGAP) FORMAT(15I5)

Read only if NAMGAP > 0 for channel I.

NOTE: Omit this card if NAMGAP is zero (0) for channel I.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5, 16-20, 31-23, 46-50, 61-65	INODE(I,N)	Enter the index number of the node where the vertical velocity of channel I convects transverse momentum across a section boundary. (Note: INODE will be either at the bottom of the channel (INODE(I,N)=1), or the top of the channel, (INODE(I,N)=NONODE+1), where NONODE is the number of axial levels in the section containing channel I. INODE is defined in the section where the vertical momentum equation is solved (see Section 3.1.1.)
6-10, 21-25, 36-40, 51-55 66-70	KGAPB(I,N)	Enter the index number of the gap below the section boundary. Enter zero if there is no gap below the section boundary. (Note: If KGAPB is not zero, the positive velocity of channel I at INODE(I,N) convects transverse momentum out of KGAPB into KGAPA. The negative velocity of channel I at INODE(I,N) convects transverse momentum from KGAPA into KGAPB; but if KGAPB is zero, this momentum is dissipated.)
11-15, 26-30,	KGAPA(I,N)	Enter the index number of the gap above the section boundary.

41-45,
56-60
71-75

Enter zero if there is no gap above the section boundary.

Note: If KGAPA is not zero, the positive velocity of channel I at INODE(I,N) convects transverse momentum from KGAPB (if KGAPB \neq 0) into KGAPA. (If KGAPA is zero, this momentum is dissipated.) The negative velocity of channel I at INODE(I,N) convects transverse momentum from KGAPA to KGAPB, (if KGAPB \neq 0; if KGAPB is zero, this momentum is dissipated.)

Card VESSEL2.4 is read NCH/NR times.

VESSEL2.4 I,(LR(I,L),L=1,6),(LS(I,L),L=1,8) FORMAT(15I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter the identification number of a channel with thermal connections to rods or unheated conductors.
6-10	LR(I,L)	Enter the index number of the rods connected to channel I. Enter the negative of the rod index number if the channel is on the inside surface.
36-40	LS(I,L)	Enter the index numbers of the unheated conductors connected to channel I. Enter the negative of the index number if the channel is on the inside surface.

Note: Within a given section rods and unheated conductors can connect to only one channel on the outside surface. Those with tube or wall geometry types can also connect to one channel on the inside surface. Rods can cross section boundaries, and connect to channels in different sections, but unheated conductors are defined only within a single section.

VESSEL Group 3: Transverse Channel Connection (Gap) Data, read by subroutine SETIN

This group is omitted if there are no transverse connections between channels.

VESSEL 3.1 NGROUP,NK FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter three (3).
6-10	NK	Enter the number of transverse connections (gaps).

Cards VESSEL3.2 and VESSEL3.3 are read in pairs NK times.

VESSEL3.2 K,IK(K),JK(K),GAPN(K),LENGTH(K),WKR(K),FWALL(K),IGAPB(K),
IGAPA(K),FACTOR(K),(IGAP(K,N),JGAP(K,N),N=1,3)
FORMAT(3I5,4F5.0,2I5,F5.0,6I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	K	Enter the gap identification number. (Note: Gap numbers must be unique but they do not have to be sequential. NK gaps must be input.)
6-10	IK(K)	Enter the identification number of the lower-numbered channel of the pair that connect through gap K.
11-15	JK(K)	Enter the identification number of the higher-numbered channel of the pair that connect through gap K.
16-20	GAPN(K)	Enter the nominal gap width, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-25	LENGTH(K)	Enter the distance between the center of channel IK(K) and the center of channel JK(K), in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
26-30	WKR(K)	Enter the loss coefficient (velocity head) for gap K.

31-35	FWALL(K)	<p>Enter the wall friction factor for the gap. Valid entries are:</p> <p style="padding-left: 40px;">0.0 = no walls 0.5 = one wall 1.0 = two walls</p>
36-40	IGAPB(K)	<p>Enter the index number of the gap below gap K. Enter zero if there is no gap below gap K. (The velocity of IGAPB(K) convects vertical momentum at node 1 into (or out of) channel IK(K) out of (or into) JK(K).)</p>
41-45	IGAPA(K)	<p>Enter the index number of the gap above gap K. (The velocity of IGAPA(K) convects vertical momentum at the top node of the section into (or out of) channel IK(K) out of (or into) JK(K).)</p> <p>Enter zero if there is no gap above gap K. (NOTE: Refer to Section 3.1.2 for further discussion of this input convention.)</p>
46-50	FACTOR(K)	<p>Enter 1.0 if gap positive flow (from channel IK(K) to channel JK(K)) is in the same direction as positive flow for the global coordinate system.</p> <p>Enter -1.0 if gap positive flow is opposite to positive flow for the global coordinate system. (Default = 1.0). (See Section 3.1.2 for explanation of this input.)</p>
51-55	IGAP(K,N)	<p>Enter the index numbers of gaps facing the IK(K) side of gap K. If the gap faces a wall, enter -1. If the gap faces a vessel connection to a one-dimensional component, enter -2.</p>
61-65		
71-75		
56-60	JGAP(K,N)	<p>Enter the gap numbers facing the JK(K) side of gap K. If the gap faces a wall, enter -1. If</p>
66-70		

76-80

the gap faces a vessel connection to a one-dimensional component, enter -2.
Up to three (3) sets of (IGAP,JGAP) may be entered.

Note: The input for FACTOR, IGAP and JGAP is required only if the three-dimensional form of the transverse momentum equation is desired.

VESSEL3.3. GMULT(K),ETANR(K) FORMAT(2F5.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	GMULT(K)	Enter the number of actual gaps modeled by gap K.
6-10	ETANR(K)	Enter the crossflow de-entrainment fraction.

VESSEL3.4 NLMGAP FORMAT(I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NLMGAP	Enter the number of gaps that convect orthogonal transverse momentum. (This is required only for the three-dimensional form of the transverse momentum equation.) Enter zero if the three-dimensional form of the transverse momentum equation is not desired.

Card VESSEL3.5 is read NLMGAP times.

VESSEL3.5 (KGAP1(N),KGAP2(N),KGAP3(N),N=1,NLMGAP) Read only if
 NLMGAP > 0. FORMAT(12I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5, 16-20, 31-35, 46-50	KGAP1(N)	Enter the index number of a gap whose velocity transports transverse momentum from one gap to another.
6-10, 21-25, 36-40, 51-55	KGAP2(N)	Enter the index number of the gap that receives the transverse momentum convected by the positive velocity of gap KGAP1. A nonzero value must be entered.
11-15, 26-30, 41-45, 56-60	KGAP3(N)	Enter the index number of the gap that the positive velocity of KGAP1 transports transverse momentum out of. (The positive velocity of KGAP1 transports momentum from KGAP3 to KGAP2. Note: The negative velocity of KGAP1 will transport transverse momentum in the opposite direction; i.e., from KGAP2 into KGAP3.) A nonzero number must be entered.

Up to four sets of (KGAP1, KGAP2, KGAP3) entries are specified per card. If NLMGAP > 4, repeat this card until all NLMGAP sets have been entered.

VESSEL Group 4: Vertical Channel Connection Data, read by subroutine SETIN

VESSEL4.1 NGROUP,NSECTS,NSIM,IREBAL FORMAT(4I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter four (4).
6-10	NSECTS	Enter the number of sections in this problem.
11-15	NSIM	Enter the number of simultaneous solution groups.
16-20	IREBAL	Enter the rebalancing option. Valid entries are: 1 = rebalance 0 = no rebalancing

Cards VESSEL4.2 and VESSEL4.3 are read in a group NSECTS times.

VESSEL4.2 ISEC,NCHN,NONODE,DXS(ISEC) FORMAT(3I5,F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	ISEC	Enter the section number. Begin with section number 1 on the bottom of the vessel and proceed toward the top, incrementing ISEC by 1.
6-10	NCHN	Enter the number of channels in section ISEC.
11-15	NONODE	Enter the number of vertical levels in section ISEC.
16-25	DXS(ISEC)	Enter the vertical node length in this section, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).

Card VESSEL4.3 is read NCHN times for each section.

VESSEL4.3, I,(KCHANA(I,J),J=2,7),(KCHANB(I,J),J=2,7) FORMAT(13I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter the identification number of a channel in section ISEC.

6-10, 11-15, etc.	KCHANA(I,J)	Enter the indices of channels in the section above ISEC that connect to channel I. If channel I does not connect to any channels above, enter I in KCHANA(I,2).
36-40 41-45, etc.	KCHANB(I,J)	Enter the indices of channels in the section below ISEC that connect to channel I. If channel I does not connect to any channels below, enter I in KCHANB(I,2).

VESSEL4.4 IWIDE FORMAT(I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	IWIDE	Enter the maximum difference between the index numbers of adjacent cells in a simultaneous solution group. (Refer to Section 3.1.3 for explanation of vessel cell numbering scheme and simultaneous solution groups.)

VESSEL4.5 (MSIM(I),I=1,NSIM) FORMAT(12I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	MSIM(I)	Enter the last cell number in each simultaneous solution group. (Note that this input asks for cell number, not vertical level number. Refer to Section 3.1.3.)

Twelve (12) values are entered per card. If NSIM is greater than 12, repeat card VESSEL4.5 until NSIM values have been entered.

VESSEL Group 5: Geometry Variation Data, read by subroutine SETIN

The input for this group allows the user to specify vertical variations in the continuity area, momentum area or wetted perimeter for channels, and in the transverse width for gaps. It can be omitted if such variations are not needed.

VESSEL5.1 NGROUP,NAFACT FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter five (5).
6-10	NAFACT	Enter the number of geometry variation tables to be entered.

Cards VESSEL5.2 and VESSEL5.3 are read in a group NAFACt times.

VESSEL5.2 NAXL(I) FORMAT(I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NAXL(I)	Enter the number of points in this variation table.

VESSEL5.3 (JAXL(I,N),AFACT(I,N),N=1,NAXL(I)) FORMAT(8(I5,F5.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,11-15, 21-25,31-35 41-45,51-55	JAXL(I,N)	Enter the node number at which to apply the area variation factor for table I, point N.
6-10,16-20, 26-30,36-40 46-50,56-60	AFACT(I,N)	Enter the variation factor for table I, point N. Area = AFACT(I,N)*AN(I), or Gap width = AFACT(I,N)*GAPN(K)

Eight pairs of (JAXL,AFACT) are entered per card. Repeat card VESSEL5.3 until NAXL pairs have been entered. The tables are numbered sequentially in the code, in the order they are read in on cards VESSEL5.2 and VESSEL5.3.

VESSEL Group 6: Channels and Gaps Affected by Variation Tables, read by subroutine SETIN

This group is read only if vertical variation tables have been specified in group 5.

VESSEL6.1 NGROUP,N1 FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter six (6).
6-10	N1	Enter the total number of channel and gap variation table cards to be read.

Card VESSEL6.2 is read N1 times.

VESSEL6.2 IACT,IAMT,IPWT,(ICRG(M),M=1,12) FORMAT(16I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	IACT	Enter a positive integer corresponding to a variation table number, for channel continuity area variation. Enter a negative integer, whose absolute value corresponds to a variation table number, for gap width variation.
6-10	IAMT	Enter a variation table number for channel momentum area variation. Enter zero (0) if IACT is negative.
11-15	IPWT	Enter a variation table number for wetted perimeter variation. Enter zero (0) if IACT is negative.
16-20	ICRG(M)	Enter the index numbers of the channels (or gaps if IACT is negative) that the tables identified in IACT, IAMT and IPWT are to be applied to. (Up to 13 channels or gaps may be specified per card.)

VESSEL Group 7: Local Loss Coefficient Data, read by subroutine SETIN

VESSEL7.1 NGROUP,NCD FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter seven (7).
6-10	NCD	Enter the number of loss coefficient specifications to be read. (These include vertical momentum losses only. Transverse losses are specified in group 3.)

Card VESSEL7.2 is read NCD times.

VESSEL7.2 CDL,J,(ICDUM(I),I=1,12) FORMAT(F5.0,13I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	CDL	Enter the loss coefficient (velocity head).
6-10	J	Enter the node number where the loss coefficient is applied. (NOTE: The vertical node number is relative to the beginning of the section containing the channel(s) listed in ICDUM(I).)
11-15	ICDUM(I)	Enter the index number(s) of channel(s) the loss coefficient will be applied to at node J. (Up to twelve (12) channels may use the specified loss coefficient CDL at vertical node J.)

VESSEL Group 8: Rod and Unheated Conductor Data, read by subroutine SETIN

VESSEL8.1 NGROUP,NROD,NSROD,NC,NRTAB FORMAT(5I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter eight (8).
6-10	NROD	Enter number of rods.
11-15	NSROD	Enter number of unheated conductors.
16-20	NC	Conduction model flag; Enter one (1) for radial conduction only. Enter two (2) for radial and axial conduction.
21-25	NRTAB	Enter number of temperature initialization tables to be read.

ROD GEOMETRY DATA

This card is read to define the geometry of structures that generate heat, including nuclear fuel rods. It is read NROD times.

VESSEL8.2 N,IFTYP(N),IAXP(N),NRENODE(N),DAXMIN(N),RMULT(N),RADIAL(N),HGAP(N)
FORMAT(4I5,4F10.0) Read only if NROD > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	N	Enter rod identification number. (Note: Rod index numbers must be entered sequentially, from 1 to NROD. Skipping numbers is not permitted.)
6-10	IFTYP(N)	Enter geometry type identification number. (Refer to group 9 for geometry type input data.)
11-15	IAXP(N)	Enter axial power profile table identification number. (Refer to group 11 for axial power profile input data.)
16-20	NRENODE(N)	Enter renoding flag for heat transfer solution for rod N

6-10	ISTYP(N)	Enter geometry type identification number. (Refer to group 9 for geometry type input data.)
11-20	HPERIM(N)	Enter wetted perimeter on outside surface, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-30	HPERIMI(N)	Enter wetted perimeter on inside surface, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). (Enter zero for a solid cylinder.)
31-40	RMULS(N)	Enter multiplication factor (number of elements modeled by unheated conductor N). (This number can contain fractional parts.)

ROD TEMPERATURE INITIALIZATION TABLES

Cards VESSEL8.4 through VESSEL8.7 are read to specify which temperature tables apply to which rods and unheated conductors. The sequence is repeated NRTAB times, and all rods and conductors must be accounted for.

VESSEL8.4 I,NRT1,NST1,NRAX1 FORMAT(4I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter identification number of temperature table.
6-10	NRT1	Enter number of rods using Table I.
11-15	NST1	Enter number of unheated conductors using Table I.
16-20	NRAX1	Enter number of pairs of elements in Table I.

VESSEL8.5 (IRTAB(I,L),L=1,NRT1) FORMAT(12I5) Read only if NRT1 > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	IRTAB(I,L)	Enter identification numbers of rods using Table I for temperature initialization.

Enter the negative of the rod identification number if the temperature boundary is to be applied to the inside surface of the rod.

Note: The steady-state conduction equation is solved for these rods using the temperatures from Table I as a boundary condition on the rod surface.

VESSEL8.6 (ISTAB(I,L),L=1,NST1) FORMAT(12I5) Read only if NST1 > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	ISTAB(I,L)	Enter identification numbers of unheated conductors using Table I for temperature initialization.

Note: A flat radial temperature profile is assumed initially in unheated conductors.

VESSEL8.7 (AXIALT(I,L),TRINIT(I,L),L=1,NRAX1) FORMAT(8F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	AXIALT(I,L)	Enter the vertical position relative to the bottom of the VESSEL, in inches, (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-30		
41-50		
61-70	TRINIT(I,L)	Enter the temperature to be applied at AXIALT(I,L), in °F (if ICOBRA = 0) or °K (if ICOBRA = 1).
11-20		
31-40		
51-60		
71-80		

Note: The vertical locations of the bottom and top of each rod or unheated conductor using table I must be contained within the range AXIALT(I,1) to AXIALT(I,NRAX1).

VESSEL Group 9: Conductor Geometry Description, read by subroutine SETIN

The geometry types are read in this group. The geometry types are numbered sequentially in the order they are read in. Nuclear rod geometry types are read using cards VESSEL9.2 through VESSEL9.5. All other geometry types are read using cards VESSEL9.6 and VESSEL9.7.

VESSEL9.1 NGROUP,NFUEL,IREF,ICONF,IMWR FORMAT(5I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter nine (9).
6-10	NFUEL	Enter number of geometry types to be read in. Note: A geometry type may be used by both rods and unheated conductors, but for the unheated conductor, any heat generation specified for the type will be ignored.
11-15	IREF	Fuel relocation flag (1=on, 0=off) (This is used only for nuclear fuel rods using the dynamic gap conductance model).
16-20	ICONF	Fuel degradation flag (1=on,0=off) (NOTE: if IREF=1, then ICONF=1.)
21-25	IMWR	Flag for metal-water reaction (zirconium dioxide only) =0, off =1, Cathcart (for Best Estimate analysis) =2, Baker-Just (for Evaluation Model analysis)

NUCLEAR FUEL GEOMETRY TYPES

These data are read only for nuclear fuel geometry types. If FTYPE(I) is not entered as NUCL, the geometry data is interpreted by line VESSEL9.6.

VESSEL9.2 I, FTYPE(I), DROD, DFUEL(I), NFUEL, IMATF, IMATC, IMATOX(I), DCORE, TCLAD, FTDENS(I), IGPC(I), IGFORC, IRADP (Read only for nuclear geometry types.)
 FORMAT(I5, 1X, A4, 2F10.0, 4I5, 3F5.0, 3I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter the geometry type identification number. (Note: Geometry type index numbers must be entered sequentially, from 1 to NFUEL. Skipping numbers is not permitted.)
7-10	FTYPE(I)	Enter NUCL
11-20	DROD	Enter rod outside diameter, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-30	DFUEL(I)	Enter fuel pellet diameter, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
31-35	NFUEL	Enter number of radial nodes in fuel pellet.
36-40	IMATF	Fuel material properties flag; Enter zero (0) for built-in UO ₂ properties. Enter a positive integer corresponding to the identification number of a material properties tables for user-input properties. (Refer to group 10 for material properties input data.)
41-45	IMATC	Clad material properties flag; Enter zero (0) for built-in zirconium properties. Enter a positive integer corresponding to the identification number of a material properties table for user-input properties. (Refer to group 10 for material properties input data.)
46-50	IMATOX(I)	Clad oxide property flag; Enter zero (0) for built-in zirconium dioxide properties.

Enter a positive integer corresponding to the identification number of a material properties table for user-input properties (see group 10).

51-55 DCORE Enter diameter of central void for cored fuel, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
 Enter zero for uncored fuel.

56-60 TCLAD Enter clad thickness, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).

61-65 FTDENS(I) Enter fuel theoretical density as a fraction (used only if built-in UO₂ properties have been flagged; i.e., if IMATF = 0).
 (Note: Do not enter zero.)

66-70 IGPC(I) Gap conductance option flag;
 Enter zero (0) for constant gap conductance (as specified by HGAP(N) on card VESSEL8.2).

Enter a positive integer for user-specified nonuniform gap conductance (entered on card VESSEL9.4 in a table of IGPC(I) elements).

Enter a negative integer for the dynamic gap conductance model. (|IGPC(I)| is the number of entries in the cold gap width vs axial location table, read on card VESSEL9.4.)

71-75 IGFORC Flag for temporal forcing function on gap conductance (valid only if IGPC(I) > 0);
 Enter zero (0) for constant gap conductance.
 Enter a positive integer for a temporal forcing function with IGFORC table entries.

76-80

IRADP

Enter number of entries in radial power profile table for the fuel pellet.

Enter zero (0) for a uniform radial power profile.

VESSEL9.3 PGAS(I),VPLEN(I),ROUFF(I),ROUFC(I),(GSFRAC(L),L=1,6), OXIDET
 Read only if FTYPE(I) = NUCL and IGPC(I) < 0.
 FORMAT(4F10.0,6F5.,F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	PGAS(I)	Enter cold pin gas pressure for nuclear fuel rod geometry type I, in psia (if ICOBRA = 0) or N/m ² (if ICOBRA = 1).
11-20	VPLEN(I)	Enter gas plenum volume, in in. ³ (if ICOBRA = 0) or m ³ (if ICOBRA = 1).
21-30	ROUFF(I)	Enter fuel pellet surface roughness, inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
31-40	ROUFC(I)	Enter surface roughness of clad inner surface, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). (Note: Fuel and clad surface roughness should correspond to those used in FRAPCON-2 since the correlation is empirical.) Fuel surface ROUFF = 0.000085 inches Clad surface ROUFC = 0.000045 inches
41-45	GSFRAC(1)	Enter molar fraction of helium gas present.
46-50	GSFRAC(2)	Enter molar fraction of xenon gas present.
51-55	GSFRAC(3)	Enter molar fraction of argon gas present.
56-60	GSFRAC(4)	Enter molar fraction of krypton gas present.
61-65	GSFRAC(5)	Enter molar fraction of hydrogen gas present.
66-70	GSFRAC(6)	Enter molar fraction of nitrogen gas present.
		Note: $\sum_{L=1}^6 \text{GSFRAC}(L) = 1.0$
71-80	OXIDET	Enter initial oxide thickness for the zircaloy metal-water reaction rate equation in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). (Used only if IMWR > 0.)

VESSEL9.4 (AXJ(I,L),AGFACT(I,L),L=1,|IGPC(I)|) Read only if FTYPE(I) = NUCL and |IGPC(I)| > 0. FORMAT(8F10.0)

Columns	Variable	Description
1-10	AXJ(I,L)	Enter top-most vertical position, measured from the bottom of the rod, at which the cold gap width (or gap conductance) AGFACT(I,L) is applied. (All vertical levels below AXJ(I,L) and above AXJ(I,L-1) will have AGFACT(I,L) for gap width (or gap conductance.) Units on AXJ(I,L) are inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-30		
41-50		
61-70		
11-20	AGFACT(I,L)	Enter cold gap width if IGPC(I) is negative. Units are inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
31-40		
51-60		Enter gap conductance if IGPC(I) is positive. Units are Btu/hr-ft ² -°F (if ICOBRA = 0) or W/m ² -°C (if ICOBRA = 1).
71-80		

VESSEL9.5 (RODP(L),POWR(L),L=IRADP) Read only if FTYPE(I) = NUCL and IRADP > 0. FORMAT(8F10.0)

Columns	Variable	Description
1-10	RADP(L)	Enter the relative radial location (r/r ₀) where corresponding power factor (POWR(L)) is applied.
21-30		
41-50		
61-70		

$$\frac{r}{r_0} = \frac{(\text{radius} - \text{DCORE}/2)}{\frac{1}{2} (\text{DFUEL}(I) - \text{DCORE})}$$

Note: RADP(1) must be zero (0.0)
 RADP(IRADP) must be one (1.0)

11-20 POWR(L) Enter the relative power factor (i.e., the
 31-40 ratio of local power at location RADP(L) to
 51-60 total rod power.)
 71-80

NON-NUCLEAR GEOMETRY TYPES

These data are read for all geometry types that do not describe nuclear fuel.

VESSEL9.6 I,FTYPE(I),DROD,DIN,NFUEL,IMATOX(I),IMATIX(I) Read only if
 FTYPE(I) ≠ NUCL FORMAT(I5,1X,A4,2F10.0,3I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter geometry type identification number.
7-10	FTYPE(I)	Enter four-character alphanumeric geometry type flag: HROD = solid cylinder TUBE = hollow tube WALL = flat plate
11-20	DROD	Enter outside diameter for HROD or TUBE geometries, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). Enter the wetted perimeter for WALL geometries, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-30	DIN	Enter inside diameter for TUBE geometries, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1). Enter thickness for WALL geometries, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).

Enter zero (0.0) for (HROD) solid cylinder geometries.

31-35 NFUEL Enter the number of regions within the conductor. (Each region has a uniform power profile and consists of one material.)

36-40 IMATOX(I) Enter material property table identification number for oxide on outside surface. (Default is zirconium oxide; IMATOX(I) = 0).
Enter the index number of the material property table for material in region NFUEL if there is no oxide present.

41-45 IMATIX(I) Enter material property table identification number for oxide on inside surface (Default is zirconium oxide; IMATIX(I) = 0); applies only to TUBES or WALLS.)
Enter the index number of the material property table for material in region 1 if there is no oxide present.

Data sets for the NFUEL regions of geometry type I are entered starting at the centerline for HROD types and at the inside surface for TUBE and WALL types. Data sets are entered in sequence moving radially toward the outside surface.

VESSEL9.7 (NODER(L),MATR(L),TREG(L),QREG(L),L=i,NFUEL) Read only if
FTYPE(I) ≠ NUCL. FORMAT(4(2I5,2F5.0))

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NODER(L)	Enter the number of radial heat transfer nodes in region L.
21-25		
41-45		
61-65		
6-10	MATR(L)	Enter the material property table identification number for region L.
26-30		
46-50		
66-70		

11-15	TREG(L)	Enter the thickness of region L, in inches
31-35		(if ICOBRA = 0) or meters (if ICOBRA = 1).
51-55		Note: For TUBE and HROD geometry types,
71-75		$\sum_{L=1}^{NFUEL} TREG(L) = .5 (DROD - DIN)$
16-20	QREG(L)	Enter radial power factor for region L. (This
36-40		profile is automatically normalized to unity.)
56-60		
76-80		

VESSEL Group 10: Material Properties Tables, read by subroutine SETIN

This input group is required only if user-supplied material properties were flagged by input in group 9 (i.e., with nonzero values for IMATF, IMATC, IMATOX(I), IMATIX(I) or MATR(L) for any geometry type). If only default material properties are used, (i.e, zircaloy and UO₂), this group is omitted.

VESSEL10.1 NGROUP,NMAT FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter ten (10).
6-10	NMAT	Enter number of material properties tables to be read in.

Cards VESSEL10.2 and VESSEL10.3 are read in pairs NMAT times.

VESSEL10.2 N,NNTDP,RCOLD(N) FORMAT(2I5,F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	N	Enter the material property table identification number.
6-10	NNTDP	Enter the number of entries in material properties table N.
11-20	RCOLD(N)	Enter the cold density in lbm/ft ³ (if ICOBRA = 0), or kg/m ³ (if ICOBRA = 1) for material N. (This value is used to define the mass in the heat transfer nodes composed of material type N.)

VESSEL10.3 (TPROP(I,N),CPF1(I,N),THCF(I,N) I=1,NNTDP) FORMAT(6F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	TPROP(I,N)	Enter the temperature for entry I in material property table N. Units are °F (if ICOBRA = 0) or °K (if ICOBRA = 1).
31-40		

11-20	CPF1(I,N)	Enter the specific heat for entry I in material property table N. Units are Btu/lb-°F (if ICOBRA = 0) or J/kg°C (if ICOBRA = 1).
41-50		
21-30	THCF(I,N)	Enter the thermal conductivity for entry I in material property table N. Units are Btu/hr-ft-°F (if ICOBRA = 0) or W/m°C (if ICOBRA = 1).
51-60		

VESSEL Group 11: Axial Power Tables and Forcing Functions, read by subroutine SETIN

VESSEL11.1 NGROUP,NAXP,NQ,NGPFF FORMAT(4I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter eleven (11).
6-10	NAXP	Enter number of axial power profile tables to be read. (Minimum of one.)
11-15	NQ	Enter number of pairs of elements in the power forcing function table.
16-20	NGPFF	Enter zero (0) if power is constant. Enter number of pairs of elements in gap conductance forcing function table. Enter zero (0) if there is no forcing function on gap conductance.

Axial Power Tables

Cards VESSEL11.2 and VESSEL11.3 are read in pairs NAXP times.

VESSEL11.2 I,NAXN(I) FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter axial power profile table identification number.
6-10	NAXN(I)	Enter number of pairs of elements in axial power profile Table I.

VESSEL11.3 (Y(I,N),AXIAL(I,N),N=1,NAXN(I)) FORMAT(8F10.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	Y(I,N)	Enter vertical location, relative to bottom of the VESSEL, where axial power factor AXIAL(I,N) is applied. Use inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
21-30		
41-50		
61-70		
11-20	AXIAL(I,N)	
31-40		Enter relative power factor (the ratio of local power to average power) at vertical location Y(I,N).
51-60		
71-80		

All rods using the same table should start and end at the same vertical locations. In the table, Y(I,1) must be the vertical location of the beginning of the rods, and Y(I,NAXN(I)) must be the vertical location of the end of the rods.

Power Forcing Function

VESSEL11.4 (YQ(N),FQ(N),N=1,NQ) FORMAT(8F10.0) Read only if NQ > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	YQ(N)	Enter transient time (seconds).
21-30		
41-50		
61-70		
11-20	FQ(N)	
31-40		Enter the power factor: $FQ(N) = \frac{\text{Power at time } YQ(N)}{\text{initial power}}$
51-60		
71-80		

Gap Conductance Forcing Function

VESSEL11.5 (YGPFF(N),FGPFF(N),N=1,NGPFF) FORMAT(8F10.0) Read only if
NGPFF > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10	YGPFF(N)	Enter transient time (in seconds).
21-30		
41-50		
61-70		
11-20	FGPFF(N)	Enter conductance factor:
31-40		
51-60		$FGPFF(N) = \frac{\text{gap conductance at time YGPFF(N)}}{\text{initial gap conductance}}$
71-80		

VESSEL Group 12: Turbulence and Viscous Stress Data, read by subroutine SETIN

VESSEL12.1 NGROUP,N1 FORMAT(2I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter twelve (12).
6-10	N1	Enter number of sections in which turbulence or viscous stress will be applied.

Card VESSEL12.2 is read N1 times, once for each section with turbulence or viscous stresses. For each section ISEC, card VESSEL12.3 is read if ITURB(ISEC)=1, and card VESSEL12.4 is read if ITURB(ISEC)=2.

VESSEL12.2 ISEC,IVIS(ISEC),ITURB(ISEC) FORMAT(3I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	ISEC	Enter identification number of section using turbulence or viscous stress calculation.
6-10	IVIS(ISEC)	Enter viscous stress flag for section ISEC. Valid entries are: 0 = do not apply viscous stress 1 = apply viscous stress
11-15	ITURB(ISEC)	Enter turbulence flag for section ISEC. Valid entries are: 0 = do not apply turbulence 1 = apply turbulence with uniform turbulence properties 2 = apply turbulence with channel varying turbulence properties

Card VESSEL12.3 is read if ITURB(ISEC) = 1. This card is read only once for a section and the same properties are applied to all channels within the section.

VESSEL12.3 ZLMXXL(ISEC),ZLMXXV(ISEC),F11(ISEC),F22(ISEC),TPRNOL(ISEC),
 TPRNOV(ISEC) Read only if ITURB(I) = 1. FORMAT(6F5.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	ZLMXXL(ISEC)	Enter liquid mixing length for section ISEC, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
6-10	ZLMXXV(ISEC)	Enter vapor mixing length for section ISEC, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
11-15	F11(ISEC)	Enter vertical anisotropy factor for section ISEC.
16-20	F22(ISEC)	Enter transverse anisotropy factor for section ISEC.
21-25	TPRNOL(ISEC)	Enter liquid turbulent Prandtl number for section ISEC.
26-30	TPRNOV(ISEC)	Enter vapor turbulent Prandtl number for section ISEC.

Card VESSEL12.4 is read if ITURB(ISEC) = 2. The card is read once for each channel in section ISEC.

VESSEL12.4 I,ZLMXXL(I),ZLMXXV(I),F11(I),F22(I),TPRNOL(I),TPRNOV(I) Read only
if ITURB(I) = 2. FORMAT(I5,6F5.0)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	I	Enter identification number of a channel in section ISEC.
6-10	ZLMXXL(I)	Enter liquid mixing length for channel I, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
11-15	ZLMXXV(I)	Enter vapor mixing length for channel I, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).
16-20	F11(I)	Enter vertical anisotropy factor for channel I.
21-25	F22(I)	Enter transverse anisotropy factor for channel I.
26-30	TPRNOL(I)	Enter liquid turbulent Prandtl number for channel I.
31-35	TPRNOV(I)	Enter vapor turbulent Prandtl number for channel I.

VESSEL Group 13: Boundary Condition Data, read by subroutine SETIN

VESSEL13.1 NGROUP,NIBND,NKBND,NFUNCT,NGBND FORMAT(5I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter thirteen (13).
6-10	NIBND	Enter the total number of vertical mesh cell boundary conditions.
11-15	NKBND	Enter the total number of transverse momentum cells for which crossflow will be set to zero.
16-20	NFUNCT	Enter the number of forcing functions for the boundary conditions.
21-25	NGBND	Enter the number of groups of contiguous transverse momentum cells for which crossflows will be set to zero.

VESSEL13.2 (NPTS(K),K=1,NFUNCT) FORMAT(16I5) Read only if NFUNCT > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,...	NPTS(K)	Enter the number of points (pairs of values) in forcing function table K.

Sixteen (16) values are entered per card. Repeat card VESSEL13.2 until NFUNCT values have been entered.

Card VESSEL13.3 is read NFUNCT times.

VESSEL13.3 ((ABSCIS(K,I),ORDINT(K,I),I=1,NPTS(K)),K=1,NFUNCT) Read only if
 NFUNCT > 0. FORMAT(5(F5.0,F10.0))

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,16-20, 31-35,46-50, 61-65	ABSCIS(K,I)	Enter the time, in seconds, the factor is applied.
6-15,21-30, 36-45,51-60, 66-75	ORDINT(K,I)	Enter the forcing function factor to be applied at time ABSCIS(K,I).

Five (5) pairs of (ABSCIS,ORDINT) are entered per card. Repeat card VESSEL13.3 until NPTS(K) points have been entered for forcing function table K. Continue entering data until NFUNCT tables have been specified.

Card VESSEL13.4 is read NIBND times.

VESSEL13.4 (IBOUND(L,M),L=1,2),ISPEC(M),NPFN(M),NHFN(M),PVALUE(M),HVALUE(M)
 FORMAT(5I5,2F10.0) Read only if NIBND > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	IBOUND(1,M)	Enter the index number of the channel boundary condition M applies to.
6-10	IBOUND(2,M)	Enter the vertical node number at which boundary condition M is applied. (NOTE: The node number is referenced to the beginning of the section that the channel identified in IBOUND(1,M) resides in.)
11-15	ISPEC(M)	Enter the boundary condition type. Valid options are: 1 = pressure and enthalpy boundary condition 2 = flow and enthalpy 3 = flow only 4 = mass source (flow rate and enthalpy) 5 = pressure sink and enthalpy

- 16-20 NPFN(M) Enter the index number of the forcing function table by which the first parameter of the boundary condition will be varied. (NOTE: The forcing function tables are numbered sequentially in the order they are read in on card VESSEL13.3.) For example: If ISPEC(M) = 1 and NPFN(M) = 3, the specified pressure will be adjusted according to the third forcing function entered on VESSEL13.3. Enter zero if the boundary condition is constant.
- 21-25 NHFN(M) Enter the index number of the forcing function table by which the second parameter of the boundary condition will be varied. (For example: If ISPEC(M) = 1 and NHFN(M) = 6, the specified enthalpy will be adjusted according to the 6th forcing function specified on VESSEL13.3.) Enter zero if the boundary condition is constant.
- 26-35 PVALUE(M) Enter the first boundary value. If ISPEC(M) = 1 or 5, enter pressure, in psia (if ICOBRA = 0) or N/m^2 (if ICOBRA = 1). If ISPEC(M) = 2, 3 or 4, enter flow rate, in lbm/sec (if ICOBRA = 0) or kg/sec (if ICOBRA = 1).
- 36-45 HVALUE(M) Enter enthalpy in Btu/lbm (if ICOBRA = 0) or J/kg (if ICOBRA = 1). Enter zero (0) if ISPEC(M) = 3.

Card VESSEL13.5 is read after card VESSEL13.4 has been read NIBND times, and only if some ISPEC(M) has been specified as 5 (i.e., a pressure sink boundary condition). Card VESSEL13.5 is read once for each pressure sink, in the same order they are specified in the input for card VESSEL13.4.

VESSEL13.5 ASINK(K),SINKK(K),DXSINK(K) FORMAT(3F5.0) Read only if some ISPEC(M) = 5.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	ASINK(K)	Enter the flow area of the pressure sink, in in. ² (if ICOBRA = 0) or m ² (if ICOBRA = 1).
6-10	SINKK(K)	Enter the loss coefficient (velocity head) of the pressure sink.
11-15	DXSINK(K)	Enter the length of the momentum control volume for the sink, in inches (if ICOBRA = 0) or meters (if ICOBRA = 1).

Card VESSEL13.6 is read NGBND times.

Card VESSEL13.6 is read only if some ISPEC(M) has been specified as 4 (i.e., a mass injection boundary condition). Card VESSEL13.6 is read once for each mass injection boundary condition, in the same order they are specified in the input for card VESSEL13.4.

VESSEL13.6 AINJT(K) FORMAT(F5.0) Read only if some ISPEC(M) = 4.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	AINJT(K)	Enter the flow area of the mass injection in in. ² (if ICOBRA = 0) or m ² if (ICOBRA = 1).

VESSEL13.7 K,JSTART,JEND FORMAT(3I5) Read only if NGBND > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	K	Enter the gap number to which a zero (0.0) crossflow is to be applied.
6-10	JSTART	Enter the continuity cell number at which to start applying the zero crossflow.
11-15	JEND	Enter the continuity cell number at which to stop applying the zero crossflow. (Note: The crossflow will be set to zero for

gap K between nodes JSTART and JEND. The node numbers are given relative to the beginning of the section containing gap K.)

This card may be repeated as many times as necessary for a given gap K, to identify all axial levels that have zero crossflow. The total number of transverse momentum cells with zero crossflow boundary conditions specified by card VESSEL13.6 must sum to NKBND.

VESSEL Group 14: Output Options, read by subroutine SETIN

VESSEL14.1 NGROUP,N1,NOUT1,NOUT2,NOUT3,NOUT4,IROPP,IOPT FORMAT(8I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NGROUP	Enter fourteen (14).
6-10	N1	Enter the general vessel output option. Valid entries are: 1 = print channels only 2 = print channels and gaps only 3 = print rods and unheated conductors only 4 = print rods, unheated conductors and channels only 5 = print channels, gaps, rods and unheated conductors
11-15	NOUT1	Enter the number of channels to be printed (used if N1 = 1, 2, 4 or 5). If NOUT1 = 0, all channels will be printed. If NOUT1 > 0, an array of NOUT1 channel numbers must be entered on card VESSEL14.2.
16-20	NOUT2	Enter the number of rods to be printed, (used if N1 > 2). If NOUT2 = 0, all rods will be printed. If NOUT2 > 0, an array of NOUT2 rod numbers must be entered on card VESSEL14.4.
21-25	NOUT3	Enter the number of gaps to be printed (used if N1 = 2 or 5). If NOUT3 = 0, all gaps will be printed. If NOUT3 > 0, an array of NOUT3 gap numbers must be entered on card VESSEL14.3.
26-30	NOUT4	Enter the number of unheated conductors to be printed (used if N1 > 2). If NOUT4 = 0, all unheated conductors will be printed.

If NOUT4 > 0, an array of NOUT4 unheated conductor numbers must be entered on card VESSEL14.5.

31-35 IPROPP Enter the property table print option. Valid entries are:

- 0 = do not print the property table
- 1 = print the property table

36-40 IOPT Enter the debug print option. Valid entries are:

- 0 = normal printout only
- 1 = debug printout (print extra data for channels, rods and gaps)

VESSEL14.2 (PRINTC(I),I=1,NOUT1) FORMAT(16I5) Read only if N1 ≠ 3 and NOUT1 > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,...	PRINTC(I)	Enter the index numbers of channels to be printed. Sixteen values are entered per card. Repeat this card until NOUT1 values have been entered.

VESSEL14.3 (PRINTG(I),I=1,NOUT3) FORMAT(16I5) Read only if N1 = 2 or 5 and NOUT3 > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,...	PRINTG(I)	Enter the index numbers of gaps to be printed. Sixteen values are entered per card. Repeat this card until NOUT3 values have been entered.

VESSEL14.4 (PRINTR(I),I=1,NOUT2) FORMAT(16I5) Read only if N1 > 2 and NOUT2 > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,...	PRINTR(I)	Enter the index numbers of rods to be printed. Sixteen values are entered per card. Repeat this card until NOUT2 values have been entered.

VESSEL14.5 (PRINTS(I),I=1,NOUT4) FORMAT(16I5) Read only if NOUT4 > 0.

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,...	PRINTS(I)	Enter the index numbers of unheated conductors to be printed. Sixteen values are entered per card. Repeat this card until NOUT4 values have been entered.

VESSEL14.6 NDUMEND
FORMAT(I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NDUMEND	Enter zero (0) to terminate VESSEL group control card input.

VESSEL Connections to One-Dimensional Components, read by subroutine COBRAI

COBRA.4 NCSR
FORMAT(I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NCSR	Enter the number of connections to the vessel. Minimum is 1.

Card COBRA.5 is read NCSR times.

COBRA.5 (ISRL(N),ISRC(N),ISRF(N),JUNS(N),N=1,NCSR) FORMAT(4I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	ISRL(N)	Enter the index number of the channel where the connection is located.
6-10	ISRC(N)	Enter the index number of the vertical node in channel ISRL(N) where the connection is located if the connection is made at the top or side of the cell. Enter the negative of the node number if the connection is made at the bottom of the cell.
11-15	ISRF(N)	Enter the gap number facing the vessel connection if the connection is at the side of the cell. Enter 0 if the connection is on the side and faces a wall. Enter -1 if the connection is vertical.
16-20	JUNS(N)	Enter the junction number connecting a one-dimensional component to the vessel.

RESTART INPUT DATA FLAG, read by subroutine RDCOMP

Card COBRA.6 is read only on a restart where fewer than NCOMP components have been specified via the input file.

COBRA.6 EFLG Read only if INIT = 4 and total number of components in the input is less than NCOMP. FORMAT(A3)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-3	EFLG	Enter END to stop reading component data from input file and read remaining one-dimensional component data from TRCRST restart file.

Graphics Options, read by subroutine IGRAF

GRAF.1 MOVIE,TMOVIE FORMAT(I5,F10.2)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	MOVIE	Enter the movie process option. Valid entries are: 0 = no movie (recommended option) 1 = save data for particle tracker movie. Note: The COBRA/TRAC code has the capability to dump data that can be used to make particle-tracker movies of simulations. However, the software to actually make the particle-tracker movies is not part of the COBRA/TRAC package, and will be released and documented separately.
6-15	TMOVIE	Enter the time interval at which to save data for the movie. Valid only if MOVIE = 1.

GRAF.2 NCGRAF,MXGDMP,IGRFOP,NLLR FORMAT(4I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	NCGRAF	Enter the maximum number of components for which graphics data may be saved. (This <u>must</u> be equal to the value assigned to RESPEC parameter MH.) Enter zero (0) if not used.
6-10	MXGDMP	Enter the maximum number of time steps for which graphics data will be saved. Absolute maximum is MI (RESPEC parameter). (Note: This cannot be changed on a restart.)
11-15	IGRFOP	Enter the vessel graphics option, as follows: 0 for normal vessel dump (all variables saved for all vessel computational cells) N, where N is the number of user-selected vessel variables to be saved for graphics

GRAF.4 (IGRFIT(I),I=1,NCGRAF) Read only if NCGRAF > 0.
 FORMAT(12I5)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5,...	IGRFIT(I)	Enter the index numbers of components for which data is to be saved. NCGRAF entries will be read, but zeroes are permissible. (A zero entry can be changed to a component number on a restart, and a component number can be changed to a zero. But to continue saving data for a given component on a restart, it must be entered in the same location in the IGRFIT array. For example, if component #43 was entered originally in the 5th location, (i.e., IGRFIT(5)=43) all subsequent restarts must specify component #43 as the fifth element of the array.)

GRAF.5 ((IGRF(I,J),J=1,2),GRFN(I),I=1,IGRFOP) Read only if IGRFOP > 0.
 FORMAT(5(2I5,F5.0))

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5	IGRF(I,1)	Enter a number signifying the vessel variable to be saved. Valid entries are:
16-20		Channel variables--
31-35		1 = pressure
46-50		2 = void fraction
61-65		3 = entrained liquid fraction
		4 = liquid fraction
		5 = liquid temperature
		6 = vapor temperature
		7 = liquid density
		8 = vapor density
		9 = vapor flow

- 10 = liquid flow
- 11 = entrained liquid flow
- 12 = vapor generation rate
- 13 = heat transfer rate to liquid
- 14 = heat transfer rate to vapor
- 15 = drop mass entrainment rate
- 16 = drop mass de-entrainment rate
- 17 = drop interfacial area density
- 18 = ETAENP
- 19 = HASHL
- 20 = HASCL
- 21 = HASHV
- 22 = flow regime
- 23 = boundary condition type
- 24 = interfacial drag coefficient (XK)
- 25 = interfacial drag coefficient (XKGE)
- 37 = mixture density; $(\alpha\rho_v + (1-\alpha)\rho_l)$

Gap variables--

- 26 = vapor crossflow
- 27 = liquid crossflow
- 28 = entrained liquid crossflow

Rod Variables--

- 29 = heat transfer mode
- 30 = liquid temperature (seen by rod
IGRF(I,2))
- 31 = vapor temperature (seen by rod
IGRF(I,2))
- 32 = heat transfer coefficient to liquid
- 33 = heat transfer coefficient to vapor
- 34 = heat flux
- 35 = rod surface temperature
- 36 = peak temperature on rod identified in
IGRF(I,2) at axial node GRFN(I)

6-10	IGRF(I,2)	Enter channel, rod or gap number for which
21-25		selected variable will be saved. Parameters
36-40		1-25, and 37 are channel variables; 26-28 are
51-55		gap variables; 29-36 are rod variables. If
66-70		IGRF(I,1) = 35 or 36 and the <u>inside</u> surface
		temperature is desired, enter the <u>negative</u> of
		the rod number.
11-15	GRFN(I)	Enter the vertical node number for which the
26-30		selected variable will be saved.
41-45		For rod variables, enter the vertical elevation
		in inches. (Must be within 1 inch of a
56-60		vertical node location.
71-75		For IGRF(I,1)=36, enter the radial node index
		(Node 1 is on the inside of the rod)

2.4 TIME DOMAIN DATA, read by subroutine TIMSTP

After all component data has been entered the user must define the time domain for the simulation. The total time can be divided into several domains of specified duration. Each time domain can have different minimum and maximum time step sizes and different edit intervals. To terminate the calculation a time domain with a negative time step size is entered. Two cards are required to specify the data for each time domain.

TIME.1 DTMIN,DTMAX,TEND,RTWFP,TMAX FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	DTMIN	Enter the minimum time step allowed for this domain, in seconds. Enter a <u>negative</u> value to terminate the calculation.
15-28	DTMAX	Enter the maximum time step allowed for this domain, in seconds.
29-42	TEND	Enter the end of this time domain, in seconds.
43-56	RTWFP	Enter the ratio of conduction solution and fluid solution time step sizes. (Used to obtain steady-state conditions. The conduction solution can generally take time steps greater than the fluid solution. For transient calculations, RTWFP should be one.)
57-70	TMAX	Enter the maximum CPU time allowed for this run. If this CPU limit is reached during this simulation time domain, the run will terminate. (Dump files will be written, so the calculation can be restarted.)

TIME.2 EDINT,GFINT,DMPINT,SEDINT,TCKLEND FORMAT(5E14.6)

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-14	EDINT	Enter the print interval for this time domain. Output will be printed every EDINT seconds.
15-28	GFINT	Enter the graphics interval for this time domain. Data for graphics will be saved every GFINT seconds.
29-42	DMPINT	Enter the restart dump interval. Data for restarts will be saved every DMPINT seconds. (NOTE: For larger problems, the restart dumps create relatively large files, and can become unmanageable if too many dumps are written during a simulation. Regardless of the value for input DMPINT, the dump logic automatically writes dumps at the beginning of a calculation, at the end of a calculation, and after every 60 minutes of CPU time during the calculation. This is usually quite sufficient, and DMPINT can be set to a large value.)
43-56	SEDINT	Enter the "short edit" interval for this time domain. (A "short edit" is an abbreviated print.) Short edits will be performed every SEDINT seconds.

57-70

TCLKEND

Enter the wall clock time (decimal, 24 hour clock) at which to stop the calculation and create a restart dump and edit. (This is useful for running the calculation during particular shifts; for example, TCLKEND can be entered as 7.5 to stop the calculation at 7:30am.

Enter zero if this feature is not used.

Repeat cards TIME.1 and TIME.2 for each time domain desired. A final time domain with a negative value for DTMIN must be entered to terminate the calculation.

2.5 SPECSET REDIMENSIONING PROGRAM

Allocation of array storage in COBRA/TRAC reflects a major difference between COBRA-TF and TRAC. TRAC uses two large arrays for variable storage, with variable locations calculated for each problem using pointer arrays. COBRA/TRAC has retained the pointer arrays and locating logic for the one-dimensional components. COBRA-TF uses a set of COMDECKS containing array variables which can be redimensioned using a preprocessor program, SPECSET. When redimensioning the COBRA/TRAC code, both of these storage allocation methods must be accommodated.

This section discusses the procedure for redimensioning the COBRA/TRAC code on a CDC machine with the UPDATE facility. The procedure consists of a series of updates (outlined below) that define the sizes of the COMDECK arrays for the VESSEL component, and the length of the A and ALCM arrays for the one-dimensional components.

The COMDECKS for the vessel component of COBRA/TRAC consist of sets of common blocks with dimension parameters instead of numbers defining the arrays. An example of a typical COMDECK line is

```
COMMON/DUMMY/ A(MC,MX), B(MT)
```

These common blocks are, obviously, unusable in this form. No computer would successfully compile them. The SPECSET program is used to convert the dimension parameter to numbers. The user supplies input to SPECSET to define the parameters, (for example, MC=20, MX=100, MT=5), and SPECSET creates a new COMDECKS file with lines such as

```
COMMON/DUMMY/ A(20,100), B(5)
```

This new COMDECK file is copied into the COBRA/TRAC code via UPDATE, and the code can then be compiled. The user must supply values for 41 dimension parameters for SPECSET to redimension COBRA/TRAC. The line-by-line input for SPECSET is listed in Section 2.5.1, and the 41 dimension parameters which must be defined are listed in Table 2.1.

The A and ALCM arrays to define the size of the storage allocated for the one-dimensional components can be changed only by changes to the code (specifically, in common decks BLANKCOM and LCMSPACE, and subroutine BLKDAT). The appropriate sizes for the A and ALCM arrays for a particular problem are difficult to estimate a priori. The user could, for a particular problem, work through the logic in the code by hand to add up the storage allocations for each component, and thus arrive at a value for the storage required. This is a very tedious process, however. It is much simpler to optimistically assume that the standard dimensions in the code (of 5000 for the A array and 10,000 for the ALCM array) will be large enough for the problem, and execute the code. In processing the input data, the code automatically adds up the storage required by the problem, and checks it against the existing dimensions. If the arrays are too small, the code prints an error message telling how much more room is needed.^(a)

The Job Control Language (JCL) and input files required for redimensioning COBRA/TRAC are illustrated below. Although the procedure shown redimensions both the vessel arrays (through SPECSET) and the one-dimensional arrays (through UPDATE changes) it should be noted that either may be done separately. The one-dimensional arrays can be redimensioned, starting with any complete update library of the code. Similarly, SPECSET redimensioning can be executed without redimensioning the one-dimensional arrays. The complete procedure is as follows;

1. Create a source file of the COMDECKS.

```
UPDATE2(I=CDECK,L=0,S=TAPE3)
```

CDECK is a card-image file of the COMDECKs ABSORD through RESPAR, with the dimension parameters in place of numbers.

TAPE3 is a source file created by UPDATE from CDECK, which will be read by SPECSET.

(a) TRAC originally contained a variable dimensioning capability that would automatically adjust the array sizes for a given problem. But since this logic is of necessity machine dependent, it was not included in COBRA/TRAC.

2. Create a compile file of program SPECSET and compile it
 UPDATE2(I=RESPEK,C,L=0)
 RESPEK is a card-image file of program SPECSET.
 FTN(I=COMPILE,CNAME=SPECS,SL=0,R=0)
 SPECS is an executable binary of SPECSET.
3. Execute SPECSET. (Input file TAPE2 and the COMDECKs file TAPE3 must be available. See sample input in Table 2.2)
 SPECS
 SPECSET creates a file on TAPE4 that contains the vessel COMDECKs with numbers in place of the dimension parameters.
4. Remove the old COMDECKs from COBRA/TRAC.
 UPDATE2(P=CT012U,N=NEW1,I=A,L=0,C=0,F)
 A is an update input file that removes the old COMDECKs. It contains 1 line;
 *PURDECK ABSORD.RESPAR
 CT012U is an old program library of COBRA/TRAC.
 NEW1 is the new program library of COBRA/TRAC created by the update.
5. Complete the purge of the old COMDECKs from COBRA/TRAC.
 UPDATE2(P=NEW1,N=NEW2,I=B,C=0,L=0,F,E)
 B is a dummy update input file consisting of 1 line;
 *ID DUMY
 NEW1 is the old program library of COBRA/TRAC. (created in step 4)
 NEW2 is the new program library of COBRA/TRAC (which has no COMDECKs).

6. Add the new COMDECKs to COBRA/TRAC.

UPDATE2(P=NEW2,N=NEW3,I=C,L=0,F)

C is an update input file that adds the new COMDECKs, on TAPE4, to COBRA/TRAC. It consists of 1 line;

*ADDFILE TAPE4,YANK\$\$\$

NEW2 is the old program library of COBRA/TRAC. (created in step 5)

NEW3 is the new program library of COBRA/TRAC (which contains the new COMDECKs).

7. Change the dimensions on the A and ALCM arrays. (This step is optional; if the standard values of the arrays are large enough for the problem at hand, this step can be omitted.)

UPDATE2(P=NEW3,N=NEWPL,I=D,L=0,F)

D is an update input file containing the changes;

*D BLANKCOM.3,4

COMMON A(nnnnn)

INTEGER IA(nnnnn)

(The A and IA array must be dimensioned to the same value, since they are equivalenced.)

*D BLKDAT.55

DATA LAST/nnnnn/

(The variable LAST defines the last element of the A array. It must be set to the value the A array is dimensioned to.)

*D LCMSPACE.3,4

COMMON/LCMSP/ALCM(mmmmm)

DIMENSION ILCM(mmmmm)

(The ALCM and ILCM array are equivalenced.)

*D BLKDAT.54

DATA LLAST/mmmmm/

(The variable LLAST defines the last element in the ALCM array. It must be set to the value the ALCM array is dimensioned to.)

(The file D may contain other changes to the code, also.)

NEW3 is the old program library of COBRA/TRAC.

NEWPL is the new program library of COBRA/TRAC, (which contains the new COMDECKs, the new A and ALCM arrays, and any other changes included in file D.)

8. Compile the new program library (NEW3 or NEWPL, depending on whether or not step 7 was executed).

(If step 7 was skipped, change the name of file NEW3 to NEWPL.)

```
UPDATE2(P=NEWPL,I=HOPE1,C,L=0)
FTN(I=COMPILE,ROUND=0,B=TLIB1,SL=0,R=0)
UPDATE2(P=NEWPL,I=HOPE2,C,L=0)
FTN(I=COMPILE,ROUND=0,B=TLIB2,SL=0,R=0)
UPDATE2(P=NEWPL,I=HOPE3,C,L=0)
FTN(I=COMPILE,ROUND=0,B=TLIB3,SL=0,R=0)
FTN(I,CSIZE=1300000,SS=400,CNAME=COBRAB,LF=LF12,&
CR=0,SL=0,GLIB=TLIB1,GLIB=TLIB2,GLIB=TLIB3)
```

HOPE1, HOPE2 and HOPE3 each contain update compilation directives for about 1/3 of the subroutines in the code.

TLIB1, TLIB2 and TLIB3 are FTN libraries each containing the compiled subroutines of the code defined by HOPE1, HOPE2 and HOPE3, respectively.

LF12 contains the overlay structure.

COBRAB is the executable binary file.

The above described JCL is for the LANL LTSS system. Non LTSS users will have to modify the above procedure to be consistent with their operating system.

Card SPEC.3 is repeated until all 41 dimension parameters have been specified. The SPECSET program will fail if even one has been omitted. The parameters can be entered in any order, as long as none are left out. The equal sign between MM and IVAL is not read by the code, but it helps clarify the input for the user. On restarts SPECSET parameters may not be changed from those used to create the restart dump, otherwise, the dump will be read improperly.

TABLE 2.1. SPECSET Dimensioning Parameters

Parameter	Definition
MA	Maximum number of pairs of entries for the vertical variation tables. Each table will be dimensioned for MA pairs of elements. (This parameter should be equal to or greater than the maximum value of NAXL(I), entered in group 5 on card VESSEL5.2.)
MB	Maximum number of channels per section. (This parameter should be equal to or greater than the maximum value for NCHN, entered in group 4 on card VESSEL4.2.)
MC	Maximum number of channels. (This parameter should be equal to or greater than the maximum value of I, entered in group 2 on card VESSEL2.2).
MD	Number of records read to describe transverse orthogonal convection. (This parameter should be equal to or greater than NLMGAP, entered in group 3 on card VESSEL3.4.)
ME ^(a)	Maximum number of connections to one-dimensional components at a given level in the VESSEL. (This parameter is not checked directly by the code. The user must check it against his input on card COBRA.5.)
MH	Number of components dumped for graphics.
MI	Maximum number of graphics dumps to be made.
M5	Number of variables dumped for short graphics dump. (This parameter should be greater than or equal to IGRFOP input on card GRAF.2.)
M6 ^(a)	Vessel loop connection logic parameter. It must be equal to or greater than the value obtained from the following.

(a) Care should be used when selecting this parameter since it may increase in a subsequent restart calculation. The maximum value expected for the entire transient should be specified.

TABLE 2.1. (continued)

Parameter	Definition
	$M6 > NVC \cdot (2 \cdot MVCL + 2)$ <p>where NVC = total number of vessel connections (specified by input on line COBRA.4).</p> <p>MCVL = the maximum number of vessel connections in any one loop, (where a loop is defined as a flow path through a series of one-dimensional components that exits the vessel at one junction and returns to the vessel through at least one other junction or ends at a break or fill.)</p>
M8(a)	Maximum number of junctions. (This parameter should be equal to or greater than NJUN, entered on card INPUT.4.)
MF	Maximum number of fuel geometry types. (This parameter should be equal to or greater than NFUELT, entered in group 9 on card VESSEL9.1.)
MG	Maximum number of transverse connections between channels (i.e., gaps). (This parameter should be equal to or greater than the maximum value of NK, entered in group 3 on card VESSEL3.2.)
ML	Maximum number of vertical variation tables. (This parameter should be equal to or greater than NAFAC entered in group 5 on card VESSEL5.1.)
MR	Maximum number of rods. (This parameter should be equal to or greater than NROD, entered in group 8 on card VESSEL8.1.)
MS	Maximum number of pressure sink and mass source boundary conditions (i.e., up to MS type 4 boundary conditions and up to MS type 5 boundary conditions can be specified in input group 13.) (This parameter should be equal to or greater than the maximum number of either type 4 or type 5 boundary conditions specified in group 13 on card VESSEL13.4.)
MT	Maximum number of material property types. (This parameter should be equal to or greater than NMAT, entered in group 10 on card VESSEL10.1.)

TABLE 2.1. (continued)

Parameter	Definition
MU	Maximum number of specified vertical boundary conditions. (This parameter should be equal to or greater than NIBND, entered on card VESSEL13.1.)
MV	Maximum number of forcing functions on the specified cell boundary conditions. (This parameter should be equal to or greater than NFUNCT on card VESSEL13.1.)
MX	Maximum number of vertical nodes in a section. (This parameter should be equal to or greater than the maximum value of NONODE+2, entered in group 4 on card VESSEL4.2.)
MY	Maximum number of locations plus 1, in any one channel, where vertical velocities convect transverse momentum across section boundaries. (This parameter should be at least 1 greater than the maximum value of NAMGAP, entered in group 2 on card VESSEL2.2.)
MZ	Maximum number of liquid level records to be read by input line GRAF.3. This is the maximum number of groups of channels that can be considered in the liquid level graphics calculation.
NA	Maximum number of continuity cells. This parameter should be equal to or greater than the sum of all nodes in all channels; $\text{NSECTS} = \sum_{N=1} \text{NCHN}(N) * \text{NONODE}(N)$ <p>where NSECTS = total number of sections (entered on card VESSEL4.1.) NCHN(N) = number of channels in section N, (entered on card VESSEL4.2.) NONODE(N) = number of vertical levels in section N, (entered on card VESSEL4.2.)</p>
NB	Maximum number of flow connections to a cell, (including all connections for vertical flow and transverse flow).

TABLE 2.1. (continued)

Parameter	Definition
NE	Maximum bandwidth of pressure solution matrix. Bandwidth for a given problem is $(2 \cdot IWIDE + 1)$, where IWIDE is specified in group 4 on card VESSEL4.4. NE must be equal to or greater than $(2 \cdot IWIDE + 1)$.
NF	Maximum number of pairs of entries in tables of forcing functions on boundary conditions, rod power and gap conductance. Each forcing function may have up to NF pairs of entries. (This parameter should be equal to or greater than the maximum of NPTS(K), entered on card VESSEL13.2, and NQ and NGPFF, entered on card VESSEL11.1.)
NI	Maximum number of cells in any one simultaneous solution group. (This parameter should be equal to or greater than the maximum difference between adjacent elements of the MSIM(I) array, entered in group 4 on card VESSEL4.5.)
NL	Maximum number of transverse momentum cells that have crossflow set to zero. (This parameter should be equal to or greater than NKBND, entered in group 13 on card VESSEL13.1.)
NN	Maximum number of radial nodes for conduction in a rod. (This parameter should be at least 2 greater than the maximum value of NFUEL, entered in group 9 on card VESSEL9.2 and greater than the number of nodes specified for a nonnuclear geometry type.)
NP	Maximum number of entries in a material property table. (This parameter should be equal to or greater than the maximum value of NNTDP, entered in group 10 on card VESSEL10.2.)
NQ	Maximum number of sections. (This parameter should be equal to or greater than NSECTS, entered in group 4 on card VESSEL4.1.)
NR	Maximum number of heat transfer surfaces on all rods, including both internal and external surfaces in contact with the fluid. (This parameter is not checked directly in the code. The user must check it against his input for the LR array in group 2 on card VESSEL2.4.)

TABLE 2.1. (continued)

Parameter	Definition
NS ^(a)	Maximum number of junction connections to one-dimensional components. (This parameter should be equal to or greater than NCSR, entered on card COBRA.4.)
NT	Maximum number of unheated conductors. (This parameter should be equal to or greater than NSROD, entered in group 8 on card VESSEL8.1.)
NV	Maximum number of heat transfer surfaces on all unheated conductors, including both internal and external surfaces in contact with the fluid. (This parameter is not checked directly in the code. The user must check it against his input for the LS array in group 2 on card VESSEL2.4. NV must be equal to or greater than the maximum number of nonzero entries in the LS array.)
NX	Maximum number of axial nodes in rods, including those created by fine mesh renoding. (Recommended value is 80 if renoding is used.)
NY	Maximum number of radial nodes for conduction in an unheated conductor. This parameter should be equal to or greater than the maximum value of the sum of all nodes in a non-nuclear geometry type.
	$\left[\sum_{L=1}^{NFUEL(I)} NODER(L) \right]_{\max}$
	where NFUEL(I) = number of regions in geometry type I (entered on card VESSEL9.6).
	NODER(L) = number of nodes in region L of geometry type I, (entered on card VESSEL9.7).
NZ	Maximum number of vertical levels in the VESSEL. (This parameter should be at least 2 greater than NDX, entered in group 1 on card VESSEL1.1.)
N3	Maximum number of axial power profile tables. (This parameter should be equal to or greater than NAXP, entered in group 11 on card VESSEL11.1.)

TABLE 2.1. (continued)

Parameter	Definition
N4	Maximum number of rod or unheated conductor temperature initialization tables. (This parameter should be equal to or greater than NRTAB, entered in group 8 on card VESSEL8.1.)
N5	Maximum number of entries in a given rod or unheated conductor temperature initialization table. (This parameter should be equal to or greater than the maximum value of NRAX1, entered in group 8 on card VESSEL8.4.)
N8	Maximum number of nuclear fuel rods using the dynamic gap conductance model. (This parameter is not checked directly in the code. The user must check it against his input in group 8 on card VESSEL8.2 by summing the number of heater rods using the dynamic gap conductance model.)
(b) NC	$2 * NN$
ND	$25 + 5 * NB$
NG	$NN + 2$
NH	$NX + 2$
NJ	$NB + 1$
NK	$NB + 5$

(b) The parameters NC through NK are dimension parameters calculated internally by SPECSET, based on parameter values input by the user. These parameters are not specified directly in the input for SPECSET. They are listed here only to inform the user that these parameters are used in the common blocks to define certain arrays sizes.

TABLE 2.2. Sample Input File TAPE2 for SPECSET Program

LCM
0
MA= 3
MB= 44
MC= 282
MD= 272
ME= 8
MF= 11
MG= 313
ML= 64
MR= 10
MS= 1
MT= 2
MU= 117
MV= 1
MY= 9
MX= 5
NT= 249
NV= 383
NY= 6
N3= 2
N4= 1
N5= 3
N8= 10
NA= 556
NB= 8
NE= 51
NF= 90
NI= 54
NL= 80
NN= 8
NP= 8
NQ= 10
NR= 10
NS= 12
MZ= 85
NX= 100
NV= 56
NZ= 22
MH= 10
MI=1000
M5= 950
M6= 120
M8= 58

2.6 GRAFIX POSTPROCESSOR PROGRAM

This section describes the input requirements of the GRAFIX postprocessor program. GRAFIX is an interactive program that reads the graphics dump file TAPE11^(a) created by COBRA/TRAC and, under guidance from the user, translates the data into a plot file. This file contains the plot coordinates, axes labels, optional measured data coordinates and calculated data. The GRAFIX program does not produce the plots; instead, it produces a file that can be read by a plotting program (NEWPLOT) using CALCOMP plot commands for final plotting. The Programmer's Manual (Volume 5 of this document) explains how GRAFIX and NEWPLOT work and should be consulted when modifying the COBRA/TRAC code package for a particular computer system.

GRAFIX can prepare data for three types of plots:

- variable versus time
- variable versus vertical level (in the VESSEL component only)
- rod temperature versus radial location at specific elevations and at specified times (for the VESSEL component only)

The user can specify up to five separate calculated data curves on any one plot, plus optional measured data. The user can control the physical size of the plot, the scaling of the data, and the axis labels. GRAFIX will also perform some simple unit conversions on the calculated data if requested.

GRAFIX can be run in two different modes: 1) interactively; prompting the user with questions and waiting for answers from the keyboard, or 2) "batch" mode, using a command file or data saved from a previous interactive session for input. In the interactive mode, the user's responses are entered in unformatted "free-field" mode. This eliminates the need to place the entries in specific column positions. Multiple entries on a line are separated by commas. The only restriction is that the user must include a decimal point for real variables.

(a) See Section 2.3, (Graphics Options input), and Section 3.4.2 for a discussion of how to save graphics data during COBRA/TRAC simulations.

If the the user does not wish to be troubled with entering plot labels, GRAFIX will write three labels on the plot according to what it knows about the variables being plotted. These labels are necessarily somewhat vague, and for report-quality graphics, the user should enter his own labels. The default labels are useful, however, for quick looks at the data.

Plot size and label locations are specified in inches. The origin of the axes is the (0.0 in., 0.0 in.) coordinate of the plot. Label positions are specified by an (x,y) coordinate at which the label is to start. (This coordinate is the position of the lower left corner of the first character of the label.) The size of the label, also in inches, defines the height of the characters in the label. Labels may be rotated by specifying the angle of rotation, in degrees. (Usually, x-axis labels are rotated 0.0 degrees, y-axis labels are rotated 90.0 degrees, but any rotation is acceptable.) The length of each label can be determined by the following formula:

$$L = \frac{6}{7} * H * N$$

where L = length, in inches

H = height of the label in inches

N = number of characters in the label

Using this formula, it is possible to define a starting location for each label that will place it exactly where the user wants it.

If default scaling is used, GRAFIX will search for the minimum and maximum values and scale the data accordingly. This will not generally produce elegantly partitioned axes, but is useful for scanning the data. The data to be plotted may also be scaled to fit between user-specified minima and maxima for the x and y axes, for report quality plots. Unless a conversion factor is being used, the data will be plotted as found in the graphics dump file.

Because GRAFIX is normally run interactively, it is not possible to list the line-by-line input instructions in quite the same manner as for the other programs in the COBRA/TRAC package. Instead, a typical interactive session is given below. This session uses most of the current GRAFIX options.

HELLO THERE! GRAFIX IS AT YOUR SERVICE

THE DATA FILE WAS LAST WRITTEN ON 82/08/26. 08:01:34.
THE FILE CONTAINS 19 DUMPS FOR 10 COMPONENTS
THE COMPONENTS ARE:

NO.	TYPE	TIME FIRST DATA WAS SAVED
1	BREAK	15.0178
2	PIPE	.0001
3	VESSEL	.0001

IF YOU WISH TO ENTER OPTIONS INTERACTIVELY
ENTER 0 (ZERO). IF YOU WANT TO USE INTERACTIVE
DATA SAVED FROM AN EARLIER RUN (TAPE3) ENTER 1

? 0

ENTER THE COMPONENT NUMBER TO BE PLOTTED
ENTER 0 (ZERO) TO TERMINATE THIS RUN

? 3

YOU HAVE SELECTED THE VESSEL

ENTER 1 FOR TIME PLOTS
ENTER 2 FOR AXIAL PLOTS
ENTER 3 FOR RADIAL PLOTS

? 1

YOU HAVE SELECTED A TIME PLOT.
THE FOLLOWING CALCULATIONS OR CONVERSIONS ARE
CURRENTLY AVAILABLE:

1 = A * FACTOR
2 = (A + B) * FACTOR
3 = (A - B) * FACTOR
4 = (A + C1) * C2 + FACTOR

ENTER YOUR CHOICE

? 1

CHOICES 1 OR 4 MAY BE AVERAGED OVER
AXIAL RANGE. ENTER 1 TO AVERAGE; 0 OTHERWISE

? 0

ENTER THE NUMBER OF CURVES FOR THIS PLOT.
(MIN.=1;MAX.=5)

? 2

THE FOLLOWING VARIABLES ARE POSSIBLE SELECTIONS
FOR 'A' OR 'B'

1=LIQUID FRACTION	21=XKGE(I,J)
2=VAPOR FRACTION	22=SEENT(I,J)
3=ENTRAINED LIQUID FRACTION	23=SENT(I,J)
4=LIQUID FLOW (CONTINUITY)	24=IFIJ(I,J)
5=VAPOR FLOW (CONTINUITY)	25=HTCL(J,N)
6=ENTRAINED FLOW (CONTINUITY)	26=HTCV(J,N)
7=LIQUID FLOW (MOMENTUM)	27=PROD(J,N)
8=VAPOR FLOW (MOMENTUM)	28=MODE(J,N)
9=ENTRAINED FLOW (MOMENTUM)	29=ROD TEMPERATURE
10=PRESSURE	30=LIQUID CROSSFLOW (CONTINUITY)
11=LIQUID TEMPERATURE	31=VAPOR CROSSFLOW (CONTINUITY)
12=VAPOR TEMPERATURE	32=ENTRAINED CROSSFLOW (CONTINUITY)
13=LIQUID DENSITY	33=HTCLS(J,N)
14=VAPOR DENSITY	34=HTCVS(J,N)
15=HASHL(I,J)	35=MODES(J,N)
16=HASHV(I,J)	36=TLIDS(J,N)
17=GAMA(I,J)	37=TVAFS(J,N)
18=HASCL(I,J)	38=TTSTR(I,J,N)
19=HASCV(I,J)	
20=XK(I,J)	

ENTER SELECTIONS FOR A,B,C1,C2,FACTOR
-- 0 IF NOT USED FORMAT IS (2I5,3F10)

? 10,0,0,0,1,1,

ENTER SHIFT FOR TIME AXIS: 0 IF NOT NEEDED

? 0,

ENTER CHANNEL: ROD OR GAP NUMBER FOR 'A' - CURVE 1

? 1

ENTER NODE NO. (AXIAL FOR CHANNELS OR GAPS)
RADIAL FOR ROD TEMPERATURES: 0 OTHERWISE)

? 5

ENTER CHANNEL: ROD OR GAP NUMBER FOR 'A' - CURVE 2

? 1

ENTER NODE NO. (AXIAL FOR CHANNELS OR GAPS)
RADIAL FOR ROD TEMPERATURES: 0 OTHERWISE)

? 8

DO YOU WANT MEASURED DATA ON THIS PLOT?

? NO

NOW DESCRIBE THE PHYSICAL CHARACTERISTICS OF THE PLOT

DEFAULT SIZE IS 6.0 IN (X) BY 5.0 IN (Y)

DEFAULT LABELS ARE MINIMAL DESCRIPTIONS OF THE AXES

DEFAULT SCALING USES MINS. AND MAXS. ENCOUNTERED IN THE DATA

ENTER 0 TO USE THESE DEFAULTS: 1 TO DESCRIBE PLOT FULLY

? 0

ENTER THE COMPONENT NUMBER TO BE PLOTTED

ENTER 0 (ZERO) TO TERMINATE THIS RUN

? 3

YOU HAVE SELECTED THE VESSEL

ENTER 1 FOR TIME PLOTS
ENTER 2 FOR AXIAL PLOTS
ENTER 3 FOR RADIAL PLOTS

? 1

YOU HAVE SELECTED A TIME PLOT.
THE FOLLOWING CALCULATIONS OR CONVERSIONS ARE
CURRENTLY AVAILABLE:

1 = A * FACTOR
2 = (A + B) * FACTOR
3 = (A - B) * FACTOR
4 = (A + C1) * C2 + FACTOR

ENTER YOUR CHOICE

? 2

ENTER THE NUMBER OF CURVES FOR THIS PLOT.
(MIN.=1;MAX.=5)

? 1

THE FOLLOWING VARIABLES ARE POSSIBLE SELECTIONS
FOR 'A' OR 'B'

1=LIQUID FRACTION	21=XKGE(I,J)
2=VAPOR FRACTION	22=SDENT(I,J)
3=ENTRAINED LIQUID FRACTION	23=SENT(I,J)
4=LIQUID FLOW (CONTINUITY)	24=ISIJ(I,J)
5=VAPOR FLOW (CONTINUITY)	25=HTCL(JX,N)
6=ENTRAINED FLOW (CONTINUITY)	26=HTCV(JX,N)
7=LIQUID FLOW (MOMENTUM)	27=OROD(JX,N)
8=VAPOR FLOW (MOMENTUM)	28=MODE(JX,N)
9=ENTRAINED FLOW (MOMENTUM)	29=ROD TEMPERATURE
10=PRESSURE	30=LIQUID CROSSFLOW (CONTINUITY)
11=LIQUID TEMPERATURE	31=VAPOR CROSSFLOW (CONTINUITY)
12=VAPOR TEMPERATURE	32=ENTRAINED CROSSFLOW (CONTINUITY)
13=LIQUID DENSITY	33=HTCLS(J,N)
14=VAPOR DENSITY	34=HTCVS(J,N)
15=HASHL(I,J)	35=MODES(J,N)
16=HASHV(I,J)	36=TLIDS(J,N)
17=GAMA(I,J)	37=TVAPS(J,N)
18=HASCL(I,J)	38=TTSTR(I,J,N)
19=HASCV(I,J)	
20=XK(I,J)	

ENTER SELECTIONS FOR A,B,C1,C2,FACTOR
-- 0 IF NOT USED FORMAT IS (2I5,3F10)

? 10,10,0.,0.,0.5

ENTER SHIFT FOR TIME AXIS; 0 IF NOT NEEDED
 0.,
 ENTER CHANNEL; ROD OR GAP NUMBER FOR 'A' - CURVE 1
 ? 1
 ENTER NODE NO. (AXIAL FOR CHANNELS OR GAPS)
 RADIAL FOR ROD TEMPERATURES; 0 OTHERWISE)
 ? 5
 ENTER CHANNEL; ROD OR GAP NUMBER FOR 'B' - CURVE 1
 ? 1
 ENTER NODE NO. (AXIAL FOR CHANNELS OR GAPS)
 RADIAL FOR ROD TEMPERATURES; 0 OTHERWISE)
 ? 8
 DO YOU WANT MEASURED DATA ON THIS PLOT?
 ? NO
 NOW DESCRIBE THE PHYSICAL CHARACTERISTICS OF THE PLOT
 DEFAULT SIZE IS 6.0 IN (X) BY 5.0 IN (Y)
 DEFAULT LABELS ARE MINIMAL DESCRIPTIONS OF THE AXES
 DEFAULT SCALING USES MINS. AND MAXS. ENCOUNTERED IN THE DATA
 ENTER 0 TO USE THESE DEFAULTS; 1 TO DESCRIBE PLOT FULLY
 ? 0
 ENTER THE COMPONENT NUMBER TO BE PLOTTED
 ENTER 0 (ZERO) TO TERMINATE THIS RUN
 ? 1
 YOU HAVE SELECTED A ONE-D COMPONENT.
 CURRENTLY ONLY TIME PLOTS ARE AVAILABLE.
 THE FOLLOWING COMPUTATIONS OR CONVERSIONS
 MAY BE SELECTED
 1=A * FACTOR
 2=(A + B) * FACTOR
 3=(A - B) * FACTOR
 4=(A + C1) * C2 + FACTOR
 ENTER YOUR CHOICE
 ? 1
 THE FOLLOWING VARIABLES ARE POSSIBLE SELECTIONS
 FOR A AND B
 1=VAPOR FRACTION
 2=PRESSURE
 3=LIQUID DENSITY
 4=VAPOR DENSITY
 5=LIQUID TEMPERATURE
 6=VAPOR TEMPERATURE
 7=LIQUID VELOCITY
 8=VAPOR VELOCITY
 9=MIXTURE VELOCITY
 10=LIQUID FLOW
 11=VAPOR FLOW
 ENTER SELECTIONS FOR A,B,C1,C2,FACTOR
 (0 IF NOT USED). FORMAT IS (2I5,3F10)
 ? 2,0,0.,0.,1.,

ENTER SHIFT FOR TIME AXIS: 0 IF NOT NEEDED
 ? 0.
 ENTER THE NUMBER OF CURVES FOR THIS PLOT
 (MIN.=1; MAX.=5)
 ? 1
 ENTER CELL NUMBER FOR A - CURVE 1
 ? 2
 DO YOU WANT MEASURED DATA ON THIS PLOT?
 ? NO
 NOW DESCRIBE THE PHYSICAL CHARACTERISTICS OF THE PLOT

 DEFAULT SIZE IS 6.0 IN (X) BY 5.0 IN (Y)
 DEFAULT LABELS ARE MINIMAL DESCRIPTIONS OF THE AXES
 DEFAULT SCALING USES MINS. AND MAXS. ENCOUNTERED IN THE DATA

 ENTER 0 TO USE THESE DEFAULTS; 1 TO DESCRIBE PLOT FULLY
 ? 0
 ENTER THE COMPONENT NUMBER TO BE PLOTTED
 ENTER 0 (ZERO) TO TERMINATE THIS RUN
 ? 2
 YOU HAVE SELECTED A ONE-D COMPONENT.
 CURRENTLY ONLY TIME PLOTS ARE AVAILABLE.
 THE FOLLOWING COMPUTATIONS OR CONVERSIONS
 MAY BE SELECTED

 1=A * FACTOR
 2=(A + B) * FACTOR
 3=(A - B) * FACTOR
 4=(A + C1) * C2 + FACTOR

 ENTER YOUR CHOICE
 ? 1
 THE FOLLOWING VARIABLES ARE POSSIBLE SELECTIONS
 FOR A AND B

 1=VAPOR FRACTION 11=VAPOR FLOW
 2=PRESSURE
 3=LIQUID DENSITY
 4=VAPOR DENSITY
 5=LIQUID TEMPERATURE
 6=VAPOR TEMPERATURE
 7=LIQUID VELOCITY
 8=VAPOR VELOCITY
 9=MIXTURE VELOCITY
 10=LIQUID FLOW

 ENTER SELECTIONS FOR A;B;C1;C2;FACTOR
 (0 IF NOT USED). FORMAT IS (2I5,3F10)
 ? 2:0,0,0,0,1,
 ENTER SHIFT FOR TIME AXIS: 0 IF NOT NEEDED
 ? 0.
 ENTER THE NUMBER OF CURVES FOR THIS PLOT
 (MIN.=1; MAX.=5)
 ? 1

ENTER CELL NUMBER FOR A - CURVE 1

1 DO YOU WANT MEASURED DATA ON THIS PLOT?

? NO NOW DESCRIBE THE PHYSICAL CHARACTERISTICS OF THE PLOT

DEFAULT SIZE IS 6.0 IN (X) BY 5.0 IN (Y)

DEFAULT LABELS ARE MINIMAL DESCRIPTIONS OF THE AXES

DEFAULT SCALING USES MIN. AND MAX. ENCOUNTERED IN THE DATA

ENTER 0 TO USE THESE DEFAULTS; 1 TO DESCRIBE PLOT FULLY

? 0 ENTER THE COMPONENT NUMBER TO BE PLOTTED

ENTER 0 (ZERO) TO TERMINATE THIS RUN

? 3 YOU HAVE SELECTED THE VESSEL

ENTER 1 FOR TIME PLOTS

ENTER 2 FOR AXIAL PLOTS

ENTER 3 FOR RADIAL PLOTS

? 2 YOU HAVE SELECTED AN AXIAL PLOT.

THE FOLLOWING CALCULATIONS OR CONVERSIONS ARE CURRENTLY AVAILABLE

1=A * FACTOR

2=(A + B) * FACTOR

3=(A - B) * FACTOR

4=(A + C1) * C2 + FACTOR

ENTER YOUR CHOICE

? 1 ENTER THE NUMBER OF CURVES FOR THIS PLOT.

(MIN.=1;MAX.=5)

? 1 THE FOLLOWING VARIABLES ARE POSSIBLE SELECTIONS FOR 'A' OR 'B'

1=LIQUID FRACTION	21=XKGE(I,J)
2=VAPOR FRACTION	22=SDENT(I,J)
3=ENTRAINED LIQUID FRACTION	23=SENT(I,J)
4=LIQUID FLOW (CONTINUITY)	24=ISEIJ(I,J)
5=VAPOR FLOW (CONTINUITY)	25=HTCL(J,N)
6=ENTRAINED FLOW (CONTINUITY)	26=HTCV(J,N)
7=LIQUID FLOW (MOMENTUM)	27=PROD(J,N)
8=VAPOR FLOW (MOMENTUM)	28=MODE(J,N)
9=ENTRAINED FLOW (MOMENTUM)	29=ROD TEMPERATURE
10=PRESSURE	30=LIQUID CROSSFLOW (CONTINUITY)
11=LIQUID TEMPERATURE	31=VAPOR CROSSFLOW (CONTINUITY)
12=VAPOR TEMPERATURE	32=ENTRAINED CROSSFLOW (CONTINUITY)
13=LIQUID DENSITY	33=HTCLS(J,N)
14=VAPOR DENSITY	34=HTCVS(J,N)
15=HASHL(I,J)	35=MODES(J,N)
16=HASHV(I,J)	36=TLIDS(J,N)
17=SAMA(I,J)	37=TVAPS(J,N)
18=HASCL(I,J)	38=VSTR(I,J,N)
19=HASCV(I,J)	
20=XK(I,J)	

ENTER SELECTIONS FOR A,B,C1,C2,FACTOR
 -- 0 IF NOT USED FORMAT IS (2I5,3F10)

? 10,0,0.,0.,1.,

ENTER FIRST AND LAST AXIAL NODE NOS. (2I5)

? 2,9,

ENTER CHANNEL, ROD OR GAP NUMBER FOR 'A' - CURVE 1

? 1

ENTER THE TIME INTERVAL (SECONDS)
 AT WHICH TO PRODUCE PLOTS

? 2.,

DO YOU WANT MEASURED DATA ON THIS PLOT?

? NO

NOW DESCRIBE THE PHYSICAL CHARACTERISTICS OF THE PLOT

DEFAULT SIZE IS 6.0 IN (X) BY 5.0 IN (Y)
 DEFAULT LABELS ARE MINIMAL DESCRIPTIONS OF THE AXES
 DEFAULT SCALING USES MINS. AND MAXS. ENCOUNTERED IN THE DATA

ENTER 0 TO USE THESE DEFAULTS, 1 TO DESCRIBE PLOT FULLY

? 0

ENTER THE COMPONENT NUMBER TO BE PLOTTED
 ENTER 0 (ZERO) TO TERMINATE THIS RUN

? 3

YOU HAVE SELECTED THE VESSEL

ENTER 1 FOR TIME PLOTS
 ENTER 2 FOR AXIAL PLOTS
 ENTER 3 FOR RADIAL PLOTS

? 3

YOU HAVE SELECTED A RADIAL PLOT OF ROD TEMPERATURE
 THE FOLLOWING CALCULATIONS OR CONVERSIONS ARE
 CURRENTLY AVAILABLE

$1 = A * \text{FACTOR}$
 $2 = (A + C1) * C2 + \text{FACTOR}$

ENTER YOUR CHOICE

? 1

ENTER 0 IF NORMAL ROD, 1 IF SLAB ROD

? 0

ENTER C1,C2 AND FACTOR (0 IF NOT USED)
 FORMAT IS (3F10)

? 0.,0.,1.,

ENTER THE NUMBER OF CURVES FOR THIS PLOT.
 (MIN.=1,MAX.=5)

? 2

ENTER ROD NUMBER - CURVE 1

? 1

ENTER AXIAL ELEVATION (INCHES)

? 150.,

ENTER ROD NUMBER - CURVE 2

? 1

```

        ENTER AXIAL ELEVATION (INCHES)
? 210.
    ENTER THE TIME INTERVAL (SECONDS)
    AT WHICH TO PRODUCE PLOTS
? 4.
    DO YOU WANT MEASURED DATA ON THIS PLOT?
? NO
    NOW DESCRIBE THE PHYSICAL CHARACTERISTICS OF THE PLOT

    DEFAULT SIZE IS 6.0 IN (X) BY 5.0 IN (Y)
    DEFAULT LABELS ARE MINIMAL DESCRIPTIONS OF THE AXES
    DEFAULT SCALING USES MIN. AND MAX. ENCOUNTERED IN THE DATA

    ENTER 0 TO USE THESE DEFAULTS; 1 TO DESCRIBE PLOT FULLY
? 1
    ENTER XSIZE*YSIZE*XMIN*XMAX*YMIN*YMAX

    WHERE XSIZE IS LENGTH OF X (HORIZONTAL) AXIS
          YSIZE IS LENGTH OF Y (VERTICAL) AXIS
          XMIN AND YMIN ARE MINIMUM DATA VALUES
          XMAX AND YMAX ARE MAXIMUM DATA VALUES

    FORMAT IS (6F10)
? 5.0*5.0*0.0*25.0*0.0*1000.
    ENTER THE NUMBER OF LABELS (AXIAL LABELS,TITLES,ETC.)
    MAXIMUM IS 10

? 2
    FOR EACH LABEL, ENTER XL*YL (IN INCHES ON THE PLOT)
    SIZE (INCHES HIGH), ROTATION (IN DEG.), AND THE LABEL
    FORMAT IS (4F10,4A10)
    LABEL 1
? 1.0*-1.0*0.10*0.0; THIS IS THE XAXIS LABEL
    LABEL 2
? -1.0*1.0*0.10*90.; THIS IS THE YAXIS LABEL
    ENTER THE COMPONENT NUMBER TO BE PLOTTED
    ENTER 0 (ZERO) TO TERMINATE THIS RUN

? 0
    STOP FTN
SE      LTSS TIME      17.223 SECONDS
CPU=    .418          I/O=    8.190          MEM=    8.615

    ALL DONE

```

HELLO THERE! GRAFIX IS AT YOUR SERVICE

THE DATA FILE WAS LAST WRITTEN ON 82/08/26. 08:01:34.
THE FILE CONTAINS 19 DUMPS FOR 10 COMPONENTS
THE COMPONENTS ARE:

NO.	TYPE	TIME FIRST DATA WAS SAVED
1	BREAK	15.0178
2	PIPE	.0001
3	VESSEL	.0001

IF YOU WISH TO ENTER OPTIONS INTERACTIVELY
ENTER 0 (ZERO). IF YOU WANT TO USE INTERACTIVE
DATA SAVED FROM AN EARLIER RUN (TAPE3) ENTER 1

? 1

STOP FTN
GB LTSS TIME 1.904 SECONDS
CPU= .254 I/O= 1.077 MEM= .573
LL DONE

When running GRAFIX interactively, a file named TAPE3 is created which contains a log of the users input responses. This file can be saved and used later in the "batch" mode (as shown above) to recreate the same graphics output. GRAFIX is extremely easy to use. It prompts the user for all data needed, and the proper input for a given option is, in general, self-evident.

2.6.1 Measured Data File

If measured data is to be included in the GRAFIX-generated plots, the user must prepare a file prior to running GRAFIX. This file must have a 5-character alphanumeric name and it must be available for GRAFIX to read while the program is being executed. The required format for a measured data file is shown below.

Record 1

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-5 NPTS		The number of points of measured data in the file. (A point consists of two values; time and data value, or location and data value.)

Record 2, 3, ...

<u>Columns</u>	<u>Variable</u>	<u>Description</u>
1-10, 21-30, 41-50, 61-70,	XM(I)	The x-coordinate in data units, <u>(not inches)</u> , for point I (e.g., time or axial distance)
11-20, 31-30, 51-60, 71-80	YM(I)	The y-coordinate in data units, <u>(not inches)</u> , for point I (e.g., pressure or rod temperature)

There must be four (4) pairs of measured data per record to give a total of NPTS points. (The last record may have fewer than four points, however, if NPTS is not an even multiple of four.)

There is no limit to the number of plots that can be produced during any GRAFIX session. Nor is there any restriction on the order in which the plots are produced; axial profile plots can be freely interspersed with plots of variable versus time or radial profile plots. It should be noted that for each axial or radial profile plot requested, many actual plots may be produced, since these are produced at specified time intervals.

For each plot, the size of the plot, the label information, and the coordinates to be plotted for the curves are written to logical unit 2 (TAPE2). This data is then sent to a mini-computer and plots are produced by a program (NEWPLOT) that reads TAPE2 and interprets the input for a CALCOMP plotter.

3.0 USER'S GUIDE TO COBRA/TRAC

This section is intended to aid the user in learning to set up the input model for COBRA/TRAC. There are three basic tasks the user must perform: 1) setting up the geometry data to describe the system, 2) specifying the fluid conditions and forcing functions to define the state of the system, and 3) running the code and interpreting the output.

The first two tasks are covered in Sections 3.1 through 3.5. The input for the VESSEL component is discussed in detail in Sections 3.1 through 3.4. Section 3.1 covers the VESSEL geometry input data, Section 3.2 the input for the conduction model, Section 3.3 the boundary and initial conditions for the VESSEL component, and Section 3.4 the various output options. The input required for the one-dimensional components is discussed in Section 3.5. The information presented in these subsections is intended to explain, amplify and clarify the line-by-line input for the components, as listed in Section 2.

The third task involves such endeavors as calculating the steady-state initial conditions for the system, using the dump/restart feature to run the calculation in a series of steps, and running the graphics postprocessor program to interpret the calculational results. Sections 3.6 through 3.8 are devoted to explaining procedures for using these capabilities.

3.1 COBRA/TRAC VESSEL COMPONENT GEOMETRY MODEL

The VESSEL component can be used to model flow in almost any vertical one-, two- or three-dimensional component. The exceptions are components such as pumps or pressurizers that have special boundary conditions not included in the VESSEL module. (These components can be modeled with one-dimensional PUMP or PRIZER components.) In addition, the problem must be amenable to solution by the semi-implicit numerical algorithm used in the VESSEL component.

All geometries modeled using a VESSEL component are represented as a matrix of Eulerian mesh cells. The number of mesh cells used depends on the degree of detail required to resolve the flow field, the phenomena being modeled, and practical restrictions such as computing costs and core storage limitations. In two-phase flow, the mesh cell should be large compared to the

characteristic size of the two-phase flow pattern. For example, the mesh cell size should be large relative to the bubble size so that the averaged quantities (bubble size, drag, etc.) used in the calculation will be valid. In slug or film flow, the mesh cell size should be on the order of the hydraulic diameter or larger since the physical models for these flow regimes are based on the physical dimensions of the flow path.

The equations for the flow field in the VESSEL are solved using a staggered difference scheme on the Eulerian mesh. The velocities are obtained at mesh cell faces, and the state variables (e.g., pressure, density, enthalpy and phasic volume fractions) are obtained at the cell center. The mesh cell is characterized by cross-sectional area A , height ΔX , and the width of the connections to adjacent mesh cells, S . The basic mesh cell is shown in Figure 3.1. This cell is the control volume for the scalar continuity and energy equations. The momentum equations are solved on a staggered mesh with the momentum cell centered on the scalar cell face. The momentum cell for vertical velocities is shown in Figure 3.2, and that for transverse velocities in Figure 3.3.

The vessel input has been constructed to allow the user a great deal of flexibility in defining the mesh for the irregular geometries typical of reactor vessel internals. The mesh cells are defined by input in terms of channels. A channel is simply a vertical stack of mesh cells, as illustrated in Figure 3.4. A channel may represent a subchannel between four fuel rods, a lumped region of the core, a segment of the downcomer, or any other vertical flow path appropriate to the geometry being modeled.

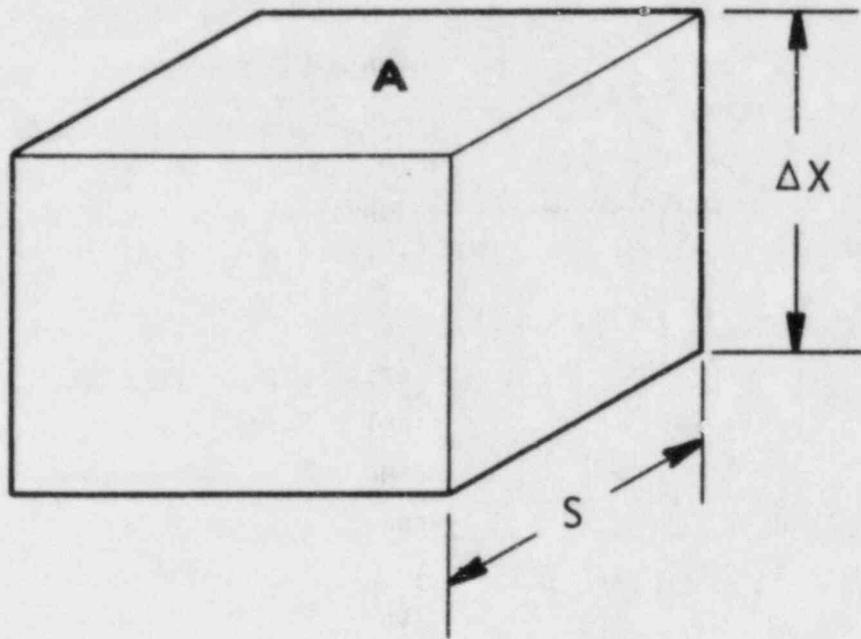


FIGURE 3.1 Basic Mesh Cell

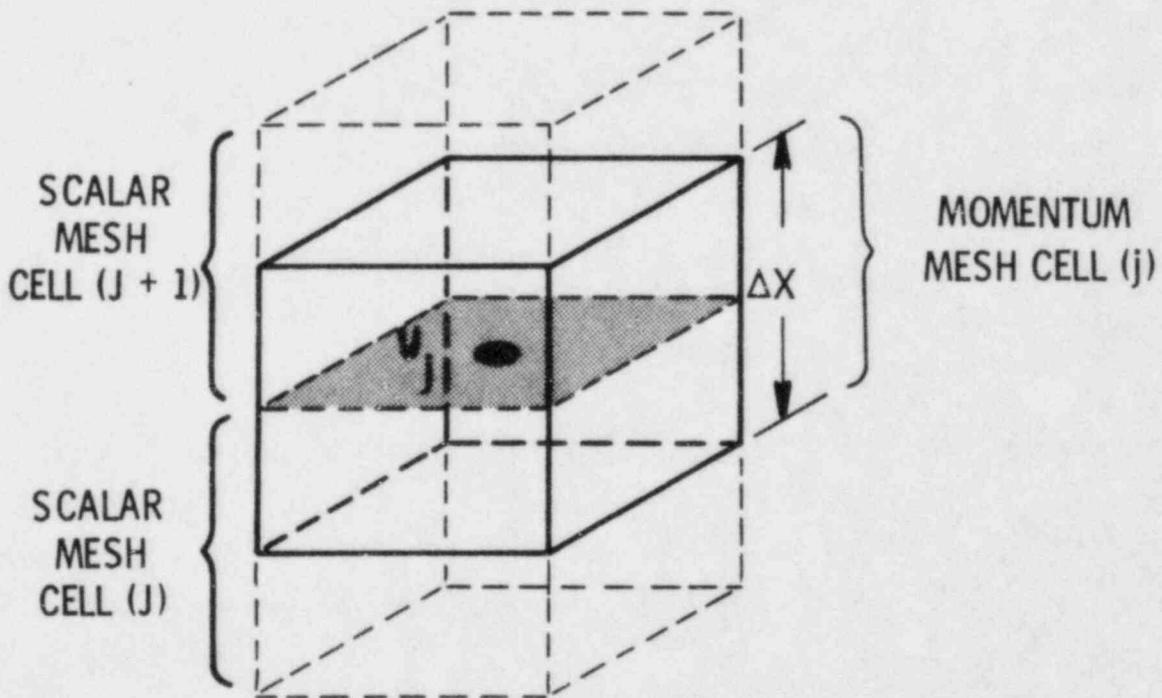


FIGURE 3.2 Mesh Cell for Vertical Velocities

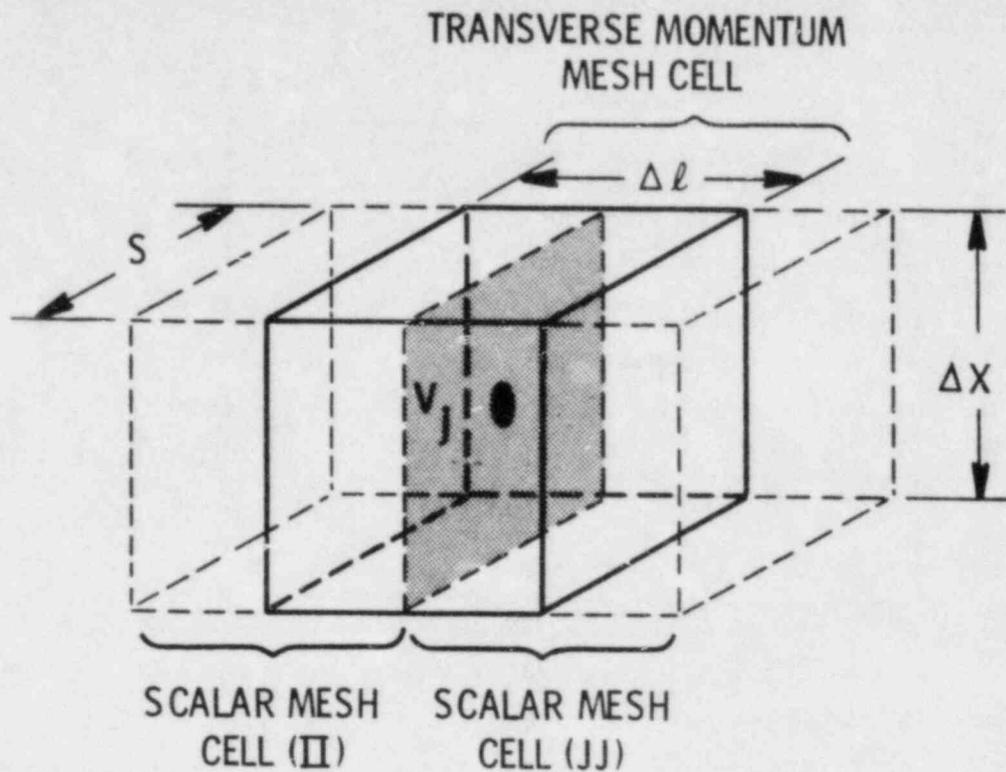


FIGURE 3.3. Mesh Cell for Transverse Velocities

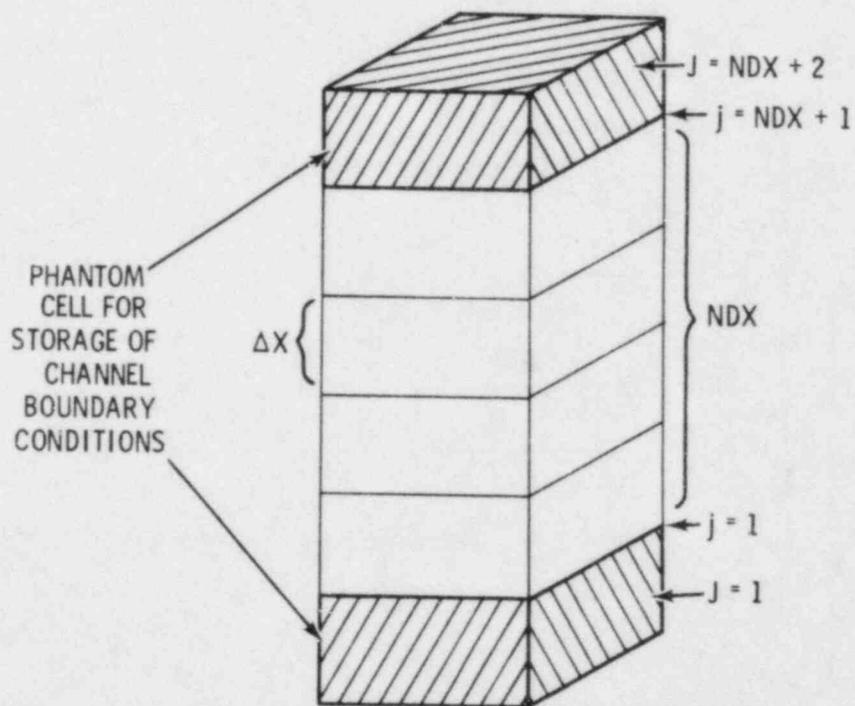


FIGURE 3.4. Basic VESSEL Component Channel

Boundary data for each channel are stored in phantom nodes at its top and bottom. Between these two phantom nodes are NDX nodes that actually enter into the calculation. Node $J = 1$ contains boundary conditions for the bottom of the channel, and node $J = NDX + 2$ contains boundary conditions for the top of the channel. Boundaries between mesh cells are identified in Figure 3.4 with the lower case j and refer to the location of the momentum cell center, where velocities are obtained. The velocity corresponding to $j = 1$ is at the top of the continuity mesh cell corresponding to $J = 1$.

Nominal flow area and wetted perimeter are specified for each channel by input. Each mesh cell within the channel is assumed to have the nominal geometry unless the user specifies a variation for area or wetted perimeter. The mesh for a particular region of the vessel is developed by placing a sufficient number of channels in the region to model the geometry and the important flow phenomena of the region. Transverse connections are specified between the channels to complete the multidimensional mesh for the region. These transverse connections are referred to as gaps. Gaps are defined by the width of the flow path between the two channels and the distance between the channel centers. The width of the gap between channels is assumed to be uniform along the total length of the channels unless an axial variation of the gap is specified. The centroid distance is always equal to the nominal value for the gap; no variation is allowed.

Examples of reactor vessel regions formed by specifying channels and interchannel connections (gaps) are shown in Figure 3.5. A UHI-equipped PWR is used in these examples for illustrative purposes. The core region, shown in Figure 3.5a, is modeled with square channels connected on a rectangular grid. The rectangular grid is appropriate for this region since the fuel rods are arranged in a rectangular array. Only nine channels are used in this case to model the core. Any number of channels can be used to model a region, from a single channel (one-dimensional) representation up to as many channels as computer storage will allow. The appropriate number for any given problem depends on the nature of the problem and the degree of detail the flow phenomena require.

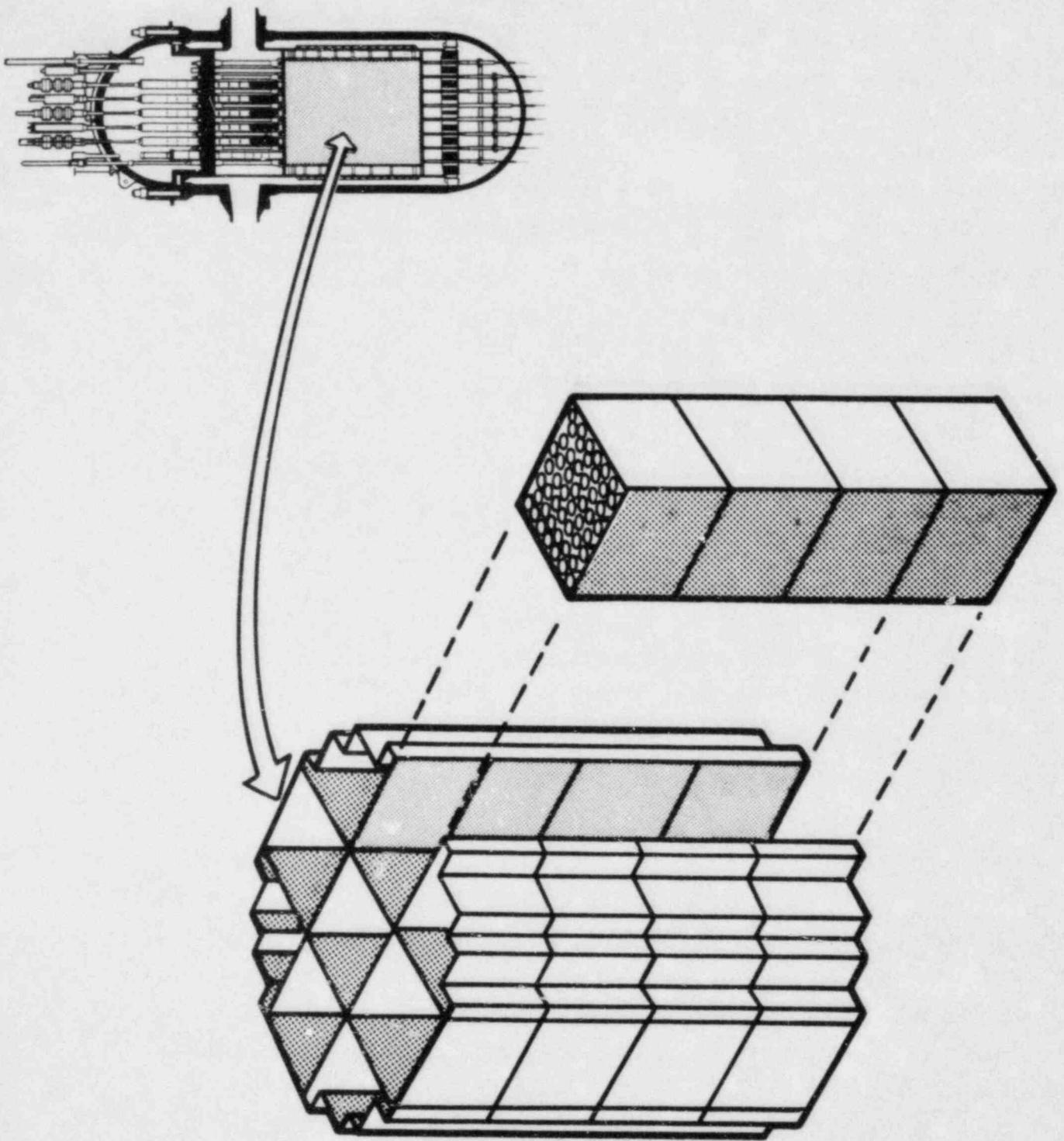


FIGURE 3.5(a). Core Region

The downcomer region, shown in Figure 3.5b, is modeled with a two-dimensional mesh in an annular ring of eight channels. The core bypass region is modeled similarly, as shown in Figure 3.5c. The mesh for the lower plenum, shown in Figure 3.5d, illustrates the use of the variations in channel flow area and gap width to model the curvature of the bottom of the pressure vessel. The model consists of nine channels with axially varying mesh cells.

The mesh for the upper plenum of a UHI plant, shown in Figure 3.5e, consists of nine large channels modeling the open flow area of the plenum and eighteen smaller channels modeling the flow in guide tubes and support columns. This demonstrates the combination of the three-dimensional mesh with a local mesh to model important flow paths that are smaller than the regular mesh. A rectangular mesh similar to that used in the core region is used to represent the open flow area in the upper plenum. Two smaller channels within each of the rectangular channels model the flow paths through the support columns and guide tubes within the region. Flow paths between guide tubes and the open plenum are modeled by transverse connections between the guide tube and upper plenum channels. The support columns are modeled as isolated channels with no transverse connections to the channels of the upper plenum.

The upper head region, shown in Figure 3.5f, is modeled in the same way as the upper plenum except that only guide tube channels are specified within the channel. In addition, small channels are specified in four of the outer channels to represent the jet formed by flow from the four UHI injection nozzles. The curved top of the upper plenum can be modeled by channel area variations in the same way as for the lower plenum.

The meshes developed for each region of the vessel are connected to form a complete model for the reactor vessel. All regions composed of channels of the same vertical length and beginning at the same level in the vessel are grouped together and referred to as a section. For example, the core, core bypass region, and downcomer all occupy the same level, so the channels modeling them form one section of the vessel. This is illustrated in Figure 3.6. If small leakage flow paths exist between any two regions within a section, such as between the downcomer and core bypass region, these leakage paths can be modeled by specifying transverse connections between channels in

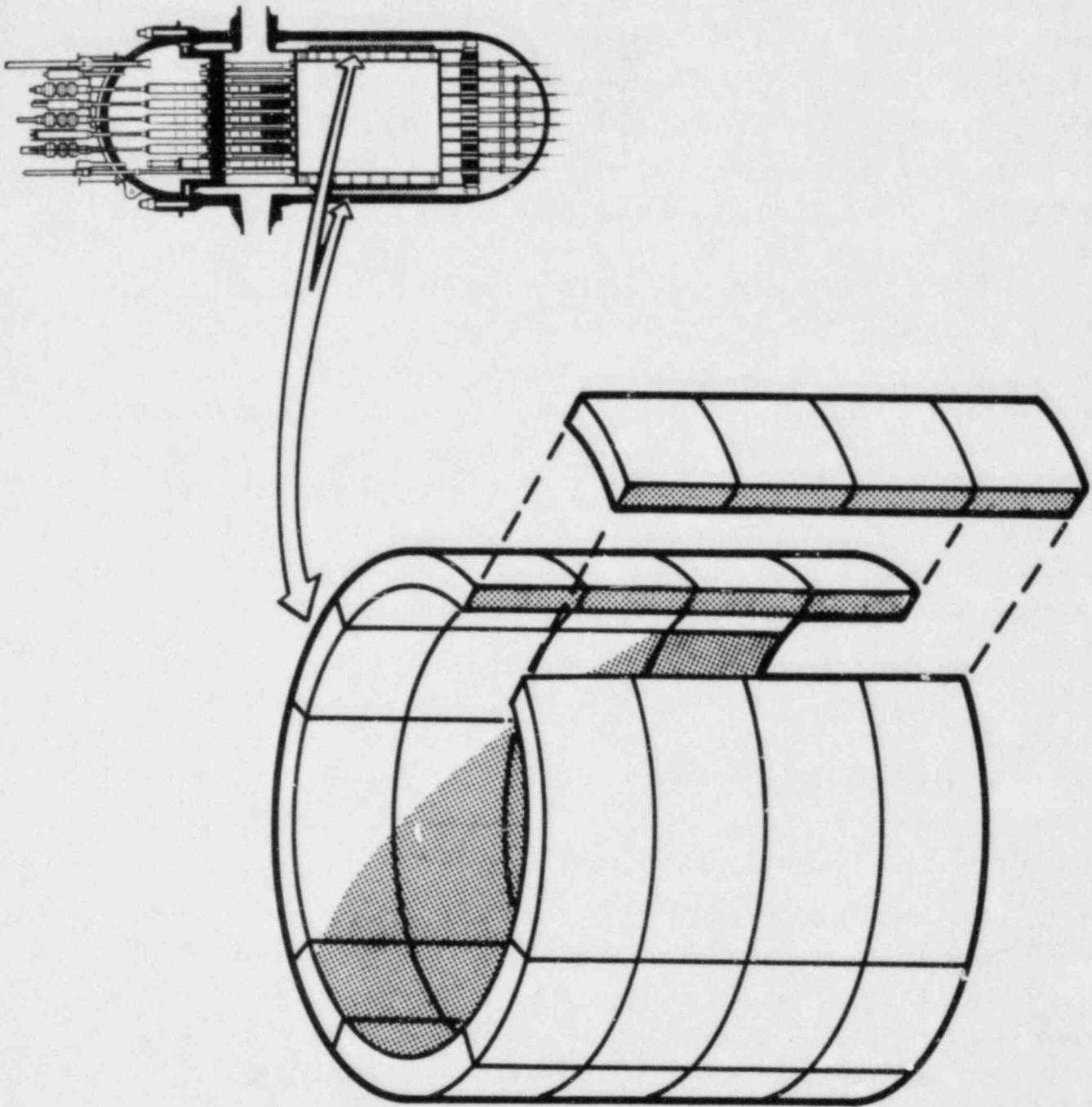


FIGURE 3.5(b). Downcomer Region

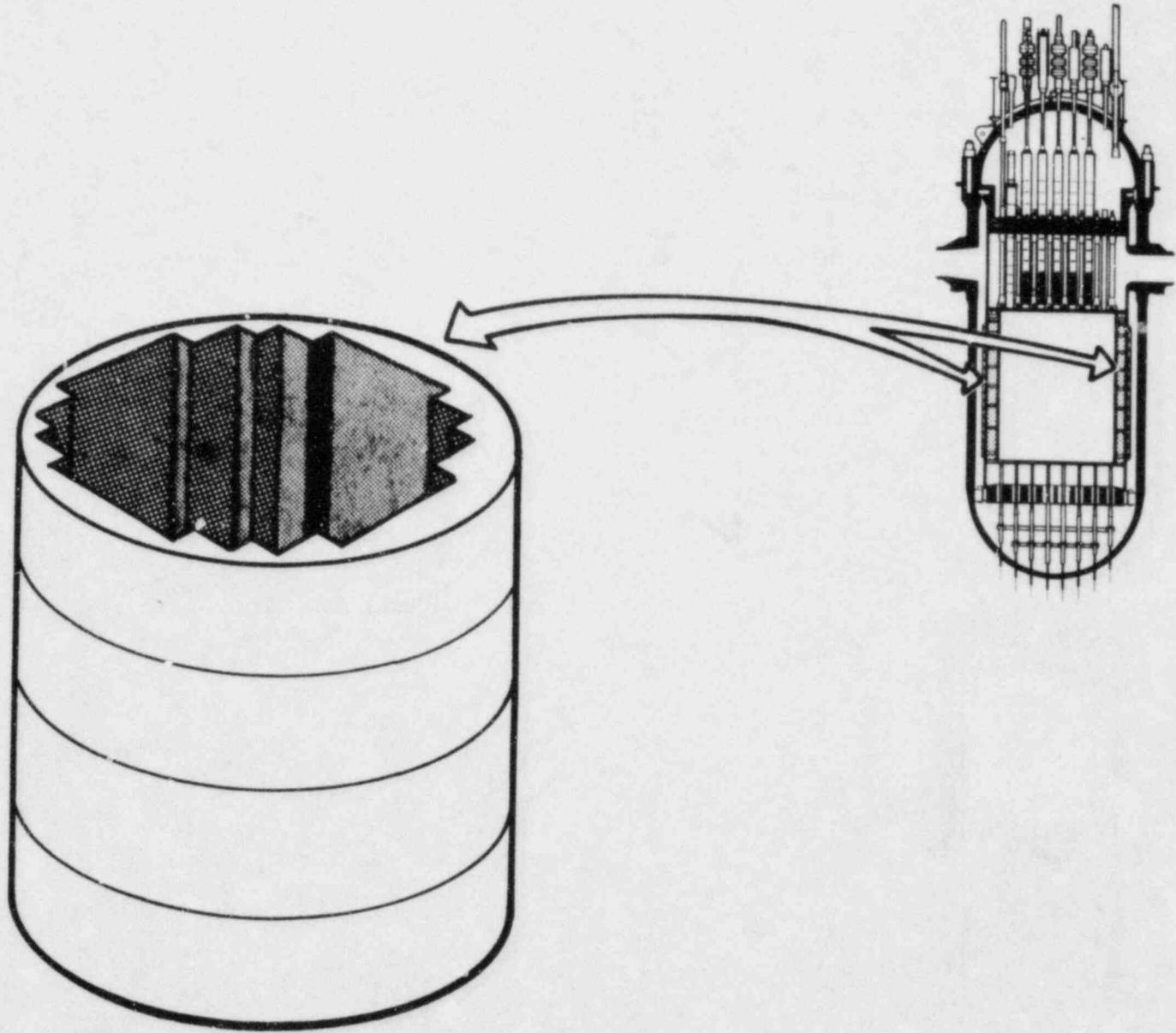


FIGURE 3.5(c). Core Bypass Region

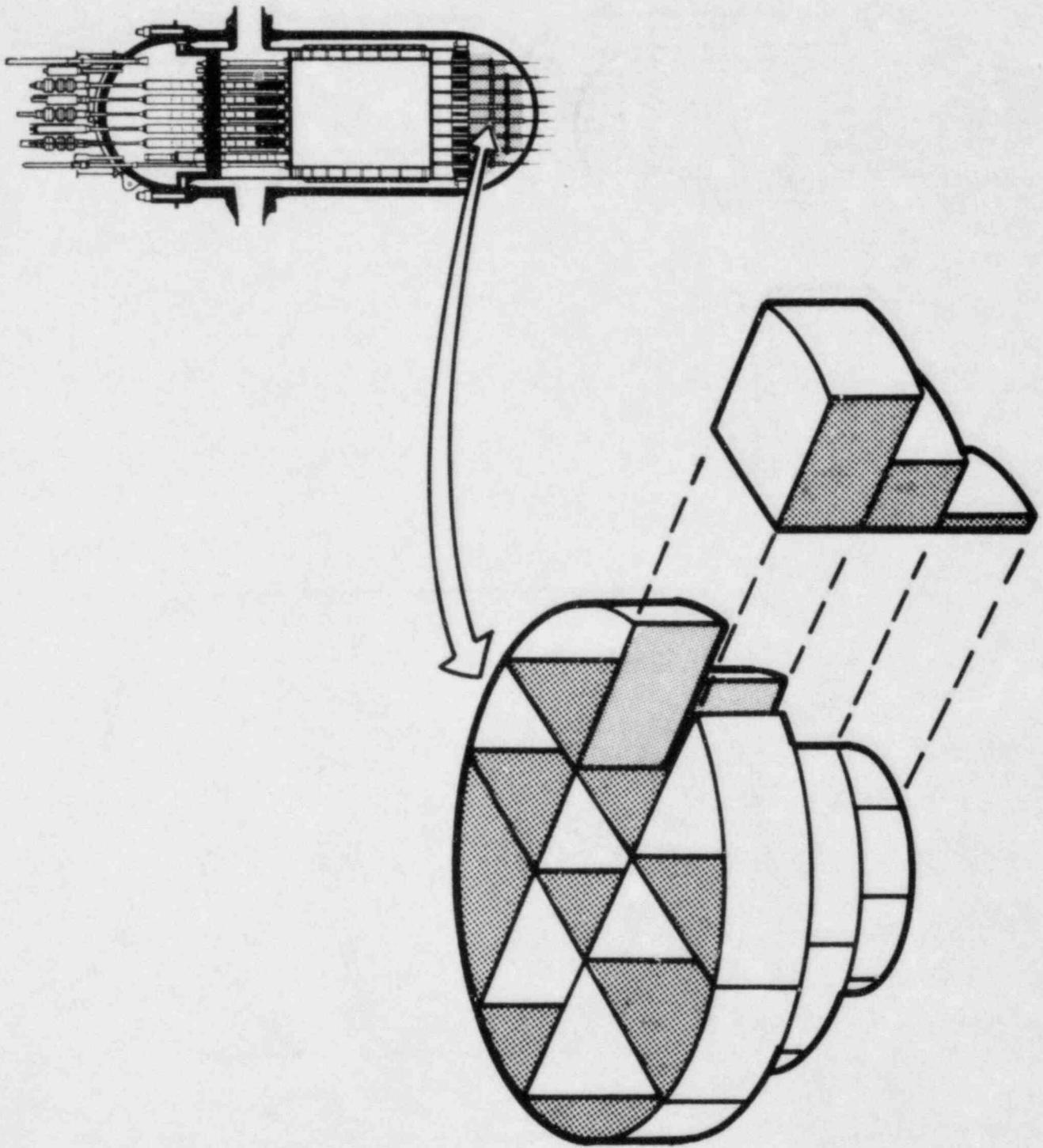


FIGURE 3.5(d). Lower Plenum Region

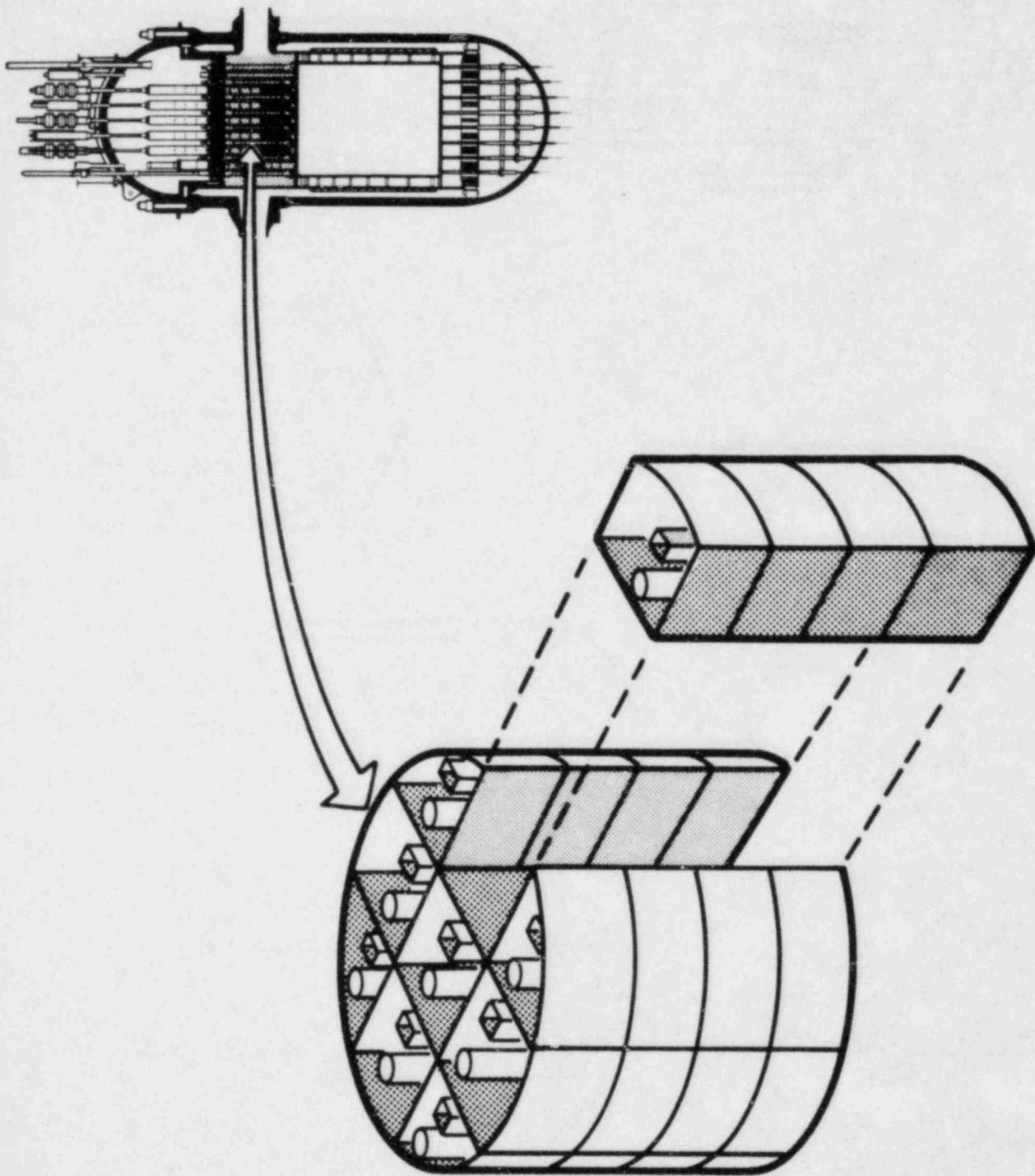


FIGURE 3.5(e). Upper Plenum Region

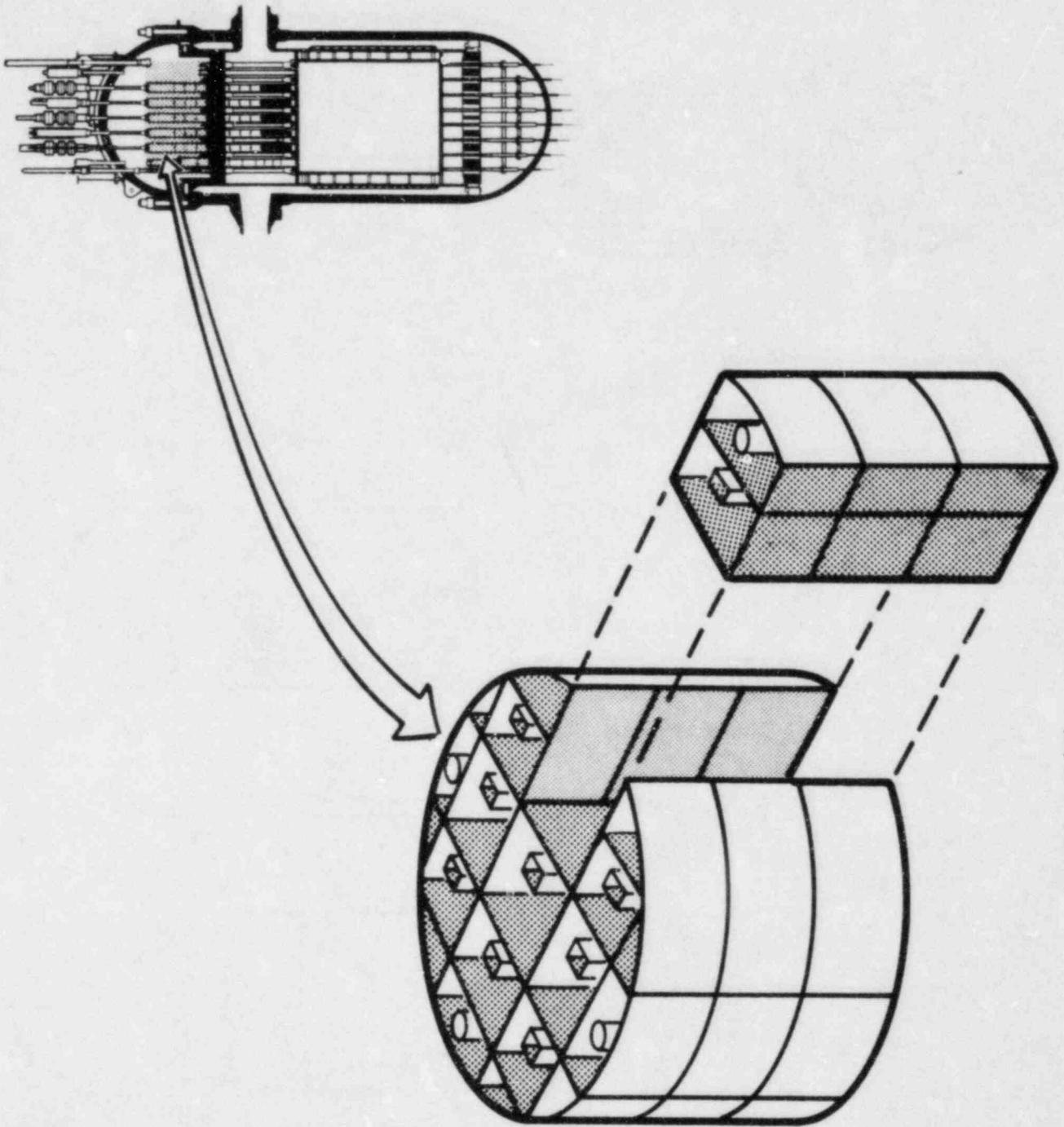


FIGURE 3.5(f). Upper Head Region

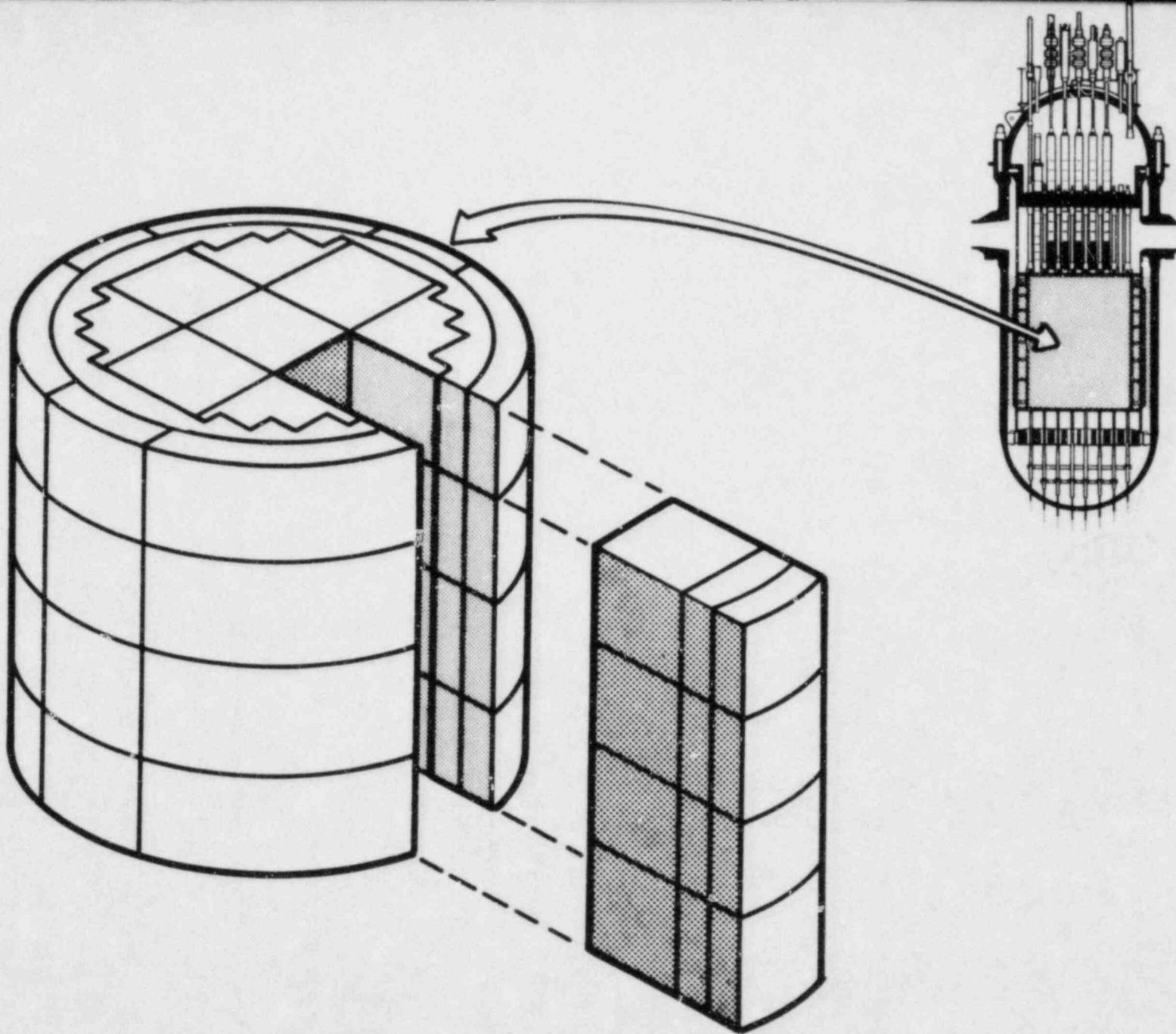


FIGURE 3.6. Core Section (Composed of Core Region, Core Bypass Region, and Downcomer Region)

the two regions. Other sections of the vessel are formed in the same way. The upper plenum and upper part of the downcomer form a section, as do the top of the downcomer and the lower portion of the upper head.

When all sections of the vessel have been defined, the sections are joined together to form the complete vessel mesh by specifying connections to the channels in adjacent sections at the top and bottom of each channel. This is illustrated in Figure 3.7. The ability to connect one or more channels to the top or bottom of a channel is referred to as channel-splitting. Each channel and gap in the problem is assigned a unique identification number by the input, and these channel numbers are used to identify the connections at the top and bottom of each channel. Connections are not specified for channels without a physical flow path at the inlet or outlet. The upper plenum and upper head, for example, are separated by a solid plate penetrated only by the guide tubes and support columns, so the channels in the open region of the plenum do not connect to channels in the open region of the upper head. Similarly, the tops of the channels in the upper head and the bottoms of those in the lower plenum model the solid structure of the vessel. Boundary conditions on the ends of channels that do not connect to channels in an adjacent section are specified by input.

Transverse connections between channels are not specified unless a physical flow path exists between the regions represented by the channels. This is very useful in modeling the complex vessel geometry but also makes it possible to model more than one vertical component with a single VESSEL. One group of channels can represent the reactor vessel, and another group, unconnected to the first, can model part or all of some other vertical component of the system such as a steam generator.

Fuel rod simulators, nuclear fuel rods and the solid structures within the vessel component can be modeled using the rod and unheated conductor model. Heat transfer through solid structures, thermal storage during a transient, and quenching of a hot dry surface can be calculated by the conduction model for the vessel component. The user has a great deal of flexibility in modeling the geometries of the solid structures in the system. They can be represented as cylindrical rods, hollow tubes or flat

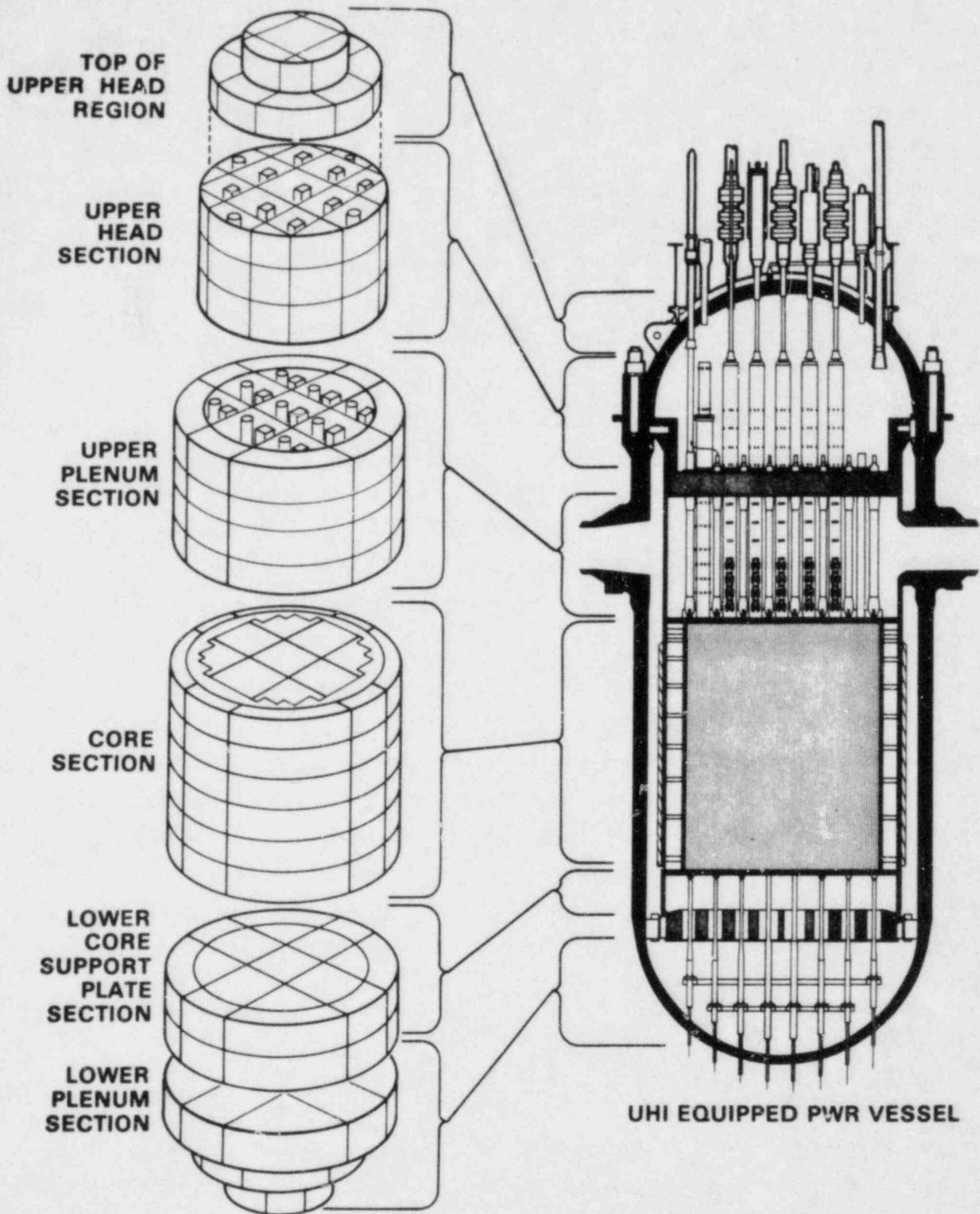


FIGURE 3.7. Complete Vessel Noding Diagram for PWR with UHI

walls composed of any number of different materials with specified thermal properties.

To summarize, the basic building block for the vessel mesh is the channel; that is, a vertical stack of single mesh cells. Several channels can be connected together by gaps to model a region of the reactor vessel. Regions that occupy the same level form a section of the vessel. Vessel sections are connected axially to complete the vessel mesh by specifying channel connections between sections. Heat transfer surfaces and solid structures that interact significantly with the fluid can be modeled with rods and unheated conductors.

The logic for setting up the vessel mesh was demonstrated with the example of a UHI-equipped PWR. Because of the generality of the mesh description, the capability of the code extends beyond that of modeling UHI plants. The geometry of other types of PWRs as well as BWRs can be modeled as readily. Several examples of systems that have been modeled using the vessel component are given in Volume 4 of this manual, COBRA/TRAC Development Assessment and Data Comparisons.

While this method for defining the vessel mesh is very versatile, it also places a burden on the user. The channel geometries and interchannel connections must be specified by input in some detail. The user must understand the modeling process and conventions in the VESSEL component well enough to define correctly the input for his problem. The vessel input will be easier to follow if the user first organizes the system he wishes to model with the VESSEL component into channels, gaps, sections and heat transfer surfaces, as illustrated for the UHI vessel. A detailed diagram of the system is an indispensable aid in this process.

The VESSEL component input is divided into groups, each of which fulfills some logical requirement of the vessel input stream. The geometry of the VESSEL mesh is specified in groups 2, 3, 4, 5, 6 and 7; group 2 describes the channels, group 3 the gaps and group 4 the sections. Groups 5, 6 and 7 are optional and can be used to specify axial variations in the geometry and local form drag loss coefficients.

3.1.1 Group 2: Channel Input Data

The input for group 2 tells the code how many channels the VESSEL component comprises and describes the nominal geometry of each channel. Each channel is uniquely identified by an index number I, flow area AN(I) and wetted perimeter PW(I), supplied on card VESSEL2.2. The channel identification numbers are completely arbitrary, and although it is usually most convenient to number them sequentially, the user is not required to do so. The only constraint on channel numbers is that they must be unique. The total number of channels input with card VESSEL2.2 must be NCHANL, the value of which is specified on card VESSEL2.1.

The variables ABOT and ATOP are provided as optional input to model the area at the top and bottom of the channel for the momentum equation solution. ABOT and ATOP default to the nominal flow area, AN(I), and in most cases this will be sufficient. But when section boundaries are located at orifice plates, flow distribution plates or other sudden changes in the geometry of the system, ABOT and ATOP can be used to supply the correct area at the bottom or top of the channel for the momentum solution. Nonzero values input for ABOT and ATOP replace the nominal channel area at those locations.

If the vertical velocities for a channel can convect transverse momentum between sections, the location of those velocities must be specified by the user. The user must realize that there can be no defaults for this input, since the channel numbering and interchannel connections at section boundaries are completely arbitrary. If this input is not provided, transverse momentum will accumulate in the affected cells of channel I at the section boundaries, causing errors in the pressure solution. The number of gaps for which channel I convects transverse momentum between sections is specified as the variable NAMGAP on card VESSEL2.2.

The connections for vertical convection of transverse momentum across the section boundary by a given channel I are specified on card VESSEL2.3. The connections are defined by the node number (INODE(I,N)) of the vertical velocity convecting momentum, and the index numbers of the gaps, above and below the section boundary, whose momentum is convected. (The node numbering convention used in the VESSEL component of COBRA/TRAC is illustrated in

Figure 3.8.) The centers of the scalar mesh cells are identified by $J = 1$, etc., and the centers of the momentum cells are identified by $j = 1$, etc). The node number $INODE(I,N)$ will be 1 for the channel velocity at the bottom boundary of the section, or $NONODE+1$ for the channel velocity at the top of the section. $KGAPB(I,N)$ is the gap number on the lower side of the section boundary, and $KGAPA(I,N)$ is the gap number on the upper side of the section boundary. If flow is positive, the vertical velocity of channel I convects transverse momentum from $KGAPB$ to $KGAPA$. If the flow reverses, momentum is convected from $KGAPA$ to $KGAPB$. This input is fairly obvious for cases where there is a gap both above and below the section boundary. But even if one or the other is nonexistent this input must be provided, to tell the code how to dissipate transverse momentum that is transported out of a gap across a section boundary. If there is a gap below, but no gap above the section boundary, $KGAPA$ is entered as zero, and momentum convected out of $KGAPB$ is considered dissipated. Similarly, if there is a gap above, but no gap below, $KGAPB$ is entered as zero. Reverse flow convects momentum out of $KGAPA$, which is considered dissipated. The area through which momentum is convected is calculated in the code as the minimum of $S_k L_k / 2$ for $KGAPA$ and $KGAPB$, and the flow area of channel I . A channel can convect transverse momentum to or from several gaps at the top and/or bottom of the section.

In defining the input for card VESSEL2.3, the user must understand how the momentum equation is solved at section boundaries, to determine if the vertical velocity will be solved in channel I at that section boundary. It does not do any good to specify the $NAMGAP$ connections for channels that are not solved at the section boundary, and conversely, they must be specified for those that are. Regardless of the geometry being modeled, there are three basic patterns for channel splitting connections. These are: 1) one channel below connected to one channel above, 2) one channel below connected to many channels above, and 3) multiple channels below connected to a single channel above. Figure 3.9 demonstrates these three patterns and shows the locations of the momentum cells solved at the boundary.

Transverse momentum convection input data is required only for channels where the momentum equation will be solved at the section boundary, and the

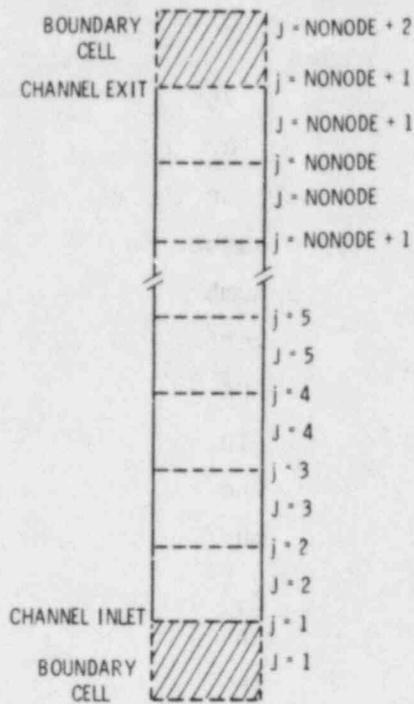


FIGURE 3.8. Channel Node Numbering Convention

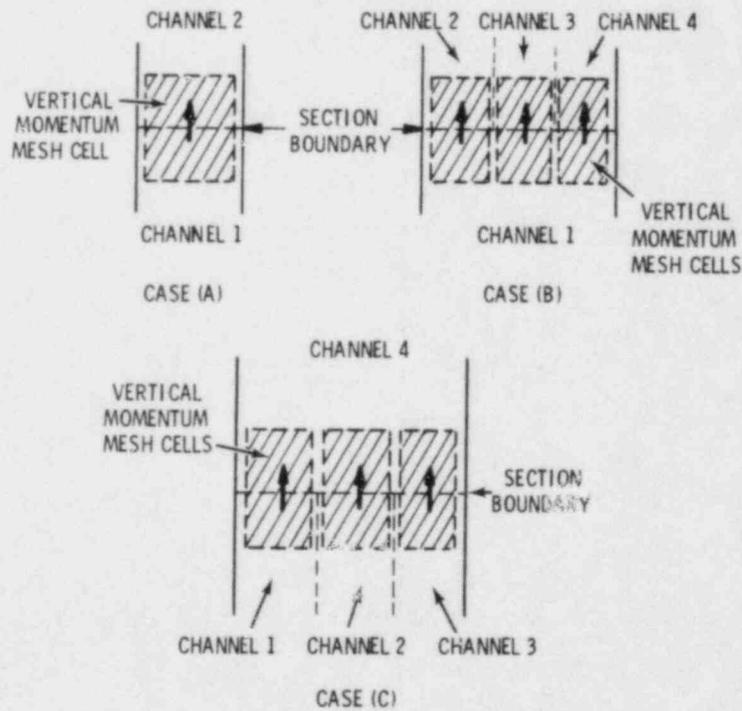


FIGURE 3.9. Channel Connections at Section Boundaries Allowed by the Channel Splitting Logic

velocities so obtained can convect transverse momentum from or to adjacent gaps. The momentum equation is solved at the top of channel 1 for case (a) in Figure 3.9 and is not solved at the bottom of channel 2. The momentum equation is solved at the bottom of channels 2, 3 and 4 in case (b), and at the top of channels 1, 2 and 3 in case (c).

The possible channel and gap configurations at section boundaries are outlined in Figure 3.10. The case where two channels below the section boundary connect to two channels above the section boundary with no change in the number of mesh cells between sections is illustrated in Figure 3.10a. The vertical momentum equation is solved in the top nodes of channels 1 and 2 to obtain vertical velocities at the section boundary. The velocity of channel 1 convects transverse momentum from the left side of the transverse momentum cell for gap 1 to the left side of the transverse momentum cell for gap 2. Similarly, the velocity of channel 2 convects momentum from the right side of the momentum cell for gap 1 to the right side of the momentum cell for gap 2. In this example, the input for card VESSEL2.3 would be supplied for both channel 1 and channel 2, with INODE(1,1) and INODE(2,1) equal to the index number of the top node in channels 1 and 2, KGAPB(1,1) and KGAPB(2,1) equal to 1 and KGAPA(1,1) and KGAPA(2,1) equal to 2. The user might be tempted to repeat this input for channels 3 and 4 at INODE(I,i) = 1, but since the vertical momentum equation is not solved at node 1 in channels 3 and 4, the redundant input is incorrect.

For the case shown in Figure 3.10b there is no gap on the top side of the section boundary, so the vertical velocities at the top of channels 1 and 2 convect momentum out of gap 1, but the momentum is assumed to be dissipated since there is no transverse momentum cell to receive it. In this case KGAPB(1,1) and KGAPB(2,1) are set to 1, and KGAPA(1,1) and KGAPA(2,1) are set to 0.

If a gap exists both above and below the section boundary as in Figure 3.10a, but there is a flow straightener (such as a core support plate) at the section boundary, then the transverse momentum can be dissipated by specifying two connections for each channel, one for positive flow and the other for negative flow. Positive flow convects momentum from gap 1 and

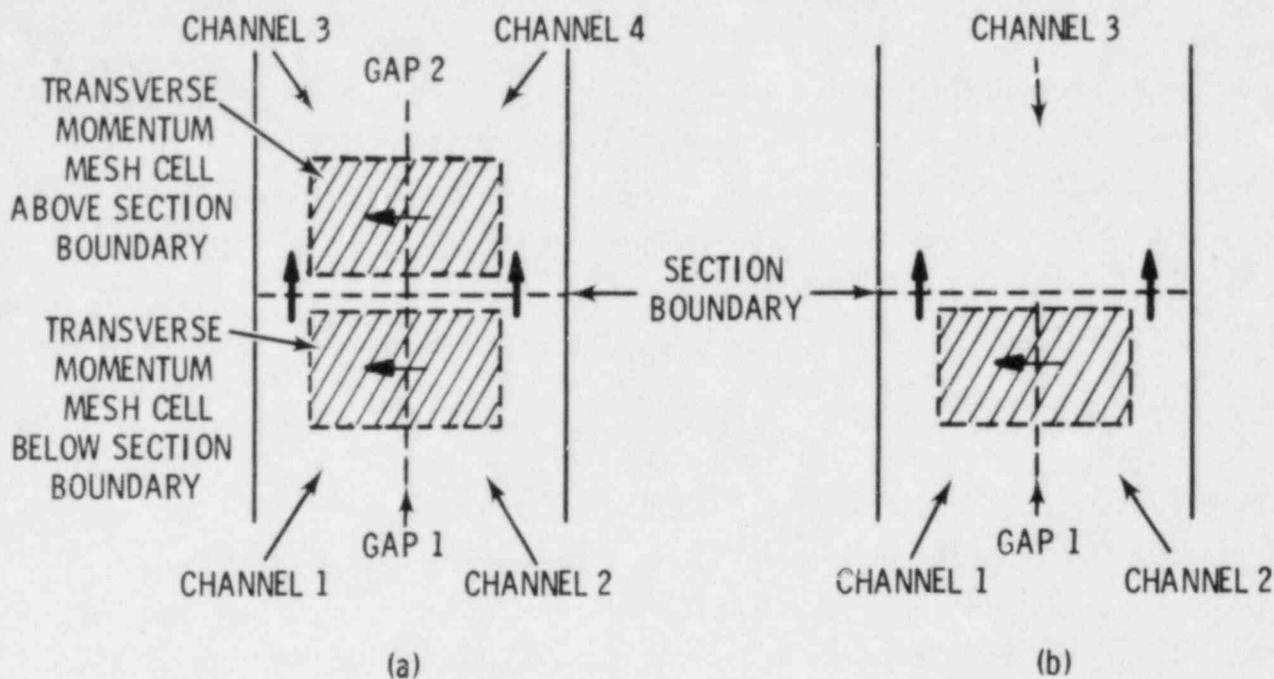


FIGURE 3.10. Typical Configurations for Convection of Transverse Momentum Between Sections

dissipates it, so $KGAPB(1,1)$ and $KGAPB(2,1)$ are specified as 1, $KGAPA(1,1)$ and $KGAPA(2,1)$ as 0. Negative flow convects momentum from gap 2 and dissipates it, so $KGAPB(1,2)$ and $KGAPB(2,2)$ are specified as 0, and $KGAPA(1,2)$ and $KGAPA(2,2)$ as 2.

If momentum is convected by a vertical velocity from one gap on one side of the section boundary to several gaps on the other side (variable mesh), then transverse momentum convection connections must be specified for each gap. The momentum taken from the single gap is apportioned among the connecting gaps by the relative size of the areas through which the momentum is convected.

As part of the description of the physical geometry of the VESSEL, the group 2 input data describes the channel connections to rods and unheated conductors for heat transfer between the fluid and solid surfaces. The input supplied on card VESSEL2.4 must be consistent with the data for the rods and unheated conductors supplied in group 8. (See Section 3.2.) All rods and unheated conductors must be connected to at least one channel, but not all channels need to be connected to a rod or unheated conductor. $NCHANR$ is the

number of thermal connections between channels and rods or unheated conductors ($NCHANR < NCHANL$). For a given channel I, the $LR(I,L)$ array contains the index numbers of the rods that connect to channel I. The $LS(I,L)$ array contains the index numbers of the unheated conductors connected to channel I. If channel I connects to the inside surface of a rod (or unheated conductor), the index number of the rod (or unheated conductor) is specified as its negative. Only one channel can be connected to the inside or outside surface of a given rod or unheated conductor within a given section.

3.1.2 Group 3: Transverse Channel Connection Data

The input for group 3 describes the transverse connections between adjacent flow channels. These connections are referred to as gaps and reflect the original conception of a gap in subchannel analysis as the lateral flow path between two fuel rods. In the COBRA/TRAC vessel model, the gap input data define the control volume for the transverse momentum equation, on either a subchannel analysis basis or for a three-dimensional analysis of the transverse flow. The input for the gaps falls into five main categories--the geometry input describing the gap and identifying its connecting channels; the friction loss and form drag coefficients for flow through the gap; description of the vertical connections between gaps across section boundaries; data describing the optional three-dimensional layout of the gap; and special input to describe gaps between channels modeling large regions of the vessel.

The geometry data for each gap is read on card VESSEL3.2. Each gap is identified by a unique number, K. Gap K connects channel $IK(K)$ to channel $JK(K)$. By convention, $IK(K)$ is the lower-numbered channel of the pair, $JK(K)$ the higher numbered, and crossflow from gap $IK(K)$ to $JK(K)$ is defined to be positive. Flow the other way is negative. The nominal gap width is specified as $GAPN(K)$, and the distance between the centers of the connecting channels is $LENGTH(K)$. These variables define the width and length of the transverse momentum mesh cell. The flow area between the connecting channels is given by the product of the gap width and the vertical length increment for the mesh, ΔX . (This flow area is computed in the code from the input data.) The gap width, $GAPN(K)$, is equal to the total width for transverse flow between the two adjacent regions modeled by channels $IK(K)$ and $JK(K)$.

Form and wall drag in gap K is specified for the transverse momentum equations using the parameters WKR(K) and FWALL(K). WKR(K) is the form drag loss coefficient (velocity head) and is specified by the user. The value of WKR(K) for a gap depends on the geometry of the flow path modeled by the gap. A value of 0.5 is typically used for flow across one row of rods or tubes. For lumped channels, this value is multiplied by the number of rod rows the gap momentum cell contains.

Wall friction in the gap is ordinarily included in the WKR parameter, but for gaps that are formed by vessel walls rather than rod arrays (as in the downcomer annulus, for example) the pressure loss in the gap is primarily a friction loss rather than a form loss and should be modeled accordingly. Wall friction factors in a gap are computed internally in the code according to the user-specified value of the FWALL(K) parameter. If FWALL(K) is zero, no wall friction is calculated. If FWALL(K) is set equal to 0.5, a wall with a surface area of $\Delta X * LENGTH(K)$ is assumed (i.e., one side of the control volume is a solid wall). A wall surface area of $2 * (\Delta X * LENGTH(K))$, (i.e., a wall on each side of the control volume), is assumed if FWALL(K) is set equal to 1.0. Both a form loss and wall friction factor can be specified for a gap, or one or the other can be set to zero, depending on the geometry of the problem. Both can be set to zero if the gap is in an open fluid region where the fluid-fluid shear within the continuous phase adequately models the transverse drag forces.

The input required to describe transverse convection of vertical momentum between gaps across section boundaries employs the same sort of reasoning as does the input in group 2 to describe the vertical convection of transverse momentum across section boundaries. The user must specify which transverse gap velocities can transport vertical momentum into or out of the channels associated with gap K at the top and bottom of the section. The transverse velocity of gap IGAPB(K) (at node NONODE+1 in the section below) can convect vertical momentum into or out of the bottom cell of the channels associated with gap K. The transverse velocity of IGAPA(K) (at node 2 in the section above) can convect vertical momentum into or out of the top cell of the channels associated with gap K.

Transverse convection of vertical momentum at a section boundary is illustrated for a specific case in Figure 3.11. In this example, vertical momentum is convected between the two vertical mesh cells by the transverse velocity of gap 1 below the section boundary, and by the transverse velocity of gap 2 above the section boundary. The input for IGAPA(K) identifies the gap in the section above gap K that convects vertical momentum between the vertical momentum mesh cells associated with the top of gap K; in the example shown in Figure 3.11, IGAPA(1) = 2. IGAPB(K) identifies the gap in the section below gap K that convects vertical momentum between the vertical momentum mesh cells associated with the bottom of gap K. IGAPB(2) = 1 in our example.

The remainder of the input on card VESSEL3.2 is required only for the three-dimensional form of the transverse momentum equation. If the subchannel formulation is desired for a particular problem, this input is omitted. It is used to define consistent transverse flow directions for the global coordinate system, and to set up connections for orthogonal transport of momentum.

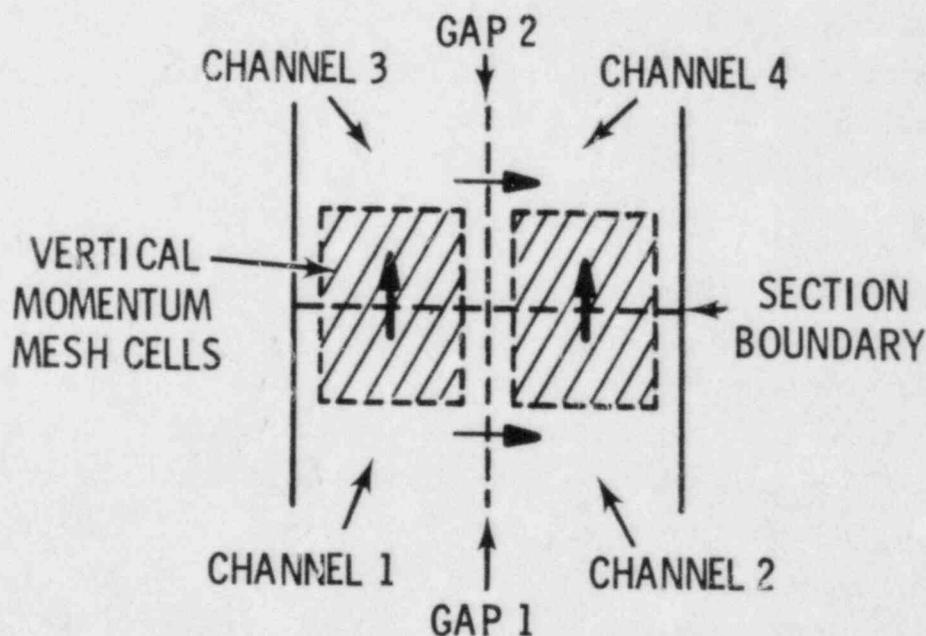
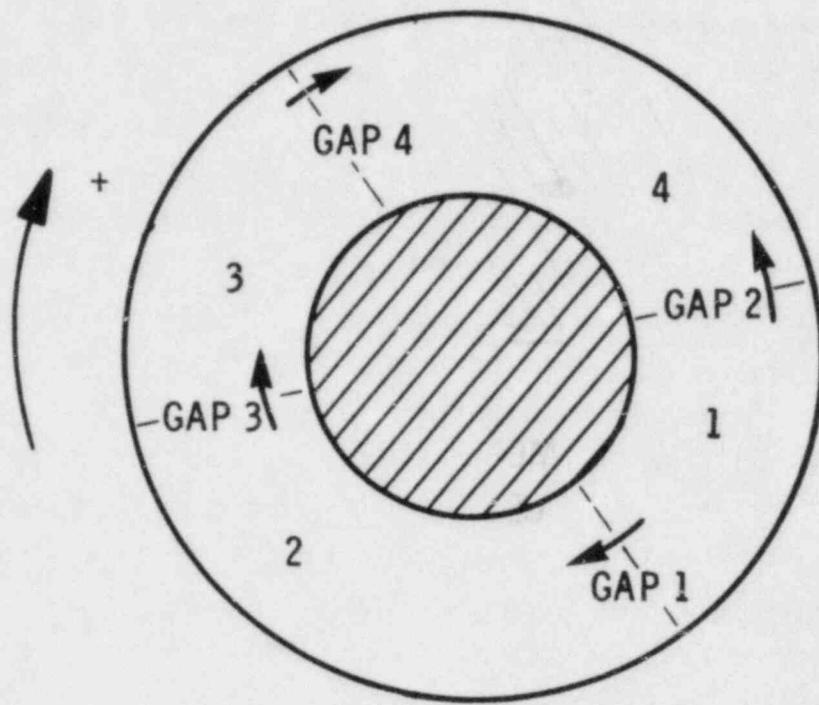


FIGURE 3.11. Vertical Momentum Mesh Cell at Section Boundary

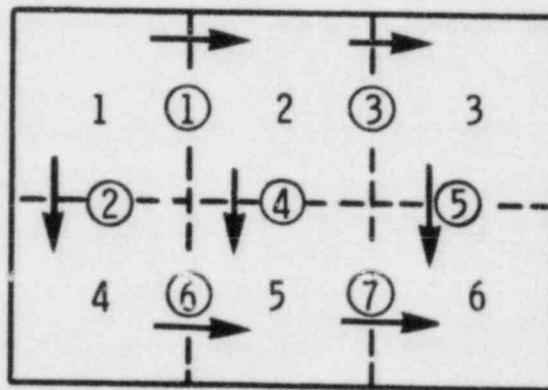
The normal crossflow sign convention is not always congruent with a global coordinate system. The user can convert the local transverse flow sign convention to an appropriate global system by specifying the variable FACTOR(K) (the tenth variable on card VESSEL3.2) to indicate the orientation of gap K in the global coordinate system. Figure 3.12 gives two examples of how FACTOR can be used. Example (a) shows an annular channel arrangement typically used in the downcomer. The annular flow channels are in effect two dimensional, so it is only necessary to define a transverse direction that is consistently positive. The normal convention for defining positive crossflow results in crossflow from channel 1 to 2 and from channel 1 to 4 both being positive. Assuming that the clockwise direction is chosen as positive for the global coordinate system, if flow from IK(K) to JK(K) is clockwise, then FACTOR(K) is set to 1.0. If flow from IK(K) to JK(K) is counterclockwise, the sign convention must be reversed by setting FACTOR(K) to -1.0. (Of course, the user may choose counterclockwise as the positive direction, if he wishes. The important thing is to be consistent.) In this example, FACTOR(2) is set to -1.0.

A three-dimensional channel array is shown in example (b) in Figure 3.12. For this case, the values for FACTOR(K) must correspond to the actual orientation of the velocity vector in the gap on the global coordinate system. The arrows in the gaps indicate the positive crossflow direction defined by the normal convention. For gaps 1, 3, 6 and 7, this is the same as the global coordinates, so FACTOR(1), FACTOR(3), FACTOR(6) and FACTOR(7) are 1.0 for this case. But the positive flow direction in gaps 2, 4 and 5 is exactly opposite of the global convention, so FACTOR(2), FACTOR(4), and FACTOR(5) must be specified as -1.0 to reverse the sign on the crossflow velocity.

For the three-dimensional form of the transverse momentum equation, the user must identify the gaps facing a given gap K. This information is required in the code to calculate the normal components ($\rho V^2 A$) of transverse momentum. The arrays IGAP(K,N) and JGAP(K,N) on card VESSEL3.2 are used to supply this data. The IGAP array holds the identification numbers of up to three gaps on the IK (or lower-numbered channel) side of gap K, and the array



(A)



(B)

FIGURE 3.12. Global Coordinate Systems

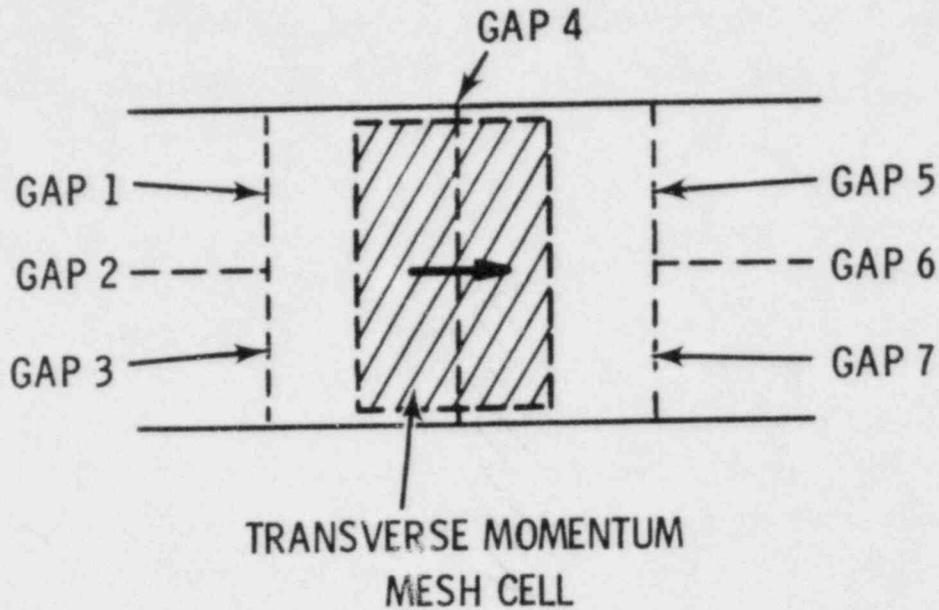


FIGURE 3.13. Normal Component of Transverse Momentum

JGAP holds the identification numbers of up to three gaps on the JK (or higher-numbered channel) side of gap K. This is illustrated in Figure 3.13. In this example, gap 4 connects channel 1 to channel 2, with the conventional positive flow direction shown by the arrow. Gaps 1 and 3 face the IK side of gap 4, and gaps 5 and 7 face the JK side. In the example, $IGAP(4,1) = 1$, $IGAP(4,2) = 3$, $JGAP(4,1) = 5$, and $JGAP(4,2) = 7$.

Card VESSEL3.3 must be read for each gap, following card VESSEL3.2. This input is designed to model gaps between channels that model large regions in the vessel. The nominal gap width, $GAPN(K)$ (supplied on card VESSEL3.2), is the total width for crossflow between the two channels. The characteristic width for two-phase flow through the gap, however, is the distance between the physical structures and not necessarily the total width of the connection between mesh cells. The parameter $GMULT(K)$, input on card VESSEL3.3, is the number of uniform spaces between structures in the region modeled by the gap. The characteristic dimension for two-phase flow is given by

$$S' = \frac{GAPN(K)}{GMULT(K)} \quad (3.1)$$

The dimension S' determines the maximum bubble size that can pass between structures. If $GAPN(K)$ is the actual width available for flow, $GMULT(K)$ is one.

$ETANR(K)$, which is also read on card VESSEL3.3, is the fraction of transverse droplet flow de-entrained on structures within the mesh cell. In a reactor vessel or test section the array of rods in the core and support columns and guide tubes in the upper plenum can contribute significantly to de-entrainment of droplets from the transverse component of the flow. The correlation suggested by Dallman and Kirchner for the de-entrainment fraction for flow across tube banks is recommended unless the user has an experimentally determined value for his particular geometry. The de-entrainment fraction, $ETANR(K)$, for a gap modeling an array of tubes is

$$ETANR(K) = 1 - (1 - \eta_R)^N \quad (3.2)$$

where η_R = de-entrainment fraction of a single row of tubes
 N = number of rows of tubes.

The de-entrainment fraction of a single row of tubes is

$$\eta_R = \eta_I (1 + 4.5 \beta^2) \quad (3.3)$$

where $\eta_I = 0.19$ for cylindrical tubes
 $= 0.27$ for square tubes
 β = pitch-to-diameter ratio of the array.

(Volume 1 of this manual contains a complete discussion of the model.)
 $ETANR(K)$ can be used to model the crossflow de-entrainment rate for geometries other than tube banks and rod bundles, but the user must determine the appropriate value for each particular application.

The final input lines for group 3 (cards VESSEL3.4, and VESSEL3.5) are required only for the three-dimensional form of the transverse momentum equation. The total number of gaps that convect orthogonal transverse

momentum is specified as the NLMGAP parameter on card VESSEL3.4. The velocity of the gap identified in KGAP1(N) convects transverse momentum from KGAP3(N) to KGAP2(N) if the velocity is positive and from KGAP2(N) to KGAP3(N) if negative. This is illustrated in Figure 3.14. In this example the velocity of gap 1 convects momentum from the left half of the momentum mesh cell for gap 4 to the left half for gap 2. The velocity of gap 3 convects momentum from the right half of the cell for gap 4 to the right half of gap 2. (Gaps 2 and 4 also convect momentum from gap 1 to gap 2.) In this example, KGAP1(1) = 1, KGAP2(1) = 2, KGAP3(1) = 4, KGAP1(2) = 3, KGAP2(2) = 2, and KGAP3(2) = 4. (Also KGAP1(3) = 2, KGAP2(3) = 3, KGAP3(3) = 1, KGAP1(4) = 4, KGAP2(4) = 3, and KGAP3(4) = 1).

Examples of transverse flow connections typically encountered in the mesh for a reactor vessel are shown in Figure 3.5. The gap between the local mesh for a guide tube and the surrounding global mesh cell, as shown in Figure 3.5e is a special case and merits some additional discussion. Such relatively small meshes are used to obtain local detail of important flow paths while still maintaining the larger, more economical global mesh size. Local meshes within a relatively larger global mesh represent a compromise in that some of the local details of the flow and some of the accuracy (relative to that obtainable with a global mesh size on the order of the local mesh) is lost. But in most cases a global mesh small enough to resolve flow details is prohibitive in both computation time and storage requirements. A reasonable prediction of local flow phenomena can be achieved using the large global mesh with a local mesh specified within it.

In general, the convection of transverse momentum by transverse velocities in the gap between the local and global mesh cells is neglected. The transverse flow between the local mesh and global mesh is assumed to be axisymmetric. The shape of the transverse momentum mesh cell is idealized and requires the user's judgement in selecting values for the gap width and length. The gap width should be chosen such that the product of the gap width and the vertical mesh length increment is equal to the physical area for the flow path between the local and global mesh cells. The shape of the momentum mesh cell is assumed rectangular and its length should be chosen to give a

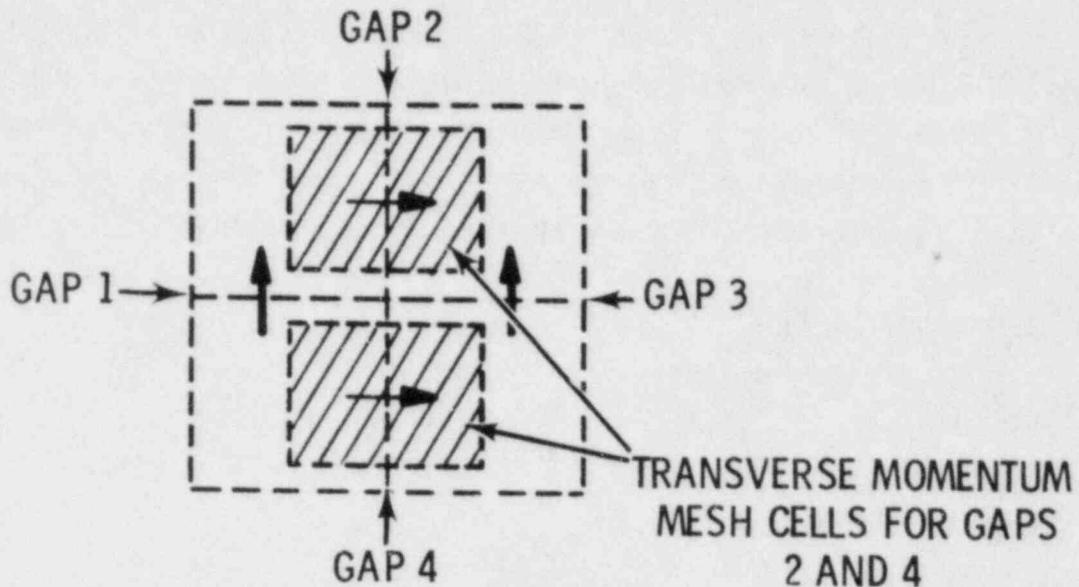


FIGURE 3.14. Convection of Transverse Momentum by an Orthogonal Transverse Velocity

physically meaningful time constant for accelerating flow through the gap. A distance equal to the diameter of the local mesh is recommended for the centroid distance (LENGTH(K)).

The local mesh may represent several flow paths that have been lumped together. The local mesh for a guide tube, for example, represents the several guide tubes occupying the region of the upper plenum modeled by the global mesh cell. In such cases the gap width should be the sum for all the guide tubes, and the centroid length should be the diameter of a single guide tube, with the flow area and wetted perimeter of the local mesh channel equal to the sum of all the guide tubes.

Local meshes that represent the paths of fluid jets should have geometry values that represent the length scales of the jet.

Setting up the gap geometry data is relatively simple even for peculiar geometries if the user bears in mind that GAPN(K) and LENGTH(K) define the transverse control volume for the gap. The size and shape of this control volume should bear some logical resemblance to the physical structure being

modeled, within the constraints of the noding philosophy used in the code. The control volume is important not only for defining the location and magnitude of the transverse flow, but also for determining vertical transport of transverse momentum through the top and bottom surfaces of the control volume. Both aspects of the momentum solution must be considered in defining the gap width and centroid length.

3.1.3 Group 4: Section Data

The input data for group 4 describes the vertical channel connections between sections and defines the rebalancing and simultaneous solution group options. The input data describing each section is read on card VESSEL4.2. It consists of the section number (ISEC), the number of channels (ICHN) in the section, the number of vertical nodes (NONODE) in the section and the vertical node length (DXS) in the section. The section boundaries are uniform at a given vertical level, so all channels within a section have the same total length. The vertical node length DXS in a section defines the length of both the continuity and momentum control volumes. DXS is constant within a section, so the total vertical height of a section is $DXS * NONODE$. DXS can vary between sections, and each section may have one or many vertical nodes. The only constraint on the DXS values chosen for different sections is that the change in DXS between adjacent sections should not be greater than a factor of two.

The vertical connections for each channel in a section are specified by the input for card VESSEL4.3 with the identification numbers of the channels it connects to in the section above and the section below. A channel may connect to up to six channels in the section above and six channels in the section below. The array KCHANA contains the index numbers of channels connected to the top of the channel, and KCHANB contains the index numbers of channels connected to the bottom. If a channel does not have a connection above or below, KCHANA or KCHANB is specified with the channel's own identification number. The code will not accept a channel with only zeros in the KCHANA or KCHANB arrays.

There are three basic patterns possible for vertical connections between channels: a) one channel below connected to one channel above, b) one channel

below connected to many channels above, and c) multiple channels below connecting to a single channel above. The momentum equation for the vertical velocity at the section boundary is solved at the top node of the bottom channel for case (a), in the bottom node of each of the upper channels in case (b), and in the top node of each of the lower channels in case (c). The momentum mesh cells at the section boundary for these three cases are shown in Figure 3.15. Velocities are solved for in the momentum cells at the top of channel 1 for case (a), at the bottom of channels 2, 3 and 4 for case (b) and at the top of channels 1, 2 and 3 for case (c).

Velocities at the momentum cell faces, which are used to calculate the momentum flux terms, are obtained by averaging the velocities at the cell centers of adjacent momentum cells. At section boundaries where the connections are like those shown in cases (a) and (c) in Figure 3.15, the velocity at the top face of the momentum cell is obtained by averaging the velocities at the centers of the momentum mesh cells on the boundary with the velocity at the center of the first momentum mesh cell in the channel above. For case (a), the velocity of channel 2, node 2, is averaged with the velocity calculated in channel 1 at the boundary. For case (c) the velocity of channel 4, node 2, is averaged with the velocities calculated in channels 1, 2 and 3 at the boundary. For case (b) the velocity calculated in the last cell of channel 1 is averaged with velocities calculated in channels 2, 3 and 4 at the boundary. A channel with multiple connections to both the bottom and top must contain at least two mesh cells. If the channel had only one cell, the code would not be able to determine which velocities to use when obtaining average velocities at the cell face for momentum cells at the bottom of the channel or at the top of the channel.

Figure 3.16 shows two samples of channel connections that are not permitted by the channel splitting logic in the code. These examples are given because they represent two of the easiest mistakes to make in setting up the channel-splitting input. The user should bear in mind that any other connections he might think of that are not in one of the patterns shown in Figure 3.15 will not work either. In (a) in Figure 3.16, the channels below overlap in their connections to the channels above. Trying to draw the

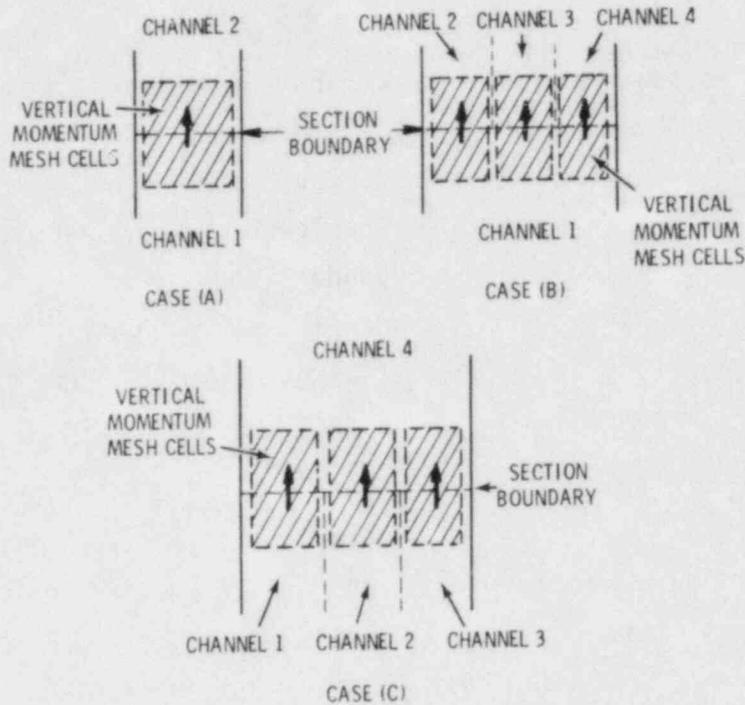
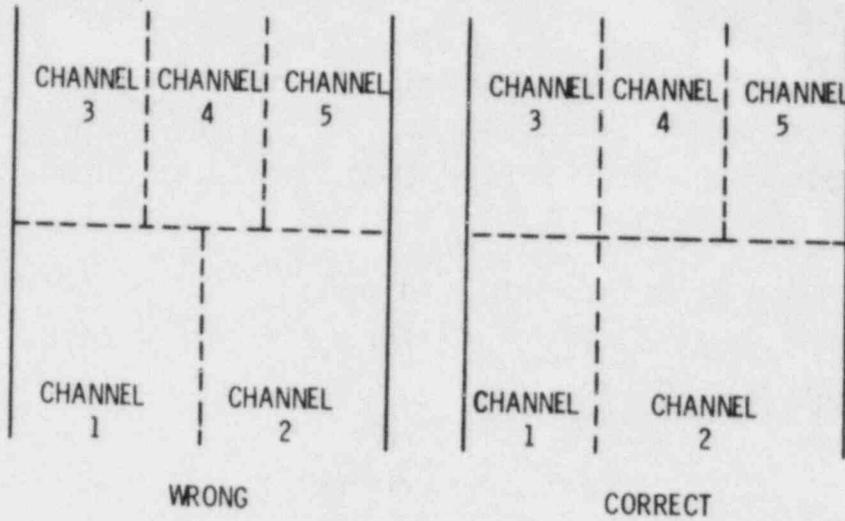


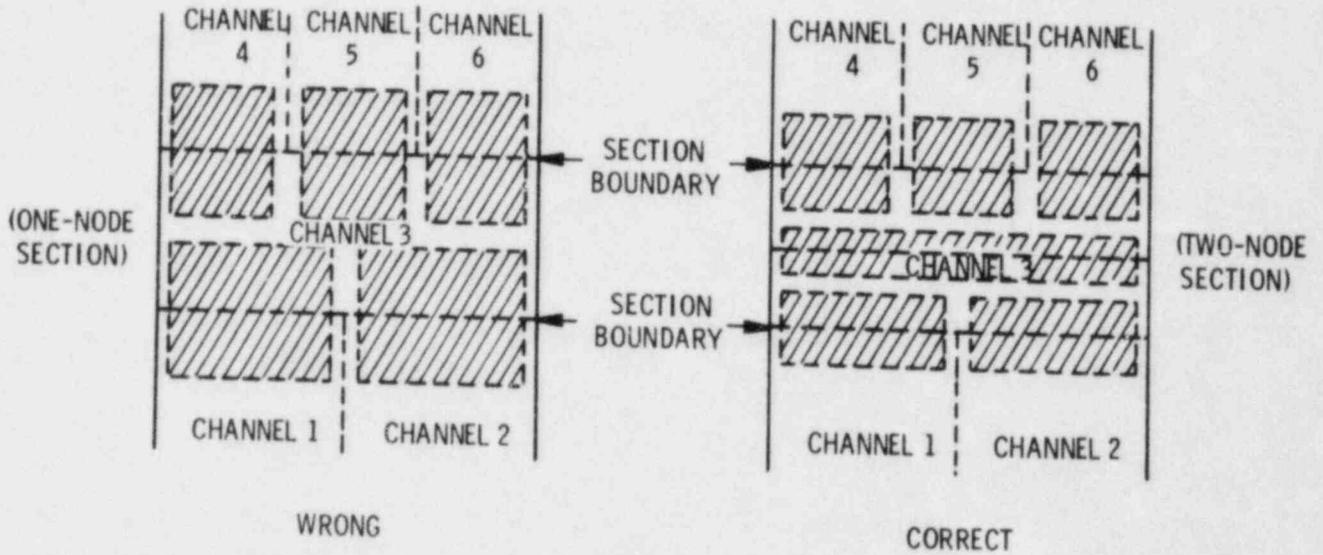
FIGURE 3.15. Allowable Vertical Connections Between Channels at Section Boundaries

momentum control volume at the inlet boundary to channel 4 is the simplest way to see why that configuration is not allowed. (The mirror image of this configuration, i.e., with channels above overlapping channels below, is not permitted either, for the same reasons.) A correct way to connect the channels in this example is also shown in Figure 3.16. Example (b) in Figure 3.16 shows a channel with multiple connections above and below. The only thing wrong with this configuration is that channel 3 is only one cell long. It can be corrected by giving the section containing channel 3 one more node.

The logic in the code's input subroutine contains some error checks on the channel vertical connections, primarily to determine consistency but also to check for the erroneous one-node section pattern illustrated in part (b) of Figure 3.16. Appropriate error messages are printed if inconsistent connections above and below are detected. There is no possible generalized check, however, for the most common error in channel splitting input, i.e.,



(a)



(b)

FIGURE 3.16. Common Channel Splitting Errors

connecting the wrong channels in a logically correct pattern. The only defense against this is for the user to have a clear picture of the channel connection pattern for his problem to check against the output data COBRA/TRAC provides.

The remainder of the input in card group 4 specifies the numerical solution procedure to be used for solving the linearized continuity and energy equations. The equations may be solved by a direct inversion of the system pressure matrix or a combination of direct inversion and iteration. In the combination solution, the pressure matrix for groups of cells is solved by direct inversion while the pressure in cells surrounding each group is held constant. A Gauss-Siedel iteration is then performed over the groups of cells to obtain a converged solution for the entire system pressure matrix.

The solution procedure used is determined by the parameter NSIM input on card VESSEL4.1. NSIM is the number of simultaneous solution groups. If NSIM is set to one (1) the solution is obtained by direct inversion. If NSIM is set to N (where N is an integer greater than one) the solution is obtained with a combination of direct inversion and iteration with N simultaneous solution groups. A direct inversion is recommended for relatively small problems with fewer than forty or fifty mesh cells. The group-by-group iteration is recommended for larger problems since the computer time required to invert the large system matrix usually exceeds that required to reach a suitably well-converged solution using the iterative procedure.

The convergence rate for the iterative solution can be greatly enhanced by specifying the rebalancing option, IREBAL = 1. When rebalancing is done, a one-dimensional solution for the linear pressure variation at each level in the vessel is obtained by direct inversion. This value is used as an initial estimate for the linear pressure variation in each cell in the group-by-group iterations.

The bandwidth of the pressure matrix is defined by the maximum difference between the index numbers of adjacent cells in a group that is being solved simultaneously. This maximum difference is specified by input as parameter IWIDE on card VESSEL4.4, and the maximum bandwidth of the pressure matrix is calculated as $(2*IWIDE+1)$. (The code must be dimensioned such that dimension

parameter NE is greater than or equal to $(2 \cdot IWIDE + 1)$. See Section 2.5.) The cell numbers used to define IWIDE are not channel node numbers. (Channel node numbers are unique only within a channel, while cell numbers are unique throughout the vessel mesh.) Cell numbers are assigned in the following manner: beginning at the first channel in the first section, the first node in the channel (node $J = 2$) is assigned cell number one (1). The cells of all the channels in that section are numbered sequentially at the same level ($J = 2$). The cell numbering is continued on the next level, starting in the same first channel. This process continues until all cells for all sections have been assigned unique cell numbers. An example illustrating this numbering convention is shown in Figure 3.17.

In the example in Figure 3.17, IWIDE is equal to 2 in simultaneous solutions groups 1 and 5. In groups 3 and 4, IWIDE is only 1. In group 2 IWIDE is 3. The value for IWIDE entered on card VESSEL4.4 for this example is 3. (The matrix must accommodate the largest bandwidth in all groups.) The user should construct a similar diagram for his problem, to determine the appropriate value for IWIDE, and also to provide the input for the MSIM array (the number of the last cell in each simultaneous solution group--card VESSEL4.5).

Simultaneous solution groups are based on the geometry of the given problem and should consist of cells that strongly influence each other's behavior, or are influenced by nearly the same external conditions. Volume 4, COBRA/TRAC Developmental Assessment and Data Comparisons contains several examples of geometries modeled with simultaneous solution groups.

3.1.4 Group 5 and Group 6: Variations in Channels and Gaps

The geometry of a channel or gap is assumed to remain constant along the entire vertical length at the nominal values specified by input. But the input for groups 5 and 6 permits the user to specify vertical variations in the geometry of channels or gaps. The input for these two groups is optional. If vertical variation data is not needed, groups 5 and 6 are omitted.

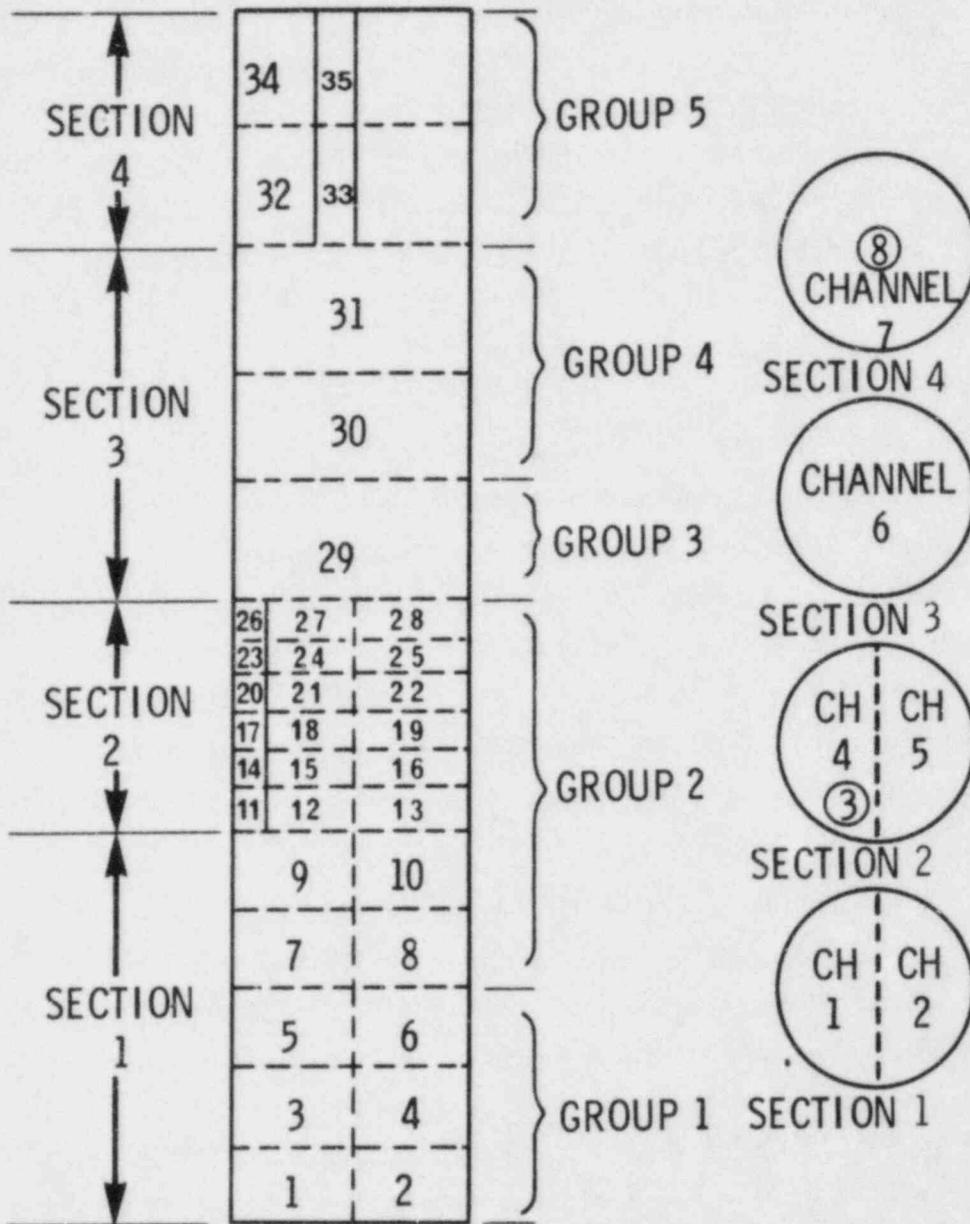


FIGURE 3.17. Example of Cell Numbering Scheme for Simultaneous Solution Groups

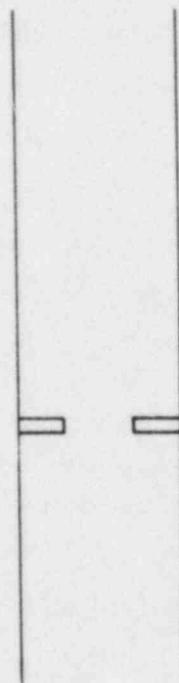
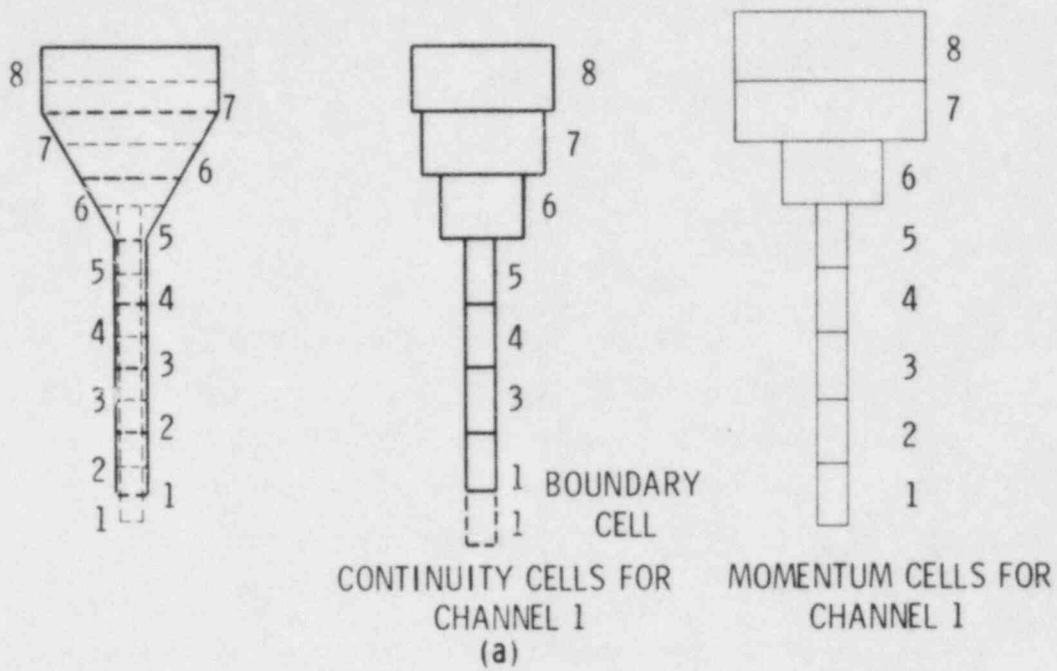
The input data for groups 5 and 6 is extremely general. The user provides tables of variation factors in group 5 and identifies in group 6 the channels or gaps that use the tables. Variation tables can be supplied to model variations in the continuity flow area, momentum flow area, wetted perimeter of channels, and the width of gaps. Variation tables are read on card VESSEL5.3 as tables of node number, JAXL(I,N), versus variation factor, AFACT(I,N). The node number JAXL(I,N) refers to the continuity cell for continuity area and gap width variations, and to the momentum cell for momentum area and wetted perimeter variations. The variation factor is defined as

$$AFACT(I,N) = \frac{\text{local value}}{\text{nominal value}}$$

The value of AFACT(I,N) is applied in the code as a multiplier on the nominal value of the quantity being varied at the corresponding JAXL(I,N) node-- continuity area, momentum area, wetted perimeter or gap width. Figure 3.18 shows two examples of channels with area variations that can be modeled with variation tables. For example (a), variations of the continuity area occur at nodes 6, 7 and 8. Variations in the momentum area occur at nodes 6, 7 and 8 also, but because the geometry of the channel is tapered and the continuity and momentum nodes are staggered, the areas vary by different amounts.

The tapered region in the channel is approximated by a stack of cells that have a uniform cross section along their individual lengths. The continuity areas of the cells should be defined so that the volumes of the cells are equal to the volumes of the regions they are intended to model. In the continuity solution, cells 6 and 7 model the tapered region. The momentum cells use the actual area at the location of the momentum cell center. In this example, the momentum area variation is modeled in cell 6.

The wetted perimeter is defined in the momentum cell, so variations in wetted perimeter must be located relative to the momentum cells. In example (a) in Figure 3.18, the gradually changing wetted perimeter must be approximated by step changes, as is the gradually changing flow area in the taper. For the circular cross section of this example, the simplest approach is to define the wetted perimeter as the perimeter of the momentum cell:



(b)

FIGURE 3.18. Examples of Vertical Variations in Continuity and Momentum Area, and Wetted Perimeter of a Channel

$$PW = \pi D^4 \quad (3.4)$$

where $D^4 = \left[\frac{4A_{\text{momentum cell}}}{\pi} \right]^{1/2}$

The input for group 5 to model the variations in example (a) of Figure 3.18 consists of three variation tables (NAFACT = 3, read on card VESSEL5.1). The tables themselves are read on card VESSEL5.2 (specifying the number of entries in a table) and on card VESSEL 5.3 (filling the arrays for the node indices, JAXL, and variation factors, AFACT). The tables are numbered sequentially in the order they are read in.

Example (b) in Figure 3.18 shows a different sort of area variation. The orifice plate in the straight pipe affects the momentum solution, but because it occurs over a relatively short distance compared to the node length, the continuity solution is largely unaffected. This could be modeled with a momentum area variation alone without any variation in the continuity cell area.

Gap width variations are specified in the same manner as vertical variations in area or wetted perimeter. The nodes that differ from nominal are identified in the JAXL array, and the corresponding variation factors in the AFACT array.

Variations encountered in most problems for COBRA/TRAC will not be as neat, generally, as the two examples shown in Figure 3.18. The user will have to make approximations appropriate for the particular geometry involved and the noding selected. Some general guidelines in setting up vertical variations are

- In complex geometries, the user should preserve the same fluid volume as in the actual system. (Volume is calculated as $A \cdot \Delta X$ in a cell.)
- The momentum area should approximate the actual flow area in the system at sudden changes. (This helps the code calculate the correct pressure losses due to momentum changes.)

- The code interpolates linearly in the table to obtain variation factors for cells within the range of the table but not named explicitly. (Cells with identification numbers lower than the first element of a variation table or greater than the last element in the table remain at nominal values.)
- The area in either the continuity cells or the momentum cells should not change by more than a factor of two between adjacent cells, even if the area in the system actually changes more abruptly. The code must be led gradually through a large change in a series of steps.

The input for group 6 specifies the channels or gaps to which the variation tables described in the group 5 input apply. Variation table assignments are read on card VESSEL6.2 for either channels or gaps. The variation tables are identified by sequence number; i.e., the first table read in group 5 is table 1; the second table read in group 5 is table 2; etc. The index number of the table to be used for continuity area variations is specified by IACT. The index number of the table to be used for momentum area variations is specified by IAMT. The index number of the table to be used for wetted perimeter variations is specified by IPWT. The numbers of the channels using the variation tables named by IACT, IAMT and IPWT are listed in array ICRG(M). Axial variation in gap width is specified by setting IACT to the negative of the variation table number, and naming the indices of the gaps using the table in array ICRG(M). When gap variations are specified, IAMT and IPWT are not used.

The input for card VESSEL6.2 is repeated until all channels and gaps having variations have been identified. It is not necessary to specify variations in continuity area, momentum area and wetted perimeter in a channel simultaneously. Any or all can be used, as appropriate for a given problem. It is possible to specify a continuity variation only, momentum area variation only, perimeter variation only, or any combination of the three.

3.1.5 Group 7: Vertical Loss Coefficients

The input for group 7 is designed to model local pressure losses in the vertical flow due to spacer grids, orifice plates and other local obstructions in the flow field. The pressure loss is modeled in the code as a velocity head loss

$$\Delta P = C_D \rho \frac{V^2}{2g_c} \quad (3.5)$$

The input for group 7 defines values for C_D and specifies where they should be applied.

The total number of loss coefficients to be specified in group 7 is the value assigned to variable NCD on card VESSEL7.1. The loss coefficient values, CDL(N), are specified on card VESSEL7.2 along with the index number of the node corresponding to the vertical location of the loss. The indices of channels that contain that particular pressure loss are specified in the array ICDUM(I). Figure 3.19 shows two examples of arrays of channels with local pressure losses that can be modeled by using group 7 input. Example (a) consists of a rod array with three spacer grids having a measured loss coefficient of 1.2. The input in group 7 for this example is shown below:

Card 7.1	7	3 (number of cards to be read)		
			<u>Loss Coefficient (CDL)</u>	<u>Axial Node (J)</u> <u>Channels(ICDUM(I))</u>
Card 7.2			1.2	3 1 2
Card 7.2			1.2	6 1 2
Card 7.2			1.2	8 1 2

Example (b) consists of an external downcomer, lower plenum and lower part of a vessel--a model of part of an experimental test section. Local losses are caused by the turbine flowmeter in the downcomer pipe and the grid spacers in the vessel. The group 7 input for this example is

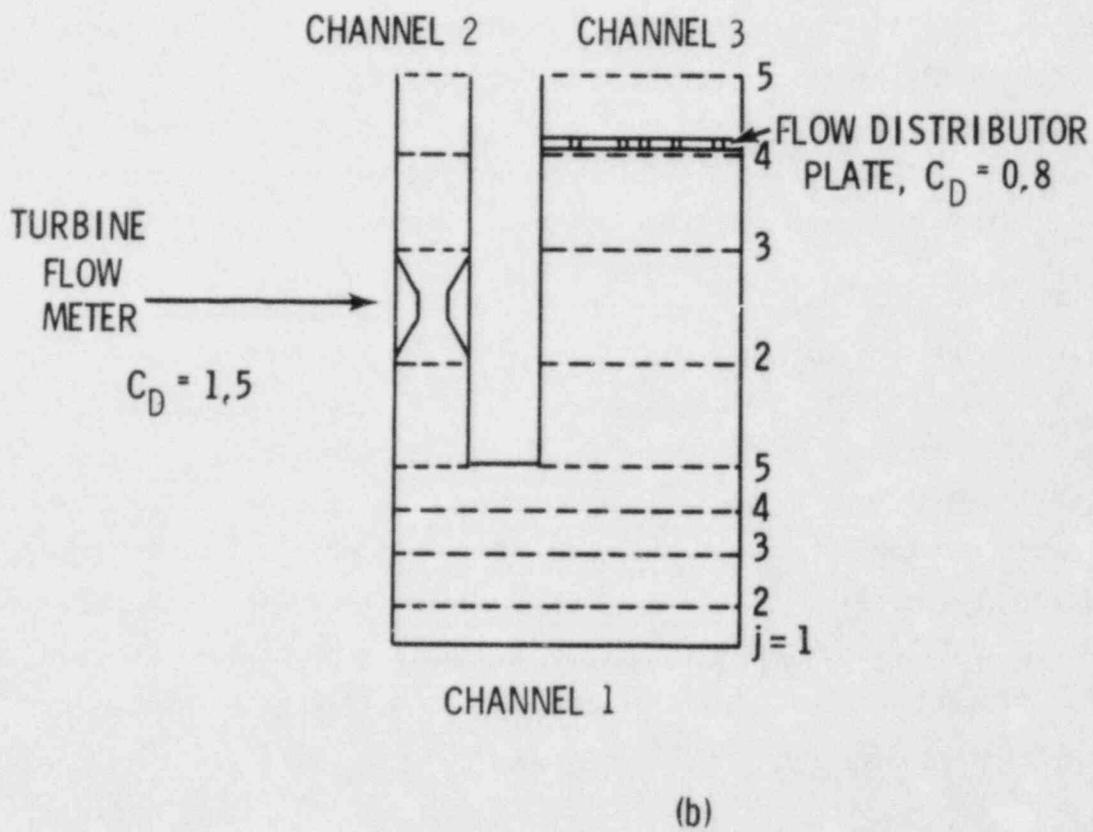
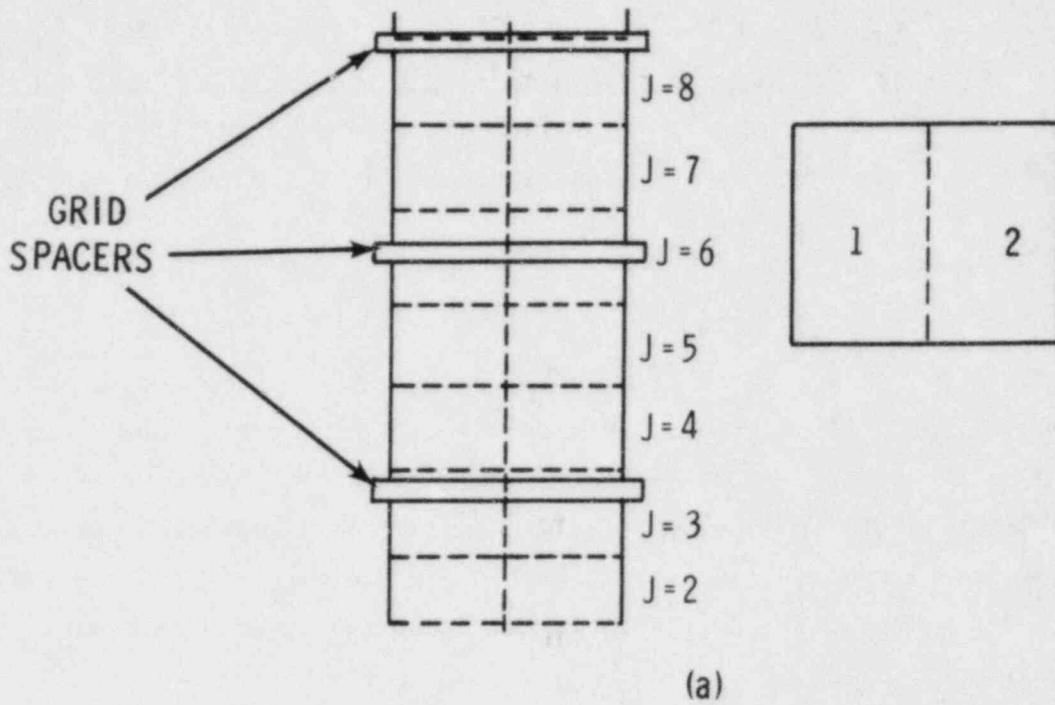


FIGURE 3.19. Examples of Channels with Local Form Losses

Card	7	2 (number of cards to be read)			
			<u>Loss Coefficient (CDL(N))</u>	<u>Axial Node (J)</u>	<u>Channels(ICDUM(I))</u>
Card 7.2			1.5	3	2
Card 7.2			0.8	4	3

As these two examples illustrate, the input for group 7 is relatively simple and easy to construct. There are two main points to remember:

- The location of a loss coefficient is determined by node and channel number. The node refers to the momentum cell, not the continuity cell. This must be kept in mind when determining the node that corresponds to the location of the local loss in the system being modeled. Care must be used when placing a loss coefficient in a momentum cell at a section boundary. The loss must be defined in the channel where the momentum equation is solved.^(a)
- The loss coefficients are defined assuming positive upflow in the channel. If the loss coefficient of a particular structure changes significantly when flow reverses through it, the code does not see the change. If reverse flow is the dominating pattern for such a situation, the user should specify the loss coefficient corresponding to reverse flow rather than the value for positive flow.

3.1.6 Group 12: Turbulent and Viscous Stress Data

Viscous and turbulent shear stresses and heat transfer may be modeled through the input provided in card group 12. This input is optional, and should be used only for problems where these shear stresses have a significant effect on the behavior of the system. Turbulent and viscous shear stresses may be specified for the entire mesh, or for only specific sections of the mesh as the user requires. Generally these terms should be neglected when performing a full system calculation, as the coarseness of the mesh does not allow for an adequate resolution of the velocity profiles that are required to determine the viscous or turbulent shear stresses.

^(a) To determine which channel the momentum equation is solved in refer to Section 3.1.1.

The user may wish to perform a separate detailed three-dimensional simulation of some region of the vessel where turbulent and viscous shear stresses are important. Turbulent shear stresses are important when modeling problems involving lateral void drift and should be used when such phenomenon is important. The viscous and turbulent shear stress calculations are activated in each section of the mesh by specifying the appropriate value for IVIS(ISEC) and ITURB(ISEC). If ITURB(ISEC) is set to a value of one (1) then turbulence properties are assumed constant over the section. If ITURB(ISEC) is set to a value of two (2) then turbulence properties are assumed to vary from channel to channel. A mixing length turbulence model is used in the code. (This model is described in detail in Volume 1.) The user must specify the mixing length, the anisotropy factor and the turbulent Prandtl numbers for each phase, in each section (if ITURB(ISEC) = 1) or in each channel of a section (if ITURB(ISEC) = 2). The mixing length for a problem is dependent on the geometry of the flow path. For rod bundles a mixing length on the order of the hydraulic diameter of a subchannel is recommended.

While the input allows the user to specify the degree of anisotropy, this is not usually known, and isotropic flow is usually assumed when computing the normal component of the turbulent shear stress. Thus, the anisotropy factors should generally be set to one, as are the turbulent Prandtl numbers, unless the user has in hand convincing data to the contrary.

3.2 VESSEL COMPONENT ROD AND UNHEATED CONDUCTOR MODEL

The conduction heat transfer model for the VESSEL component of COBRA/TRAC is designed to model nuclear fuel rods, electrically heated elements, and the various walls, tubes, canisters and pipes composing a reactor vessel or test section. The numerical procedure used in the conduction solution is based on a finite-difference resistance network approach. It is discussed in detail in Volume 1 of this manual, Equations and Constitutive Models. Basically, the conduction solution calculates the heat flow in two dimensions in a solid structure, with heat transfer to the fluid as a boundary condition on the heat transfer surface. Heat flow in the solid is calculated in the radial direction and (optionally) in the axial direction.

The input data for the conduction model is specified in groups 8, 9, 10 and 11. The input for group 8 identifies the rods and unheated conductors for a specific problem and initializes the surface temperatures. The geometries of the various rods and unheated conductors are described in group 9. There is an optional set of input data in group 9 for a dynamic gap conductance model for rods modeling nuclear fuel pins. The thermal properties of the materials modeled in the rods and unheated conductors are specified in the input data for group 10. Axial power profiles for rods and a forcing function on total power are specified in group 11. The input for each of these groups is discussed in detail, with illustrative examples, in the following subsections.

3.2.1 Group 8: Description of Rods and Unheated Conductors

The input data for group 8 identifies the rods and unheated conductors modeling the solid structures that interact significantly with the fluid in a particular problem.

Rods and unheated conductors are both used to model solid conducting structures in the vessel. There are two significant differences between them, however, one conceptual, the other numerical. Rods can model either active or passive elements, but unheated conductors are always passive. Unheated conductors cannot have internal heat sources. The quench front model with fine-mesh renoding can be applied to rods if needed, but unheated conductors

are assumed to never require it. This greatly simplifies the heat transfer logic for unheated conductors and reduces the computer storage required to model solid elements that will not experience dryout and quenching.

Figure 3.20 shows examples of VESSEL structures that can be modeled with rods. Example (a) is an array of nuclear fuel pins; example (b) is a similar array of electrically heated fuel pin simulators; example (c) is an electrically heated annular test section with both cylinders heated; and (d) is a test section with an electrically heated flat plate as the heat source. Figure 3.21 contains examples of VESSEL structures that can be modeled with unheated conductors. Example (a) is a control rod guide tube (from the upper plenum and upper head of a PWR); example (b) is a support column; example (c) is a section of a downcomer annulus (formed by the core barrel and pressure vessel); and example (d) is a one-eighth section of symmetry of the canister of a BWR fuel assembly.

The first line of input in group 8, card VESSEL8.1, identifies the number of rods in the problem (NROD) and the number of unheated conductors (NSROD). The conduction solution flag NC must be set at this point to select either radial conduction only (NC = 1) or radial and axial conduction (NC = 2). The option for radial and axial conduction is recommended only for problems where a significant axial temperature gradient is expected in the rods. Problems involving reflooding and quenching of very hot surfaces definitely should use the axial conduction option. Problems involving relatively gradual heating or cooling of the system probably can be handled adequately with radial conduction only.

The rod and unheated conductor identification parameters are read on cards VESSEL8.2 and VESSEL8.3, respectively. If NROD is zero, card VESSEL8.2 is omitted from the input; similarly, if NSROD is zero, card VESSEL8.3 is omitted. The rods are numbered sequentially from 1 to NROD, and the unheated conductors are numbered sequentially from 1 to NSROD. Each rod or unheated conductor is uniquely identified by its index number and geometry type number (IFTYP(N) for rods and ISTYP(N) for unheated conductors). The geometry type number corresponds to a set of descriptive geometry data specified in input group 9. An individual rod or unheated conductor may have a unique geometry type, or several rods or conductors may be of the same geometry type. Rods

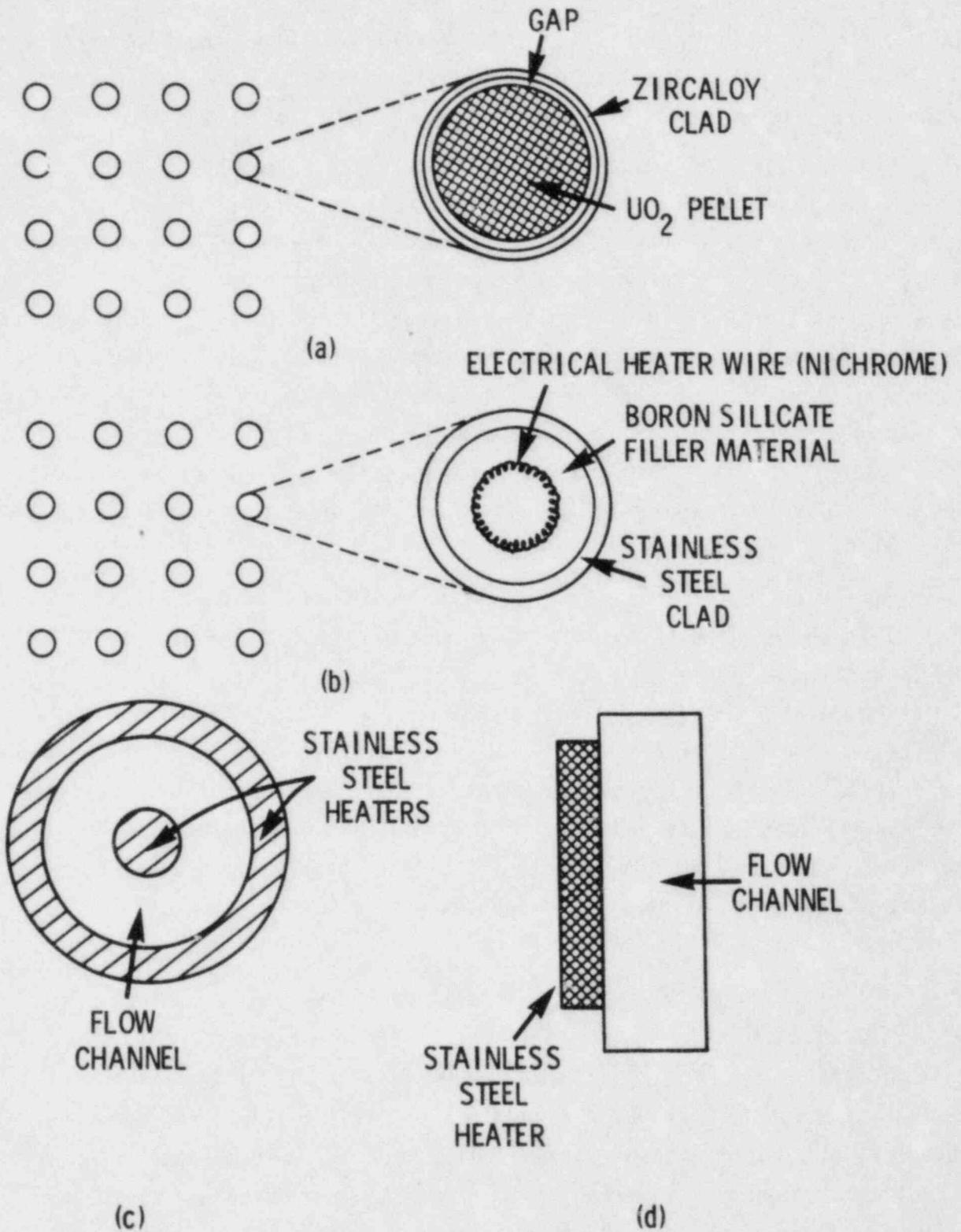


FIGURE 3.20. Examples of Heat-Generating VESSEL Structures

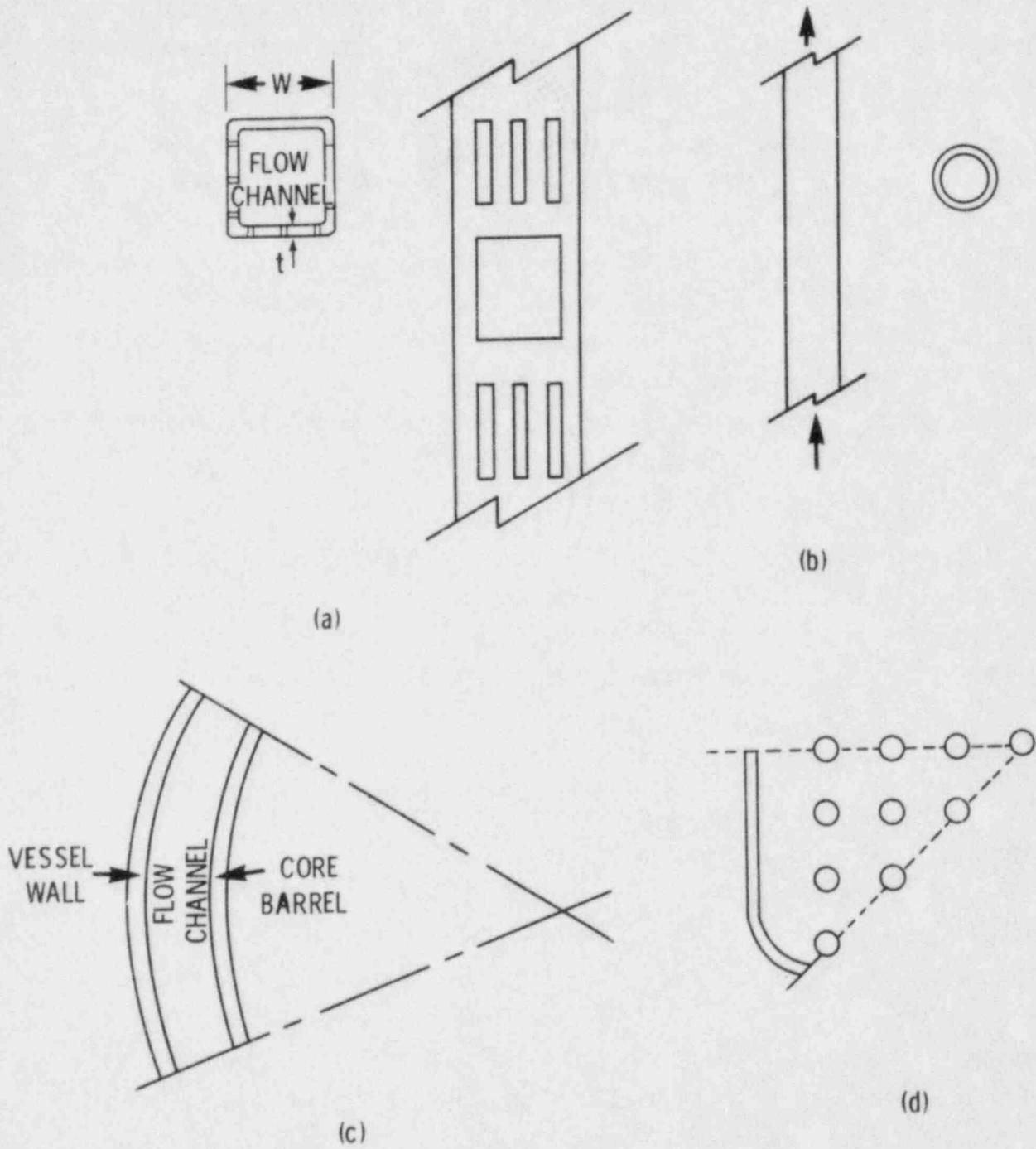


FIGURE 3.21. Examples of Unheated VESSEL Structures

and unheated conductors can use the same geometry type, but any heat generation specified for that geometry type will be ignored for the unheated conductors. (The group 9 input is discussed in detail in Section 3.2.2.)

Each rod has identified with it the number of an axial power profile table, IAXP(N). If IAXP(N) is left blank, an axially uniform table with a factor of unity is assumed. The axial power profile tables are specified in group 11, and the form of this input is discussed in Section 3.2.4. An axial power profile table can serve any number of rods. In addition to the axial power profile table, each rod has associated with it a radial power factor, RADIAL(N). The radial power factor is normalized to the average power generation rate (specified in group 1, see Section 3.3.1). In conjunction with the axial power profile table, RADIAL(N) defines the local power generation in the individual rod.

Unheated conductors do not generate heat but can transfer heat to and from the fluid and store thermal energy during a transient. For each unheated conductor the user must specify the heated perimeter of the surface, HPERIM(N). If the conductor is a tube or wall that has contact with the fluid on its inner surface, then the heated perimeter of the inner surface, HPERIMI(N), must be specified as well.

In modeling large geometries it is sometimes convenient to represent regions of the vessel by average rods or unheated conductors. A fuel pin array such as the one shown in Figure 3.20(a), might be represented as a single average rod. The upper plenum of a PWR contains a large number of guide tubes, such as the one shown in Figure 3.21(a), but the guide tubes in a given region of the plenum could be modeled by a single average unheated conductor. The number of actual structures modeled by a rod or unheated conductor is specified in the variable RMULT(N) for the rod and in RMULS(N) for the unheated conductor. (The values specified for RMULT and RMULS can include fractional parts of structures.)

The fine-mesh renoding capability developed to resolve the quench front in reflooding requires some extra input for the rods on card VESSEL8.2; (this input has no counterpart on card VESSEL8.3 for the unheated conductors). If fine-mesh renoding is to be used for a particular rod, the flag NRENODE(N) is

set to the number of calculational time steps to elapse between renoding. How often the rod should be renoded is primarily a function of the reflood rate and the size of time steps expected during the reflood portion of the transient. In general, NRENODE(N) should be set so that the quench front will not progress further than 1/2 of the minimum node size (DAXMIN(N)) between renoding. For example:

quench front velocity:

$$V_{QF} = 0.25 \text{ (in./sec)}$$

maximum time step:

$$\Delta t = 0.05 \text{ (sec)}$$

minimum node size:

$$DAXMIN = 0.05 \text{ (in.)}$$

$$NRENODE = \text{INT} \left(\frac{\frac{1}{2} (0.05)}{(0.25)(0.05)} \right) = 2 \quad (3.6)$$

If the rod is a tube quenching on the inside surface, then NRENODE(N) should be specified as a negative number, and the absolute value of NRENODE(N) is used to determine the renoding interval.

If renoding is specified for a particular rod by a nonzero value of NRENODE(N), the user must specify the minimum node size, DAXMIN(N), for that rod. The normal size of the axial nodes in the rod is the same as that of the fluid nodes. These are usually quite large, on the order of 12 to 24 in., but at the quench front the node length must be on the order of fractions of an inch to resolve the temperature gradient adequately. As a general rule, DAXMIN(N) should be about 0.05 inches.

The remainder of the input in group 8 sets up the initial surface temperatures of the rods and unheated conductors. The temperatures are specified in tables of initial temperature, TRINIT(I,L), versus axial distance, AXIALT(I,L), relative to the bottom of the vessel. The code interpolates linearly in the table along the axial length of the rod or unheated conductor it is applied to, so the first element of the AXIALT(I,L) array for a given table I must be at or below the bottom of the rod, and the last element must be at or above the top of the rod. A total of NRTAB tables

(the value of NRTAB is specified on card VESSEL8.1) must be supplied, but table I can be applied to more than one rod or unheated conductor. On card VESSEL8.4 the user must specify the number of rods $NRT(I)$, and the number of unheated conductors $NST(I)$ that use table I. The user must also identify on this card how many pairs of $TRINIT(I,L)$ and $AXIALT(I,L)$ elements make up this particular table. The $NRAX(I)$ pairs of entries for table I, $TRINIT(I,L)$ and $AXIALT(I,L)$, are read on card VESSEL8.7.

If table I is to be used to initialize any rods, card VESSEL8.5 is read to fill array $IRTAB(I,L)$ with the identification numbers of the rods. If any of the rods are tubes or walls and the table is to be used to initialize the temperatures on the inside surface, the rod identification number, N , is entered in the $IRTAB$ array as its negative. (For example, if rod 9 is a tube to be initialized on its inside surface from table 3, then $IRTAB(3,1) = -9$.) If table I is to be used to initialize any unheated conductors, card VESSEL8.6 is read to fill array $ISTAB(I,L)$ with the identification numbers of the unheated conductors. The code assumes an initially flat temperature profile in unheated conductors, so it is not necessary to specify whether the temperature applies to the inner or outer surface.

3.2.2 Group 9: Geometry Description for Conduction Model

The input for group 9 describes the characteristics of the geometry types identified in the $IFTYP(N)$ and $ISTYP(N)$ arrays for the rods and unheated conductors in group 8. Geometry types fall into two basic classes--nuclear fuel rods and all other conductors. The nonnuclear conductors can be characterized as solid cylinders, hollow tubes or flat plates.

On the first line of input for group 9, the user must specify $NFUEL$, the number of geometry types. This must be equal to the number of unique entries in the $IFTYP(N)$ and $ISTYP(N)$ arrays. Two flags for fuel relocation, $IRELF$ and $ICONF$, are also input on card VESSEL9.1. Relocation is an option in the dynamic gap conductance model for nuclear fuel geometry types.

Nuclear Fuel Rod Geometry Types

Data for nuclear fuel rod geometry types are read on card VESSEL9.2 with additional optional input specified on cards VESSEL9.3 through VESSEL9.5.

More than one nuclear fuel geometry type can be specified for a problem, but generally nuclear fuel rods in a core or test section are uniform enough for all to be modeled by the same geometry type. The input on card VESSEL9.2 sets the geometry type index, I, which must correspond to the IFTYP(N) value of the rods that are of this geometry type. The geometry type flag FTYPE(I) for nuclear geometry types is NUCL. The geometry of the nuclear fuel rod type is defined by the outside diameter, DROD; the pellet diameter, DFUEL(I); the diameter of the central core, DCORE (zero if the fuel is solid); and the clad thickness, TCLAD. The code assumes that the rod is uniform in the axial direction, so this data completely characterizes the physical dimensions of the rods being modeled by this geometry type.

On card VESSEL9.2, NFUEL defines the number of radial heat transfer nodes in the fuel pellet. NFUEL must be large enough for the code to resolve the temperature profile in the fuel pellet adequately, yet the more nodes specified, the harder the code has to work in the heat conduction calculation. Figure 3.22 shows a fuel pellet with a typical radial temperature profile modeled with 3, 6 and 9 radial nodes. By inspection it appears that 3 nodes probably are not quite enough; 6 nodes give a reasonable approximation; and more than 9 would be too many. The user must exercise a degree of judgement in determining the number of radial nodes appropriate for the problem.

The remainder of the input card VESSEL9.2 sets flags and options for the fuel material properties and the gap conductance model. The code contains properties for UO_2 fuel^(a) derived from MATPRO-11 (Revision 1) (Ref. 8). These can be flagged by setting IMATF to zero. The correlations from MATPRO-11 use the fuel theoretical density as a parameter in calculating the UO_2 properties, so if IMATF is zero, the user must supply an appropriate value for FTDENS(I). This is a fractional value, that depends on the properties of the UO_2 pellets being modeled, and is usually on the order of 0.95. Alternatively, the user may elect to specify a different material for the fuel, in which case IMATF must be set to the numerical index of a material

(a) See Volume 1 of this manual for a complete discussion of the constitutive relations and models used in the conduction solution.

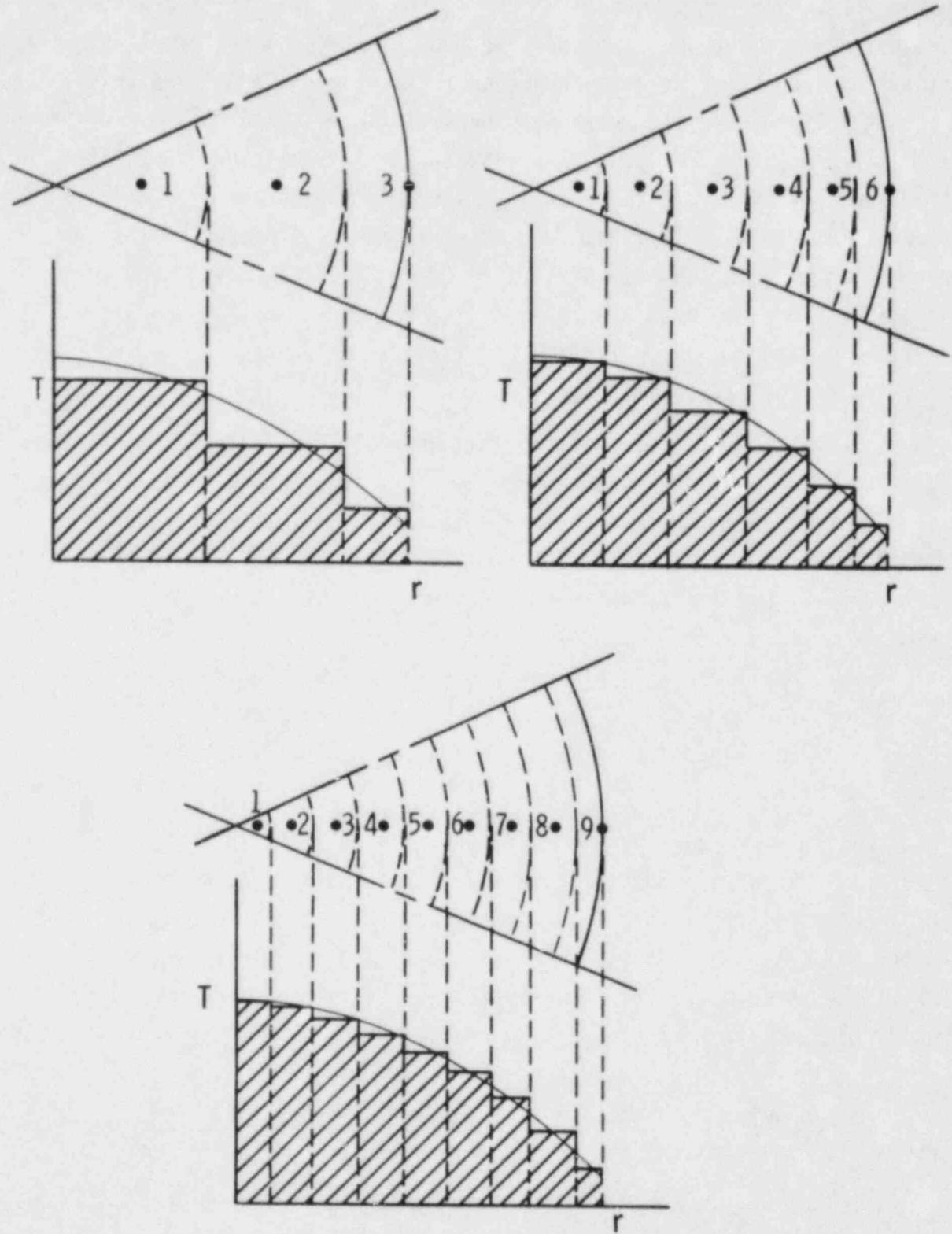


FIGURE 3.22. Fuel Pellet Radial Noding for the Conduction Model

property type specified in group 10. (See Section 3.2.3 for discussion of user-specified material properties.) For cladding material properties, the code contains properties for zircaloy and zircaloy oxide from MATPRO-11 (Revision 1), and these can be flagged by setting IMATC and IMATOX(I) to zero. If the cladding or cladding surface oxide is some other material for the nuclear rods being modeled by geometry type I, IMATC and IMATOX(I) must be assigned the numerical index of an appropriate material property type specified in group 10.

There are three options available for the gap conductance model for geometry type I: constant uniform gap conductance, axially nonuniform user-specified gap conductance, or a dynamic gap conductance model. The simplest of the three alternatives, and the easiest to implement, is the constant uniform gap conductance. It is flagged by setting IGPC to zero. The gap conductance value specified on card VESSEL8.2 in HGAP(N) is used for rods with geometry type I, and no further input is required for the gap conductance.

The second option for a user-specified nonuniform gap conductance is a variation of the first option. The user specifies the value for gap conductance but can vary it axially and in time by means of input tables. The user specifies a table of pairs of values of axial location, AXJ(L) and gap conductance, AGFACT(L,I), on card VESSEL9.4. (IGPC on VESSEL9.2 defines the number of pairs of entries in this table.) The code determines the gap conductance in each axial node of the rods with geometry type I by linear interpolation in the table specified on card VESSEL9.4 and applies the gap conductance temporal forcing function (see card group 11, Section 3.2.4), if one has been specified.

In the dynamic gap conductance model, the code calculates the gap conductance based on the thermal and geometric properties in the gap. The model takes into account the thermal expansion of the fuel pellet and cladding and elastic deformation of the cladding in determining the size of the gap. The dimensions of the gap and the thermodynamic properties of the fill gas are used to calculate the gap conductance. The user must specify the initial cold gap width in a table of axial location, AXJ(L), versus cold gap width, AGFACT(L,I), read on card VESSEL9.4. The cold gap width is the gap width

before the fuel pin is brought to full power. But since the gap conductance model does not calculate the effects of the power history of the rod, the cold gap width specified by input must include any changes from as-built conditions due to burnup-dependent factors. The number of entries in the table read on card VESSEL9.4 is the absolute value of IGPC since IGPC is entered as a negative number to flag the dynamic gap conductance model.

In addition to the cold gap width, the user must supply data on the composition of the fill gas and the internal characteristics of the gap for the dynamic gap conductance model. This data is read on card VESSEL9.3. The cold pin gas pressure, PGAS(I), is the gas pressure at 295°K. The gas plenum volume, VPLEN(I), is the volume of the extra space between the top of the fuel pellet stack and the top of the rod. Both PGAS(I) and VPLEN(I) should be determined from the manufacturing specifications of the particular fuel rods being modeled by geometry type I. The surface roughnesses of the fuel pellet and the cladding inner surface, ROUFF(I) and ROUFC(I), are used in the temperature jump discontinuity correlation in the gap conductance model. The correlation was optimized with values of $0.2(10^{-4})$ inches and $0.39(10^{-4})$ inches for ROUFC and ROUF., respectively, and even if the user knows the correct values the optimized values will probably yield a more accurate answer.

The fuel relocation model from FRAPCON-2 can be included in the dynamic gap conductance calculation. Relocation allows fuel to move radially into the fuel-cladding gap. Cracks formed in the fuel by relocation reduce the effective conductivity of the fuel. When IRELF=1, both radial relocation and conductivity degradation are included in the calculation. Since the relocation model is an empirical correlation, the surface roughnesses should correspond to those used to derive the correlation. ROUFC and ROUFF are $0.45(10^{-4})$ inches and $0.85(10^{-4})$ inches respectively, in FRAPCON-2. (These values are different from the surface roughnesses used to optimize the temperature jump discontinuity correlation.) Conductivity degradation can be calculated without relocation by setting ICONF = 1 and IRELF = 0. In this case the effect of radial relocation should be included in the specified cold gap width.

The code expects the fill gas to be composed of helium, xenon, argon, krypton, hydrogen and nitrogen. The composition of the gas for a given geometry type is specified by filling the GSFRAC(I) array on card VESSEL9.3 with values for the molar fractions ($\text{GSFRAC}(I) \leq 1$) of the gases present. The GSFRAC(I) value for any one component can be any value between zero and one, inclusive, but the array must sum to unity:

$$\sum_{L=1}^6 \text{GSFRAC}(L) = 1.0 \quad (3.7)$$

For example, if the fill gas was 100 percent helium, GSFRAC(1) would be 1.0, and GSFRAC(2) through GSFRAC(6) would all be zero.

The code assumes that the radial power profile in a fuel pellet is uniform, but the user has the option of specifying a nonuniform radial profile in the pellet. If this option is selected, the pin radial power profile is read on card VESSEL9.5 as a table of relative radial location, RODP(L), versus relative power factor, POWR(L). The relative radial location is defined as

$$\text{RODP}(L) = \frac{(R(L) - \text{DCORE}/2)}{\frac{1}{2}(\text{DFUEL} - \text{DCORE})} \quad (3.8)$$

where DCORE = diameter of central void (in inches)

DFUEL = pellet diameter (in inches)

R(L) = radial location, (in inches)

The relative power factor is defined as

$$\text{POWR}(L) = \frac{\text{power at radial location } R(L)}{\text{average pin power}} \quad (3.9)$$

The profile constructed by the table of (RODP(L), POWR(L)) is automatically normalized to unity.

Nonnuclear Geometry Types

Data for nonnuclear rod and unheated conductor geometry types is read on cards VESSEL9.6 and VESSEL9.7. The geometry types are classified as solid cylinders, hollow tubes or flat walls. The physical elements of a reactor vessel or test section can be modeled with these geometries, even if they do not conform exactly to the ideal shape. The guide tube shown in Figure 3.21(a), for example, can be modeled as a flat wall, even though it is a square tube. In modeling a solid element as a rod or unheated conductor, the important aspects to preserve are:

- the surface area in contact with the fluid
- the mass available for thermal storage
- the thickness of any element (wall or tube) that has fluid contact on both surfaces.

The physical dimensions of the geometry types should be defined so that these three characteristics of the solid elements of the system are modeled with reasonable accuracy.

The nonnuclear geometry types are defined on card VESSEL9.6 by index number I, and an alphanumeric value for FTYPE(I): HROD for a solid cylinder, TUBE for a hollow tube, and WALL for a flat plate. The dimensions of the rods or unheated conductors modeled by a geometry type I are defined by DROD and DIN. For a solid cylinder geometry, DROD is the outside diameter, and DIN is zero. For a tube geometry, DROD is the outside diameter and DIN the inside diameter. For a flat plate geometry, DROD is the perimeter of the plate surface that is in contact with the fluid and DIN is the plate thickness.

In many problems, DROD and DIN will correspond exactly to the physical dimensions of the elements being modeled by the geometry type. The electric heater rods in Figure 3.20(a), for example, could be modeled with DROD equal to the actual diameter of the heaters. Similarly, the outer tube of the annulus in Figure 3.20(c) could be modeled with a geometry type having DROD

and DIN equal to the actual outer and inner diameter of the tube. But the square guide tube in Figure 3.21(a) would have to be modeled by an equivalent conductor, such as a flat plate. The guide tube, with width W and thickness t , has outer perimeter $PW_O = 4W$. A flat plate with equivalent outer perimeter would have width $DROD = 4W$, and thickness $DIN = t$.

The number of different material regions modeled in the rods or unheated conductors of geometry type I is specified by the value of NFUEL on card VESSEL9.6. The number of heat transfer nodes in a given region is specified by input on the subsequent card, VESSEL9.7. A region defines a radial ring (or layer, for wall geometries) of uniform material properties for the conduction solution. The heater rod shown in Figure 3.20(b), for example, consists of three different materials: stainless steel, boron silicate and nichrome. Four separate regions of materials are formed by the rod geometry; the central core of boron silicate, the nichrome heater wire, the outer layer of boron silicate, and the stainless steel cladding. Four material regions (NFUEL = 4) are required to describe this heater rod geometry type.

The remaining input for card VESSEL9.6 defines the material properties used to determine the minimum film boiling temperature for the heat transfer surfaces of the geometry type. Most physical systems in contact with water, particularly at high temperatures, develop an oxide layer that significantly affects the thermal properties of the surface. The user may specify the thermal properties of the oxide material on the outside and inside surface by setting IMATOX(I) and IMATIX(I) to the index number of a material properties table specified in group 10. (The variable IMATIX(I) applies only to TUBE or WALL geometries with fluid contact on the inside surface. For solid cylinders (HROD) and TUBES or WALLS with fluid contact only on the outside surface, it is ignored.) If a particular geometry type does not have oxide scale on its heat transfer surfaces, IMATOX(I) and IMATIX(I) should be assigned values that correspond to the material properties table for the material the rods or unheated conductors are composed of at the surface. If IMATOX(I) or IMATIX(I) are set to zero, the oxide property index defaults to the built-in zirconium dioxide properties table.

The material composition and radial noding in the NFUEL regions identified for a geometry type I are read on card VESSEL9.7. The user must specify a material property table index for the region MATR(L), corresponding to a properties table supplied in group 10. The physical thickness of each region must be specified in the variable TREG(L), with the number of radial nodes in that region, NODER(L). The code automatically calculates the size of each node within the region by dividing the region into NODER(L) nodes of equal thickness.

The material temperature is calculated at the center of mass of each node. The only exception to this node placement occurs for a region where one edge forms a heat transfer surface in contact with the fluid. In this case, the node closest to the surface is given only one-half the nominal thickness. The node temperature is calculated at the surface rather than the center of the half-width node.

For the heater rod with NFUEL = 4, Figure 3.23 shows a typical temperature profile in the heater rod and the radial noding in the various regions required to resolve it. Region 1, the boron-silicate in the center of the rod, must of necessity be at a uniform temperature in steady state, and since it is a relatively small region, it can be modeled adequately with a single node. The nichrome wire fills a thin enough region in the heater rod to be considered almost a line source and cannot have a significant temperature gradient (because of the thermal properties of nichrome), so this region too can be modeled with a single node. The region between the heater wire and the cladding has a very gradual slope because of the thermal properties of boron silicate, so two nodes are enough to resolve it adequately. The region defined by the steel cladding is relatively thin, but because of the thermal properties of steel, the temperature profile may have a relatively steep slope here, so two nodes are required in this region. The input for card VESSEL9.7 for this example is summarized below:

```
Region 1  NODER(1) = 1
           MATR(1) = 1   (implies that table 1 in group 10
           TREG(1) = r1 contains properties for boron silicate)
           QREG(1) = 0
```

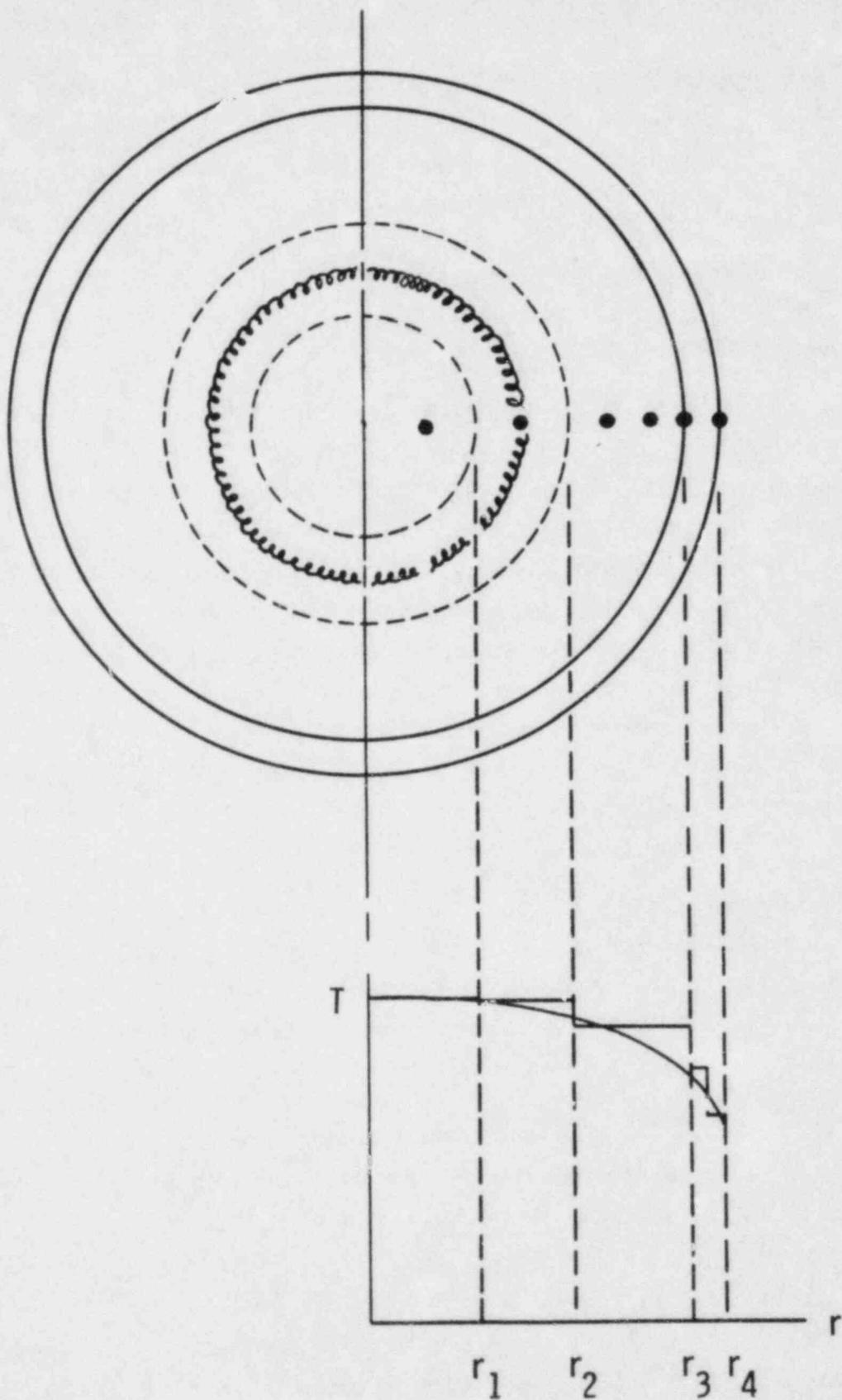


FIGURE 3.23. Radial Noding in a Typical Electrical Heater Rod

Region 2 NODER(2) = 1
 MATR(2) = 2 (implies that table 2 in group 10
 TREG(2) = (r₂ - r₁) contains properties for nichrome)
 QREG(2) = 1

Region 3 NODER(3) = 2
 MATR(3) = 1
 TREG(3) = (r₃ - r₂)
 QREG(3) = 0

Region 4 NODER(4) = 2
 MATR(4) = 3 (implies that table 3 in group 10
 TREG(4) = (r₄ - r₃) contains properties for stainless steel)
 QREG(4) = 0

The number of radial nodes to use within a region of a geometry type is governed by two factors: 1) the temperature gradient expected in the structure, and 2) the calculational cost for a large number of heat transfer nodes. The user must specify enough nodes to adequately resolve the heat transfer conditions being modeled. Beyond a certain limit, however, doubling the number of nodes does not double the accuracy of the solution, but it nearly quadruples the computation time required for elements with that geometry type. The user must exercise some judgement as to what constitutes an adequate but not excessive number of radial nodes for the heat transfer solution in a particular problem.

The radial power factor QREG(L), specified for each region, defines which regions of a nonnuclear rod generate heat internally. In heated rods, the QREG(L) values for all the regions of the rod are automatically normalized to unity, relative to the total power generated in a rod. Nonzero values of QREG are ignored in the unheated conductors.

3.2.3 Group 10: Material Properties Table

The input for group 10 supplies the material properties selected by numerical indices in the geometry type input of group 9. If all the geometry types specified in group 9 use the built-in nuclear fuel rod properties for UO₂, zircaloy and zirconium dioxide, the input for group 10 is omitted. Otherwise, properties tables must be supplied for NMAT materials, where NMAT is the number of unique indices specified in group 9 input for the variables

IMATF, IMATC, MATR(L), IMATOX(I) and IMATIX(I). The user must keep track of which material he wishes to specify with a given index number. The code cannot check this input internally.

The material properties tables are entered on cards VESSEL10.2 and VESSEL10.3. Both cards are read for each table. Card VESSEL10.2 identifies the table by its numerical index and specifies the number of entries, NNTDP, in the table. The cold-state density of the material, RCOLD(N), is also entered on this card. RCOLD(N) is the density at normal (as-built) conditions, and is used to calculate the mass of the nodes. This approach eliminates the tedious process of calculating the relocation of the radial nodes due to thermal expansion during the transient.

The properties table itself is read on card VESSEL10.3. The material properties included in the table are specific heat, CPF(I,N), and thermal conductivity, THCF(I,N), as a function of temperature, TPROP(I,N). The temperature range of the table must be great enough to encompass the temperature extremes expected in the calculation; otherwise the run will abort due to property table lookup failure. The user does may specify constant properties for a material type. This is done by entering only one element in the table; i.e, setting NNTDP to 1, and reading card VESSEL10.3 for only one set of TPROP(I,N), CPFF(I,N), and THCF(I,N). The value specified in TPROP(I,1) is in this case superfluous. The values specified in CPFF(I,1) and THCF(I,1) will be used for material type I regardless of the material temperature.

3.2.4 Group 11: Axial Power Profiles and Forcing Functions for Total Power and Gap Conductance

The input for group 11 defines the axial power profiles for the rods specified in group 8. In addition, the user may specify temporal forcing functions on total power generation rate and, for nuclear fuel rods, on the gap conductance. If no rods are specified in the group 8 input, the input for group 11 is unneeded and can be omitted entirely.

The axial power profiles and forcing functions on power and gap conductance are entered as tables of paired values--relative power versus

axial location for the power profiles, and forcing factor versus time for power and gap conductance. The first line of input on card VESSEL11.1 defines the number of axial power profile tables, NAXP, and the number of pairs of entries, NQ and NGPFF, in the power forcing function table and the gap conductance forcing function table, respectively.

The NAXP axial power profile tables are numbered sequentially, 1 through NAXP, in the order they are read in. The identification numbers must correspond to the values specified in the IAXP array on card VESSEL8.2 for the rods in group 8. Card VESSEL11.2 is read for each axial power profile table to define its index number, I, and the number of entries, NAXN(I), in the table. The table itself is read on card VESSEL11.3 as NAXN(I) pairs of Y(I,N) and AXIAL(I,N), where Y(I,N) is the axial location relative to the bottom of the VESSEL and AXIAL(I,N) is the power factor at that location. AXIAL(I,N) is defined as the ratio of local power to average power in the rod or rods the table is applied to. Figure 3.24 shows a nuclear fuel rod with a 12-foot active length and a chopped cosine power profile with peak-to-average ratio of 1.45.

The axial power table is used to interpolate linearly for axial power factors at the boundaries of the axial heat transfer cells, and the profile is integrated over each cell to obtain an average linear heat rate for the cell. When re-noding occurs, the axial power profile is re-integrated to obtain the average linear heat rate in the new cells. In the example shown in Figure 3.25, an electric heater rod with step approximation to a cosine power profile is modeled in a channel with 12-inch continuity nodes. The linear heat rate in the cell is modeled as a uniform Q_{in} along the entire length of the cell.

With channel splitting it is possible to extend a rod through several sections of the channel geometry. A rod can be used to model an average heater rod in a test section, as illustrated in Figure 3.26. Channel 3 in section 1 models a portion of the region penetrated by the cold ends of the rods modeled by the average heater rod 1. Channel 3 connects to 10 above. Channel 10, in section 2, models the portion of the core containing the rods

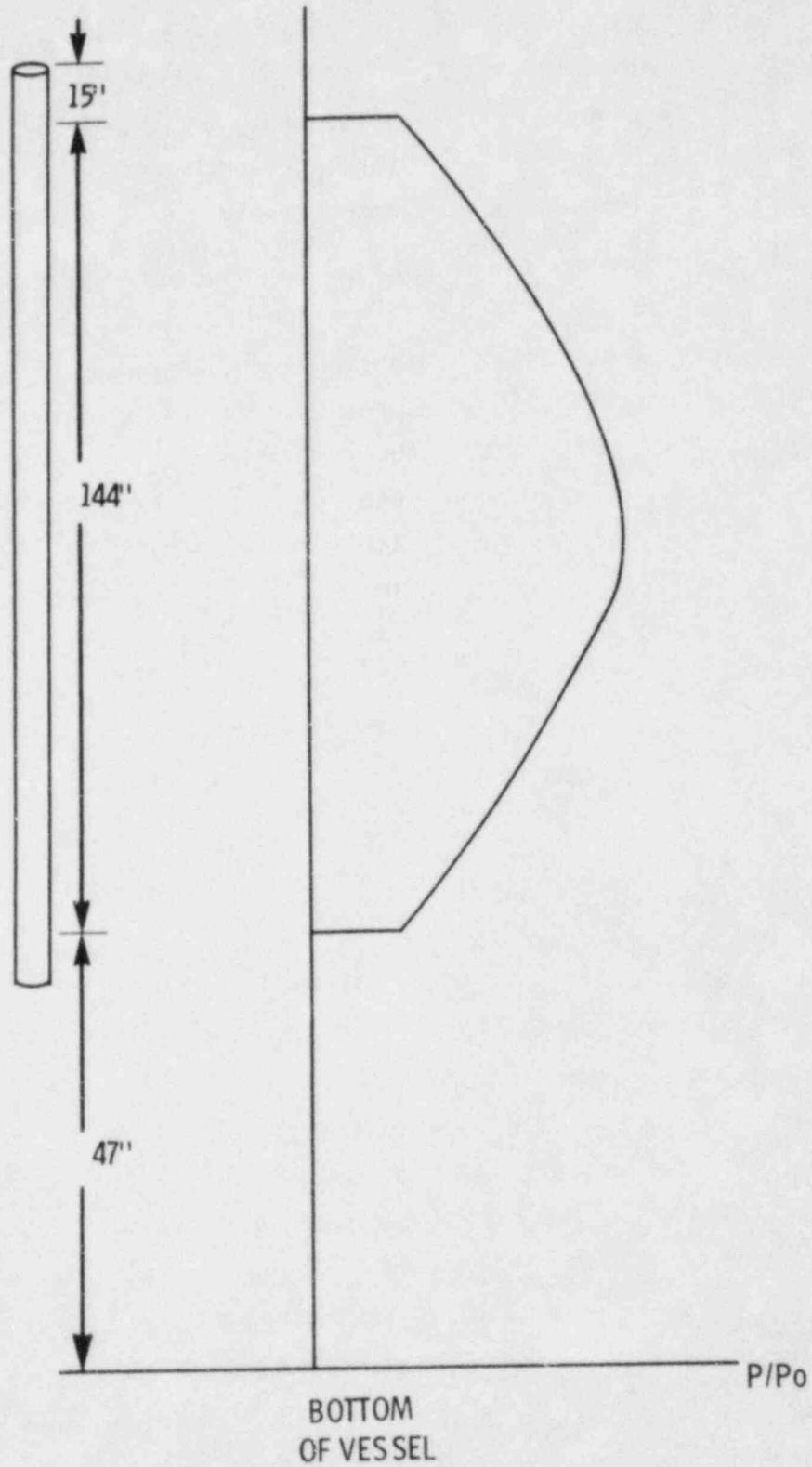
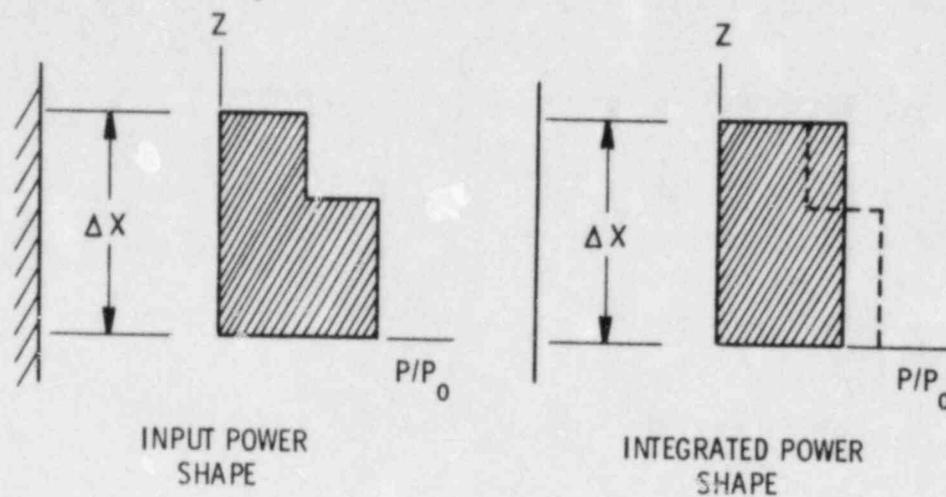


FIGURE 3.24. Nuclear Fuel Rod Power Profile



$\Delta X = \text{NODE LENGTH}$

$$Q_{in} = \int_0^{\Delta X} \frac{P(Z)}{P_0} dz$$

FIGURE 3.25. Heat Input Over One Fluid Node

modeled by heater rod 1. Channel 10 connects to channels 15, 16 and 17 above, in section 3, but the heater rod ends at the boundary between Sections 2 and 3.

In general a rod can pass through any number of contiguous vertical sections with fluid connections in different channels in each section, but the rod must begin and end at a section boundary. The vertical locations of the beginning and end of each rod must be considered when defining section boundaries for the group 4 input. Unheated conductors do not require axial power profiles and may not cross section boundaries, but they must also be considered in the section boundary locations. Unheated conductors are assumed by the code to have the same axial length as the sections that contain them.

The forcing functions specified in group 11 are read on cards VESSEL11.4 and VESSEL11.5. The forcing function on the power is entered as NQ pairs of transient time YQ(N) and power factor, FQ(N). The power factor FQ(N) is the ratio of power at time YQ(N) and the initial power. The code interpolates linearly in the power forcing function table to determine the current value of

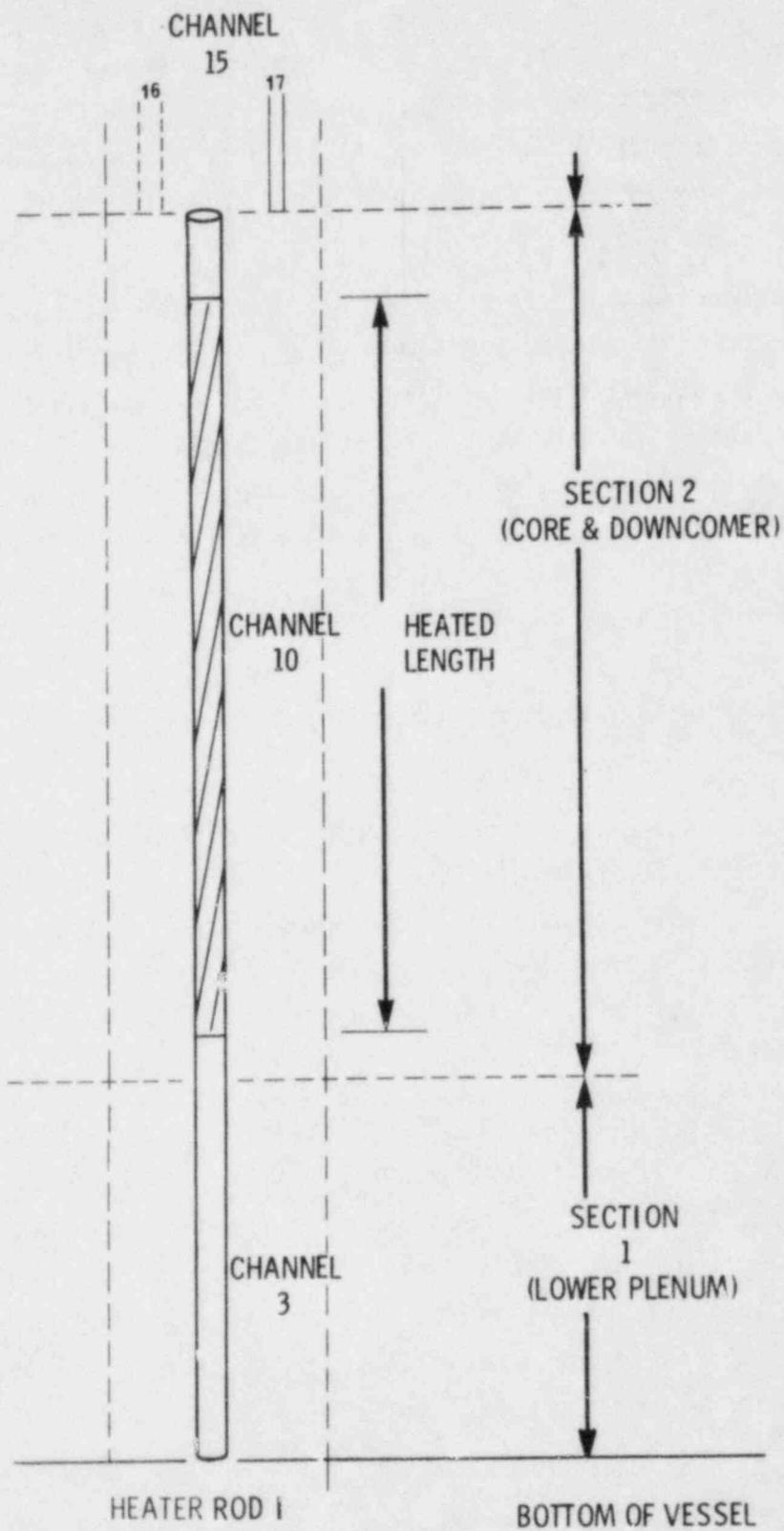


FIGURE 3.26. Heater Rod Crossing Section Boundaries

total power at each time step. Figure 3.27 shows an example of how the code interpolates a power forcing function. (The relative size of the time steps has been greatly exaggerated on graph (c) for illustrative purposes. In reality, the calculated power values over the time steps sum to a value very nearly equal to the integral of the power table curve in graph (b).)

The gap conductance forcing function is specified on card VESSEL11.5 as NGPFF pairs of transient time, YGPFF(N) and gap conductance factor, FGPF(N). The input convention for this table follows the same logic as the other tables in group 11. The gap conductance factor FGPF(N) is defined as the ratio of the gap conductance value at the time YGPFF(N) and the initial value. The forcing function is applied uniformly along the axial length for nuclear fuel rods with a gap conductance forcing function specified on card VESSEL9.2, i.e., with IGPC > 0.

3.3 BOUNDARY AND INITIAL CONDITIONS FOR THE VESSEL COMPONENT

Boundary and initial conditions for the VESSEL component consist of the initial system operating conditions, boundary conditions on individual computational cells within the VESSEL, and the junction connections that tie the vessel to the other components of the system. The initial system operating conditions define the state of the VESSEL component at the beginning of the transient. This data is supplied in input group 1, which is discussed in Section 3.3.1.

Within the vessel component itself the user may specify a variety of boundary conditions on individual computational cells. These include pressure boundaries, mass flow boundaries, mass sources and pressure sinks. In addition, the user may selectively shut off crossflow in specific nodes, a useful aid in modeling unusual transverse flow paths. The data for these boundary condition options are specified in group 13, and the input requirements are discussed in detail in Section 3.3.2.

The input data to specify the VESSEL connections to other components is discussed in Section 3.3.3. The input for this group is relatively simple, but there are several important and somewhat less-than-obvious factors the user must consider.

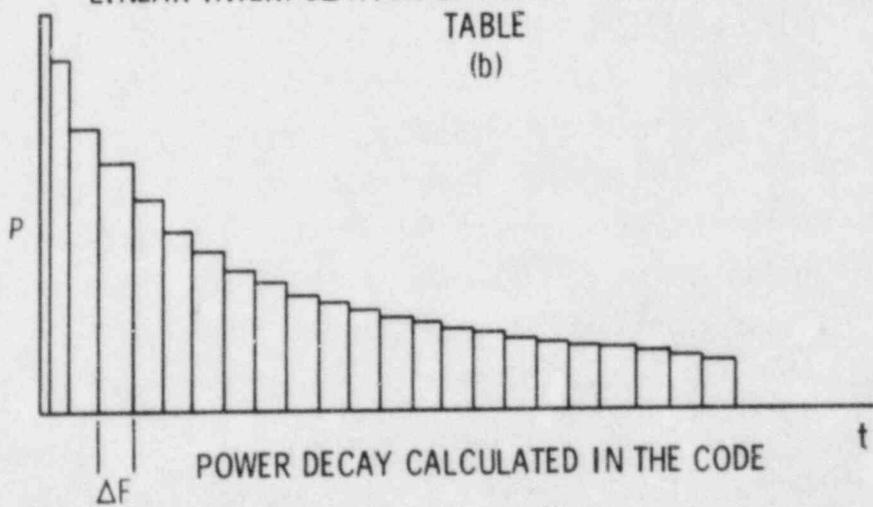
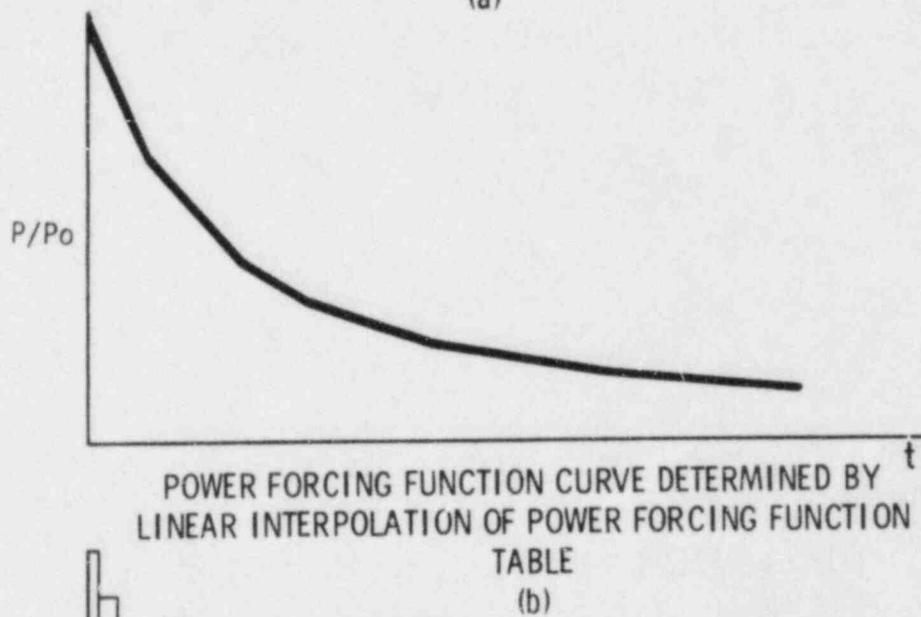
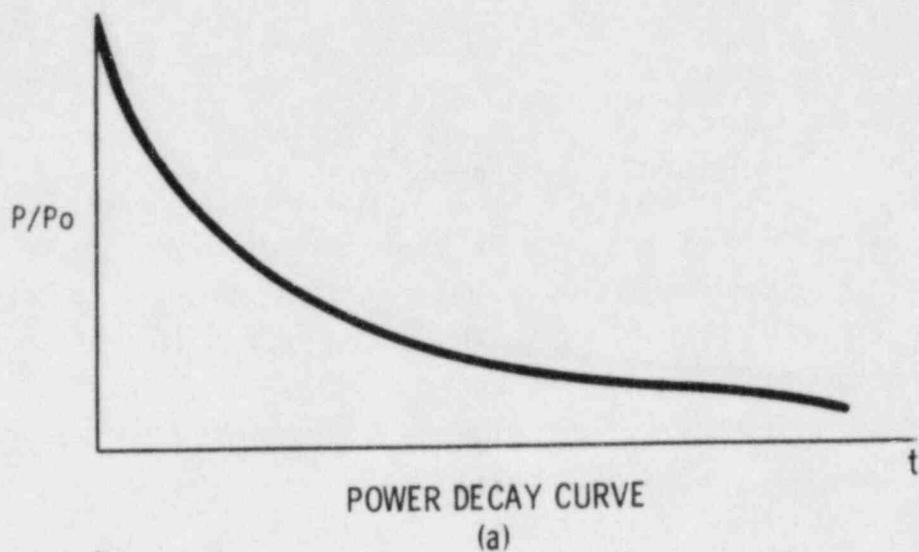


FIGURE 3.27. Power Forcing Function

3.3.1 Group 1: Initial Conditions

The input for group 1 contains two seemingly redundant cards for the geometry data, and a card to initialize operating conditions in the VESSEL. The geometry cards, VESSEL1.1 and VESSEL1.2, require the user to specify the total number of axial nodes (NDX) and the total vertical height (Z) of the VESSEL. The values specified on these cards must be compatible with the section-by-section input supplied in group 4. That is:

$$NDX = \sum_{N=1}^{NSEC} NONODE(N) \quad (3.10)$$

where $NONODE(N)$ = number of axial nodes in section N
 $NSEC$ = total number of sections

Similarly, the vertical height, Z, is

$$Z = \sum_{N=1}^{NSEC} NONODE(N)*DXS(N) \quad (3.11)$$

where $NONODE(N)$ = number of axial cells in section N
 $DXS(N)$ = axial cell length in section N

The initial operating conditions in the VESSEL are read on card VESSEL1.3 and consist of system pressure (PREF), fluid enthalpy (HIN), mass flux into the system (GIN), and the average power generation rate (AFLUX), expressed as an average linear heat rate in kW/ft or kW/m. The initial pressure (PREF) and enthalpy (HIN) are used to determine the initial properties of the fluid in the VESSEL. During the calculation the enthalpy and pressure change with the system behavior, depending on the state of fluid flowing in or out of the vessel from other components, and the local heat generation rate.

The initial mass flux specified in GIN for the VESSEL is not, in general, compatible with the initial pressures, which are calculated based on gravity head losses only. The code must iterate to a correct solution. It is usually

most expedient to specify the initial mass flux as zero; i.e., a "standing start," unless the problem involves single-phase flow at a nominally uniform flow rate in the system.

COBRA/TRAC does not currently have a steady-state implicit solution capability. Since many of the transients that the code will be used to calculate begin from a steady-state condition, the problem must be initialized with a transient calculation that brings the system from the initial conditions specified by GIN, HIN and PEF to the required steady-state condition. This is a relatively simple process, but it requires that the user know what he is doing. Section 3.6 discusses the recommended procedure.

The initial power in the vessel is specified by the variable AFLUX on card VESSEL1.3. Unless modified by a power forcing function in group 11, the initial value of AFLUX remains constant throughout the transient. AFLUX is the average linear heat generation rate per active rod. The power generation rate in the active nodes of a rod is given by

$$Q'_n = \text{AFLUX} * \text{RADIAL}(N) * \text{AXIAL}(J,N) \quad (3.12)$$

where $\text{RADIAL}(N)$ = normalized radial power factor for rod N
(from input group 8)

$\text{AXIAL}(J,N)$ = normalized axial power factor for rod N
(from axial power profile for rod N) at axial location J

The total linear power generation rate for a given VESSEL is

$$Q'_T = \sum_{m=1}^{\text{NROD}'} \sum_{n=1}^{\text{NODE}} Q'_n \quad (3.13)$$

where NODE = total number of active (i.e., heat generating) axial nodes in rod N

NROD' = total number of active rods modeled in the VESSEL

Note that NROD' is not necessarily the same as NROD, specified in group 8. A rod specified in group 8 can model any number of actual rods. In general,

$$\text{NROD}' = \sum_{N=1}^{\text{NROD}} \text{RMULT}(N) \quad (3.14)$$

3.3.2 Group 13: VESSEL Component Mesh Cell Boundary Conditions

Two main types of boundary conditions can be specified by input in Group 13. These are 1) inlet or outlet boundary conditions on channels and 2) specific boundary conditions on particular cells within the mesh for both the vertical and transverse control volumes. Inlet or outlet boundary conditions must be specified for channels that do not have vertical connections to other channels in the system at their inlet or outlet. (This, obviously, must include the inlet to channels in Section 1, and the outlet of channels in the last section NSEC. But other channels within the stack of sections can be unconnected at the top or bottom to other channels.) Mesh cell boundary conditions may be applied wherever needed, on any face of any cell within the vessel mesh. But only one boundary condition can be applied to any one node.

The number of channel mesh cell boundary conditions -- including inlet, outlet and internal mesh cells -- in a given problem is specified by the value of NIBND on card VESSEL13.1. The NIBND boundary conditions are set by reading card VESSEL13.4 for each boundary, specifying the channel number (IBOUND(1,M)), axial node number (IBOUND(2,M)), boundary type number (ISPEC(M)), the first boundary value (PVALUE), and the second boundary value (HVALUE), on card VESSEL13.4. The boundary conditions that can be specified for channel mesh cells are listed in Table 3.1 below. The specified boundary values are pressure and enthalpy or flow rate and enthalpy, and either or both of them can be modified by forcing function tables. The forcing functions are identified by table number in parameters NPFN and NHFN for the first and second boundary values, respectively.

TABLE 3.1. Channel Boundary Condition Types

<u>Boundary Type Number</u>	<u>Boundary condition</u>
1	pressure and enthalpy
2	flow rate and enthalpy
3	flow rate only (enthalpy taken from that of the cell the flow rate convects from)
4	mass source and enthalpy
5	pressure sink and enthalpy

If a type 1 boundary condition is specified, the pressure in channel $I = \text{IBOUND}(1,M)$ at node $J = \text{IBOUND}(2,M)$ is set to PVALUE and the enthalpy of any flow into the system across that boundary is set to HVALUE. For a type 2 boundary condition, the flow at the top face of node $\text{IBOUND}(2,M)$ in channel $\text{IBOUND}(1,M)$ is set to PVALUE and the enthalpy to HVALUE. The flow may be either positive or negative. For a type 3 boundary condition, the flow is set to PVALUE and HVALUE is not used. This boundary condition is used to shut off vertical flows within the mesh or to specify flow into or out of the top of a channel, where it is not desirable to specify the node enthalpy, as is done for a type 2 boundary. A type 4 boundary is used to specify a mass source in any cell. PVALUE is the source mass flow rate, and HVALUE is the source enthalpy. A type 5 boundary is used to specify a pressure sink connected to any mesh cell. PVALUE is the sink pressure, and HVALUE the enthalpy of the fluid in the sink.

In addition to the axial boundary conditions, the transverse flow between channels connected by a gap can be shut off at any elevation within the vessel by specifying input for card VESSEL13.7. This input is read if NKBND is greater than zero. NKBND is the total number of transverse mesh cells with crossflow set to zero. The input for card VESSEL13.7 consists simply of the gap identification number K and the contiguous axial levels JSTART to JEND where the crossflow will be set to zero. This format makes it very easy to zero out a sequence of crossflows, from $J = 2$ to $J = 10$, for example. If flow

is to be shut off in more than one segment of a gap, card VESSEL13.7 input can be specified for gap K more than once, until all of the axial levels to be zeroed have been specified.

The general application of the boundary conditions is summarized below:

<u>Boundary Condition</u>	<u>Internal Mesh Cells</u>	<u>Boundary Cells</u>
pressure and enthalpy (type 1)	no	yes
mass flow rate and enthalpy (type 2)	no	yes
mass flow rate (enthalpy unspecified) (type 3)	yes	yes
mass source (flow rate and enthalpy) (type 4)	yes	no
pressure sink (pressure and enthalpy) (type 5)	yes	no
lateral flow set to zero	yes	no

Unless otherwise indicated, the special boundary conditions applied to cells as specified in group 13 are constant in time. The user may, however, specify forcing functions on PVALUE or HVALUE for any or all of the special boundary conditions. The number of forcing functions is specified by the value of NFUNCT on card VESSEL13.1.

The forcing functions on the special boundary conditions are specified in the same way as any other forcing function in the VESSEL input--as tables of forcing function factors versus time. The number of elements in a given table is read on card VESSEL13.2 as NPTS; then NPTS pairs of (ABSCIS(K,I), ORDINT(K,I)) are read on card VESSEL13.3. The ABSCIS array contains the transient time, in seconds, when the corresponding forcing function factor ORDINT is to be applied. ORDINT is defined as the ratio of the value at time

ABSCIS(K,I) of the parameter being forced and its initial value. The factor is determined at any given time step by linear interpolation in the forcing function table. The forcing function tables are numbered sequentially from 1 to NFUNCT in the order they are read in.

If a pressure sink boundary condition (i.e., type 5 boundary condition) has been specified for a cell on card VESSEL13.4, additional information describing the sink geometry must be supplied on card VESSEL13.5. Flow in or out of the sink is defined for a special control volume, as shown in Figure 3.28. The flow area of the control volume is ASINK and the length of the cell is DXSINK. The sink boundary condition usually models a real hole in the system, such as a pipe penetration. In such cases, ASINK is the flow area of the hole. The control volume length, DXSINK, should be on the order of the centroid length of the channel in which the sink lies. The flow in and out of the sink generally encounters some unrecoverable pressure loss due to expansion at the orifice, and the loss coefficient associated with the sink flow is defined by SINKK on card VESSEL13.5. The type 5 boundary conditions are numbered sequentially within the code in the order they are read in on card VESSEL13.4. Card VESSEL13.5 should be read for the sinks in the same sequential order. Card VESSEL13.6 is read for the flow area of mass sources in the same manner.

3.3.3 VESSEL Connections to One-Dimensional Components

The one-dimensional components connect to the VESSEL components through junctions in exactly the same manner that they connect to other one-dimensional components. The connection input identifies the number of connections to the VESSEL component and defines their locations. The number of connections to the VESSEL (NCSR), is read on card COBRA.4. A VESSEL must have at least one connection to the rest of the system and usually has more than one. (A VESSEL modeling a full-size reactor vessel, for example, might have four junctions in the downcomer channels to connect components modeling the cold leg piping, four more junctions in upper plenum channels to connect components modeling the hot leg piping, plus several junction connections in the upper head channels to connect components modeling upper head injection lines.)

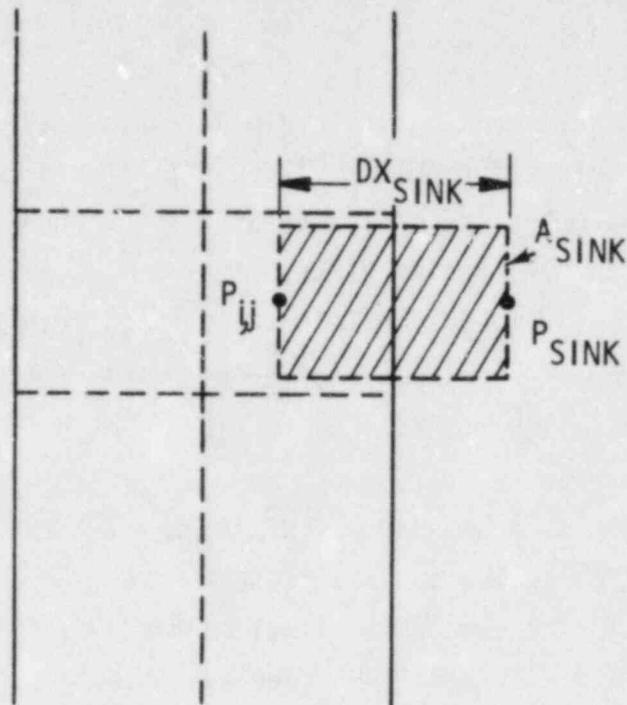


FIGURE 3.28. Control Volume for Pressure Sink Boundary Condition

The locations in the VESSEL component of the NSRC connections are read on card COBRA.5. The one-dimensional component is connected to a specific cell in the VESSEL. This cell is identified by channel number (ISRL(N)) and axial node number (ISRC(N)). One-dimensional components may be connected to the bottom, top or side of the mesh cell, depending on the geometry of the system being modeled. If the flow enters at the top of the cell or from the side, the axial node number ISRC(N) is entered as a positive value, J. If the flow enters at the bottom of the cell, however, ISRC(N) is entered as the negative of the node index number.

For the connections on the side of a cell, the input for ISRF(N) identifies the index number of the gap that faces the one-dimensional component. ISRF(N) is zero if the connection is on a side and faces a solid wall rather than a gap. (An example of this type of connection is a cold leg penetration in the downcomer.) ISRF(N) should be set to negative one (-1) for connections at the top or bottom of a cell. The information provided by the signs of ISRC(N) and ISRF(N) is used to determine the correct direction for the momentum of the fluid entering the VESSEL component through the one-dimensional component.

The VESSEL connects to a one-dimensional component through junction number JUNS(N). This junction number is identified with the end of the one-dimensional component connected to the VESSEL. The components connected directly to the VESSEL are usually PIPES, TEEs, PUMPS or VALVES. BREAKs, FILLS and Steam GENERATORS should not be connected directly to the VESSEL; however, these components can be connected to the other end of a PIPE or TEE that connects to the VESSEL.

3.4 SPECIFICATION OF OUTPUT OF RESULTS FOR VESSEL COMPONENT

The output data generated for the VESSEL component is considerably different from that for the one-dimensional components. It is in general considerably longer and more complex. (See Section 4 for a discussion of the conventional output for the sample problem.) The user has options to limit the output to only those VESSEL elements of special interest or to expand the output to include information not ordinarily specified. The input to specify the form of the output desired for the VESSEL component is supplied in group 14, which is discussed in Section 3.4.1.

Data for graphical presentation of the calculational results can be saved for both the VESSEL and the one-dimensional components. This helps provide a comprehensible and accurate picture of what the calculation is doing. The capabilities of the GRAFIX program that processes the results from the COBRA/TRAC calculation are discussed in the Programmer's Manual (Volume 5). The input instructions for GRAFIX are listed in Section 2.6 of Volume 3. But before this feature can be used the graphics dump tape must be created during the calculation. The input to set up the graphics dump is discussed in Section 3.4.2.

3.4.1 Group 14: Line-Printer Output Options for the VESSEL Component

Group 14 offers the user five separate choices for the format of the VESSEL output. These are specified by the value of N1, read on card VESSEL14.1, as

- 1 - channel data only
- 2 - channel and gap data only
- 3 - rod and unheated conductor data only
- 4 - rod, unheated conductor and channel data only
- 5 - rod, unheated conductor, channel and gap data.

In addition, the user may specify by index number the channels, rods, gaps or unheated conductors the printout should include, if he does not want to see all of them. This is done by setting NOUT1, NOUT2, NOUT3, and NOUT4 on card VESSEL14.1 to the number of channels, rods, gaps and unheated conductors, respectively, to be printed out. For most cases, the user will select N1 = 5, and set NOUT1, NOUT2, NOUT3, and NOUT4 to zero, so that all channels, rods, gaps and unheated conductors will be printed. The option to limit the output is useful mainly for very large problems.

Two additional printout options specified on card VESSEL14.1 are IPROPP and IOPT. If IPROPP is specified as a positive integer, the fluid properties table used in the code to calculate local fluid properties in the VESSEL component will be printed out. If the user does not need this information, he can suppress the printout by setting IPROPP to zero. The variable IOPT lets the user request additional channel output information, which includes the vapor and liquid interfacial drag forces, vapor and entrained droplet interfacial drag forces, and interfacial heat transfer coefficients.

If the user specifies nonzero values for NOUT1, NOUT2, NOUT3 or NOUT4 on card VESSEL14.2, he must identify the index numbers of the specific channels, rods, gaps or unheated conductors he wants to see printed out. The NOUT1 channel numbers are specified in array PRINTC(I) on card VESSEL14.2; the NOUT2 rod numbers are specified in array PRINTR(I) on card VESSEL14.4; the NOUT3 gap numbers are specified in array PRINTG(I) on card VESSEL14.3; and the NOUT4 unheated conductor numbers are specified in array PRINTS(I) on card VESSEL14.5.

3.4.2 Graphics Options

The graphics input options are used to create special data files, one for computer-generated movies and one for input to the graphics program GRAFIX.

It is recommended that the movie option be ignored for the time being. Particle-tracker movies are neither simple nor inexpensive to make, and generating the data is but the first step in a long process. The particle tracker movie software is still under development and will be documented separately at a later date. The graphics option, however, is indispensable for analyzing long and complicated simulations.

The user needs to spend some time thinking about the plots ultimately needed for the simulation. The following questions need to be answered before the simulation is begun:

- For what components should data be saved?
- How often should data be saved for the plots?

Properly used, the graphics options will allow the user to save data for any or all of the components, at any reasonable interval throughout the transient.

The variables saved for a one-dimensional component in the graphics dump include portions of the fixed length table and the variable length table for that component^(a). The geometry arrays and computational variables for the component are also saved, including the flow area, hydraulic diameter, phase densities, mixture velocities, liquid and vapor fractions, phase temperatures and wall temperatures for each cell.

The vessel component, as usual, is treated differently than the one-dimensional components. The user has the option of specifying a normal graphics dump for the vessel, or a very short dump. For each cell of each channel the normal graphics dump for the vessel saves the pressure, liquid fraction, vapor fraction, entrainment fraction, phase enthalpies, phase densities, liquid flow rate, vapor flow rate, heat deposited to the liquid, heat deposited to the vapor, entrainment rate, de-entrainment rate, hydraulic diameter and interfacial heat transfer coefficients, plus several useful variable flags. For every transverse momentum cell (i.e., every cell of every

(a) See Volume 5, the Programmer's Manual, for a description of the contents of these tables for each component.

gap), the normal graphics dump saves the liquid crossflow, vapor crossflow and entrained crossflow. For each axial level of each rod the normal vessel dump saves liquid and vapor temperatures of the fluid seen by the rod, heat transfer coefficients to the liquid and vapor, rod power, rod temperature and local heat transfer mode. This amount of data even for a moderately-sized VESSEL component can result in a data file too large to manage.

If the normal VESSEL graphics dump is too big or is not needed, the short dump option can be used. The user specifies (by channel, rod or gap number and axial level index) precisely which variables to save for the vessel component. This is often the only feasible way to save graphics data for a large VESSEL component. The normal VESSEL dump is flagged by setting IGRFOP to zero on GRAF.2. If the short dump is required, IGRFOP is equal to the number of variables saved.

The number of components for which graphics data may be saved is specified on card GRAF.2 as NCGRAF and the maximum number of graphics dumps to be made is specified in MXGDMP. Both of these values are constrained by dimension parameters. NCGRAF must be equal to MH, and MXGDMP must be no greater than MI; (see Section 2.5 Table 2.1 for a complete definition of MH and MI). It should be noted that MXGDMP cannot be changed in the course of a transient. The initial input value of MXGDMP is used to determine the size of file TAPE11, the graphics dump file, and is used subsequently in calculating locations of dumps for the various time steps. If it were changed in the course of the transient on a restart, the code would not be able to locate the correct place to read or write data in the TAPE11 file.

The short graphics dump for the vessel includes an option to save data for liquid level calculations. (The normal dump automatically saves enough data for such calculations.) This option reduces the amount of data that must be written to file TAPE11 for the calculations by performing part of the calculation in COBRA/TRAC. Instead of saving the area, void fraction and cell length for every cell in every channel included in the liquid level calculation, the code performs the liquid level calculation and saves only the height of each liquid level specified. But to do this, the user must tell the code the channels and axial levels for which the calculation is to be performed.

For a group of channels within a single section, the input to define the liquid level calculation is relatively straightforward. The input, read on card GRAF.3, consists of the number of channels in the section that are to be included in the calculation (NCHLL(N)), the beginning and ending continuity cell numbers, (JSLL(N) and JELL(N)), and the index numbers of the channels involved, (ICLL(I,N)). An example of the geometry for this type of liquid level calculation is shown in Figure 3.29, and the corresponding input for card GRAF.3 is shown below.

NCHLL(N)	JSLL(N)	JELL(N)	ICLL(I,N)
9	3	4	15 16 17 21 22 23 27 28 29

This input results in the calculation of the liquid level over the axial length from node 3 to node 4 in the 9 channels which model the core inlet region above the lower core support plate.

For a liquid level calculation that includes channels in different sections, the input is slightly more complicated. The number of channels in the first section to be included in the calculation is entered in NCHLL(N), as before. JSLL(N) is specified as the beginning axial level of the calculation in the section. For subsequent sections, however, the sign of JSLL(N) is negative indicating that the liquid level calculation is to continue through the section boundary. For the beginning section and any intermediate sections, JELL(N) is the top node in each of the sections. For the last section included in that liquid level calculation, JELL(N) is the ending axial level in the calculation. Within each section the ICLL(I,N) array is used to identify the channels included in the liquid level calculation.

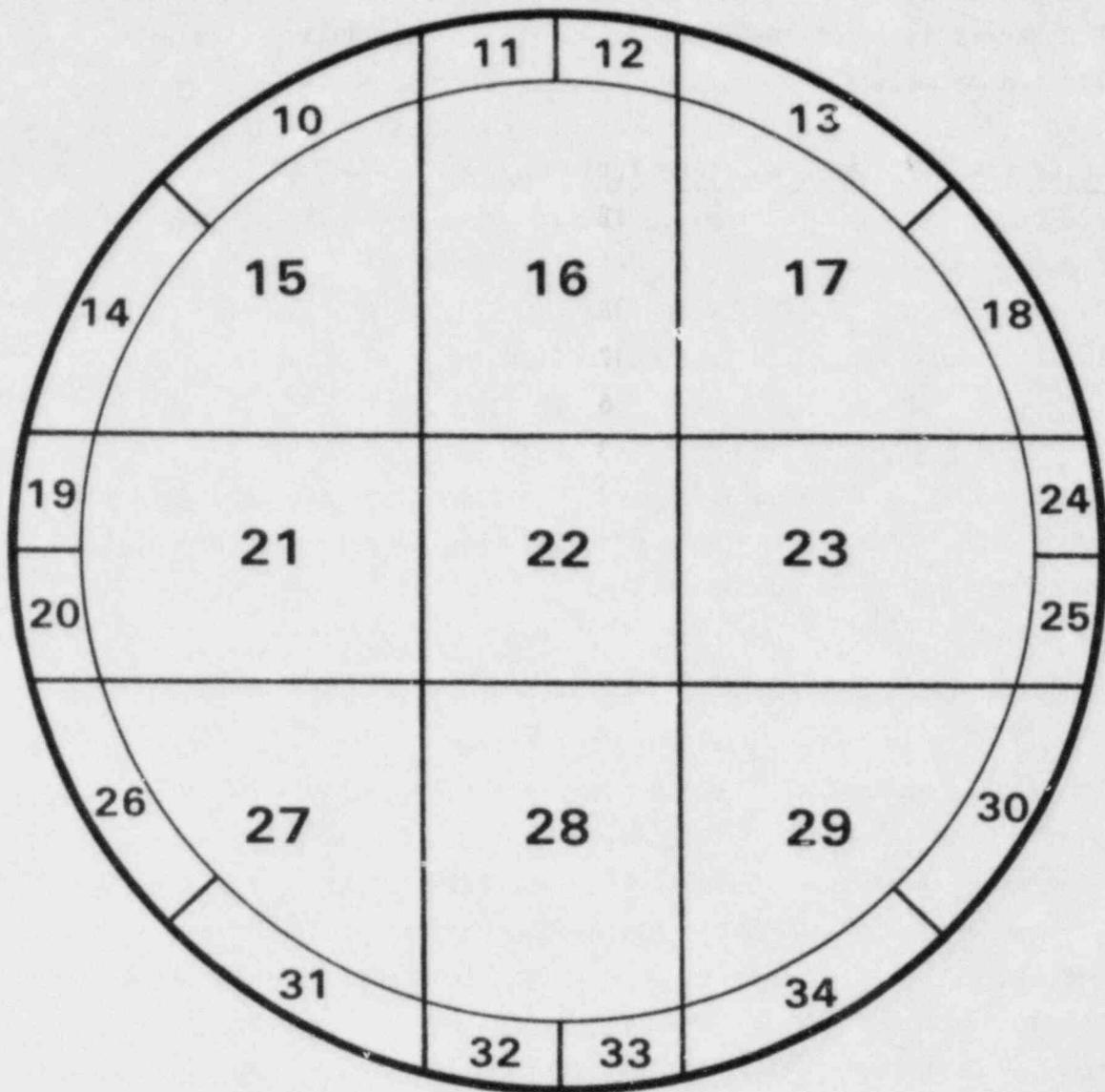


FIGURE 3.29. Core Inlet - Lower Core Support Plate Section
(Section 2 from full PWR Vessel Model)

An example of the geometry for a liquid level calculation that crosses section boundaries is given in Figure 3.30. The corresponding input for card GRAF.3 is given below.

<u>NCHLL(N)</u>	<u>JSLL(N)</u>	<u>JELL(N)</u>	<u>ICLL(I,N)</u>																	
17	2	2	211	212	213	214	215	216	217	218	219									
			220	221	222	223	224	225	226	227										
25	-2	3	237	238	239	240	241	242	243	244	245									
			246	247	248	249	250	251	252	253	254									
			255	256	257	258	259	260	261											
21	-2	3	262	263	264	265	266	267	268	269	270									
			271	272	273	274	275	276	277	278	279									
			280	281	282															

This input yields a liquid level calculation over the vertical height length from node 2 in section 8, through section 9, to node 3 in section 10, in the channels that model the upper head.

The identification numbers of the components for which graphics data is to be saved are entered on card GRAF.4 in array IGRFIT(I). Up to NCGRAF values can be specified, but zeroes may be entered if fewer than NCGRAF components need to be saved. It is possible to change the input specifying which components to dump in the course of a transient, but it must be done cautiously. The IGRFIT array serves to determine what order the components are written to TAPE11, and once a component has been given a particular slot, it can only be deleted -- it cannot be changed. New components must be added at locations in the IGRFIT array that were previously zero.

This convention can best be explained by example. Suppose that a system of two components is set up for a 100 second transient. Initially, the system consists of a pipe and a vessel. Both the pipe (component number 1) and the vessel (component number 2) are of interest, so graphics data is saved for them. The simulation is run from 0 to 25 seconds. The calculation is stopped and restarted with the pipe component replaced by a new

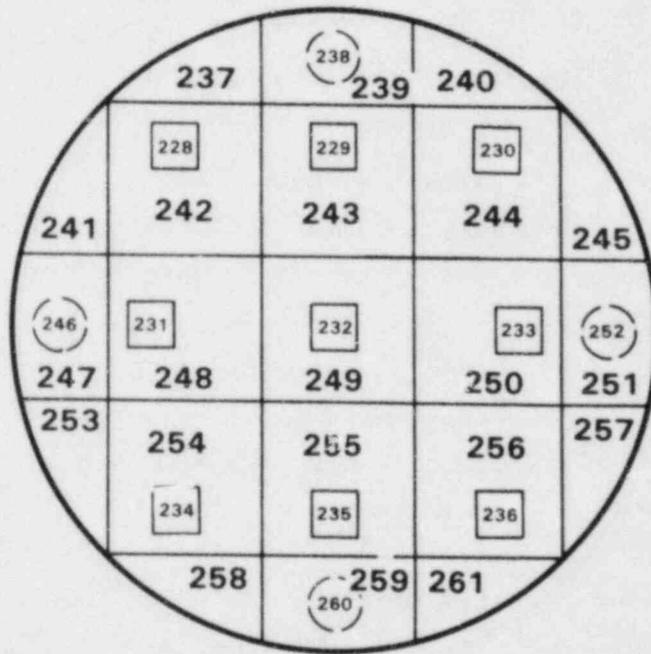
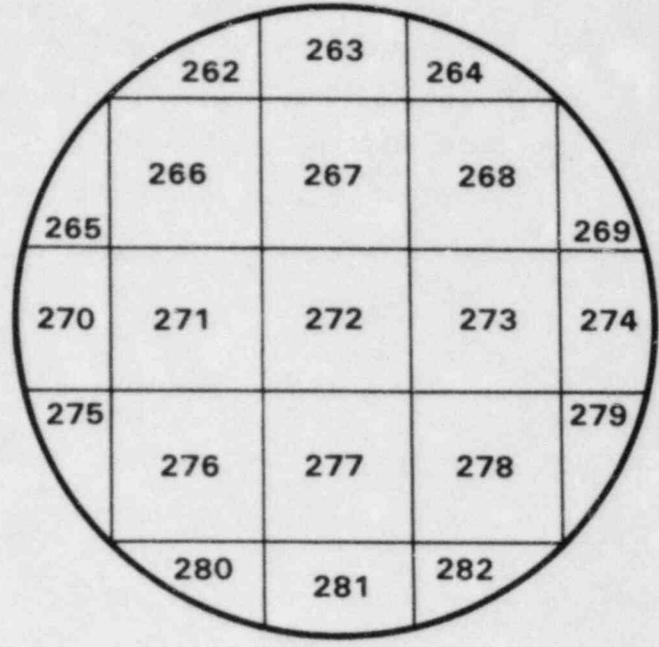
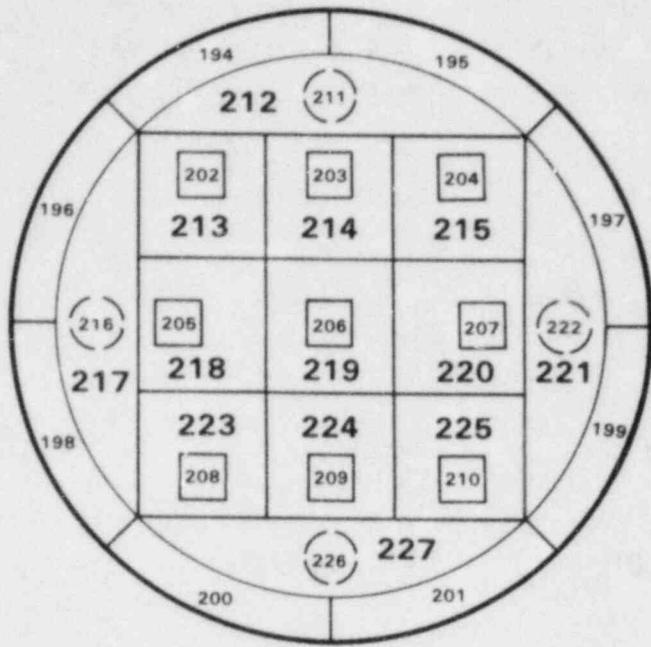


FIGURE 3.30. Upper Head - (Section 8, 9 and 10 from full PWR Vessel Model)

pipe component (number 3) and a break component (number 4). Component #1 is gone, and it is desirable to save the data for the new pipe and break components without losing the data for the original pipe. The simulation is restarted at 25 seconds and run to completion at 100 seconds.

The IGRFIT array, input on card GRAF.4, is used to specify the components to save. The following two sets of cards (GRAF.2, GRAF.4) would be used for the above example:

Set 1: run from 0 to 25 seconds, saving data for components 1 and 2;

GRAF.2

4 100 0 0

(NOTE: NCGRAF=4 implies MH=4.)

GRAF.4

1 2 0 0

Set 2: run from 25 to 100 seconds, saving data for components 2, 3 and 4;

GRAF.2

4 100 0 0

GRAF.4

0 2 3 4

The thing to remember about the IGRFIT array data is that it is position-dependent. Once a component has been assigned to a location in the IGRFIT, that location cannot be changed, other than to zero it.

3.5 ONE-DIMENSIONAL COMPONENTS

The one-dimensional components in COBRA/TRAC are modules developed for TRAC-PD2 to model the reactor primary system. These components provide models for accumulators, pressurizers, pipes, tees, pumps, steam generators and valves. In addition, there are two modules that provide boundary conditions for parts of the system not modeled, consisting of either a pressure sink/source or a flow boundary.

This section is intended primarily to aid the user in understanding how to set up the input data for the one-dimensional components. The input in COBRA/TRAC is unchanged from TRAC-P1A, and the discussions for the various modules have been lifted virtually intact from a draft of the TRAC-P1A Manual. (The solution of the conservation equations, however, has been updated to the capabilities of TRAC-PD2.) Users wishing a more detailed understanding of the solution procedures and models for the one-dimensional components are referred to the TRAC-P1A (Ref. 2) and TRAC-PD2 (Ref. 1) manuals.

In the process of learning to set up the one-dimensional component input data in COBRA/TRAC, a few things were discovered that were not explicitly stated in the TRAC manual, and should be brought to the attention of the new user. These are listed below.

- The end nodes of a component that form a junction with an adjacent component must have the same flow area (FA) and hydraulic diameter (HD) as the adjacent node on the other side of the junction. (This is because there is actually only one node at the junction, even though the input variables must be defined on both components.)
- The additive friction loss coefficient specified for each node of a component in the array FRIC is defined for node N by

$$FRIC_N = \frac{K_D (DH_N)}{(\Delta X_N + \Delta X_{N-1})} \quad (3.14)$$

where K_D = dimensionless loss coefficient for $dP = K_D \rho \frac{V^2}{2g_c}$
 DH_N = hydraulic diameter of cell N adjacent to node N
 ΔX_N = length of cell N adjacent to node N
 ΔX_{N-1} = length of cell N-1 (on the "other side" of node N)

- The solution for the one-dimensional components has built-in form loss coefficient equations for calculating the form drag at sudden area changes. These are flagged by setting the value of NFF at a node N to the negative of the friction correlation flag. For a sudden expansion or orifice of negligible length, the built-in form loss is

$$K_D = 1 - \left(\frac{A_1}{A_2}\right)^2 \quad (\text{where } A_1 > A_2) \quad (3.15)$$

For an abrupt contraction, the built-in form loss is

$$K_D = 0.5 - 0.7 \left(\frac{A_1}{A_2}\right) + 0.2 \left(\frac{A_1}{A_2}\right)^2 \quad (3.16)$$

(If the built-in form loss correlations are flagged by specifying a negative value in the NFF array, the form loss coefficients due to area change at that location should not be specified in the FRIC array. It is possible, however, to model an additional loss at that location that is due to some other impediment to the flow using a FRIC array element.)

3.5.1 Accumulator

An accumulator is a pressure vessel filled with Emergency Core Cooling (ECC) water and pressurized with nitrogen gas. During normal operation each accumulator is isolated from the reactor primary coolant system by a check valve. If the reactor coolant system pressure falls below the accumulator pressure, the check valve opens and ECC water is forced into the primary coolant system.

An accumulator component is simulated by the ACCUM module in COBRA/TRAC. This module can have only one junction connection to other components. This connection is at the highest numbered cell, NCELL. Cell 1 is assumed to be closed off, as shown in the typical accumulator noding diagram in Figure 3.31. It is assumed that the accumulator is not connected to a nitrogen pressure source, so the nitrogen pressure results only from the expansion of the initial gas volume.

The procedures for data input, initialization of arrays, advancement of time-dependent variables, and editing are similar to that given for a PIPE component (see Section 3.5.3). However, in an accumulator the vapor phase properties are those for nitrogen gas. The hydrodynamics are treated using the one-dimensional, semi-implicit drift-flux routine, DF1DS. Drift velocities are not obtained from the SLIP routine as is the case in PIPEs, but are specified in subroutine ACCUM1 to produce a sharp liquid/vapor interface during accumulator discharge. Additionally, pure liquid is discharged from the accumulator until the calculated collapsed water level reaches 90% of the height of the last mesh cell. When this occurs, the accumulator is modified so that the last cell has a zero flow area at its upper surface and the void fraction in the cell is set to 0.1. After this time the discharge properties are not controlled, and vapor-phase properties are those for steam since noncondensable gases are not treated in this version of the code.

The wall friction coefficients in an ACCUM component are set to 0.005, but the additive friction factor, FRIC, at each cell edge can be specified by the user. The walls of an accumulator are assumed to be adiabatic.

The output edit is similar to the PIPE one-dimensional component with the addition of three variables specific to an accumulator: the discharge volumetric flow rate, total liquid volume discharge, and collapsed liquid level.

3.5.2 BREAK and FILL Modules

These modules differ from the other component modules in that they do not model any system component per se, and no hydrodynamic or heat transfer calculations are performed for them. In all other respects, they are treated as any other component, with the same input, initialization and

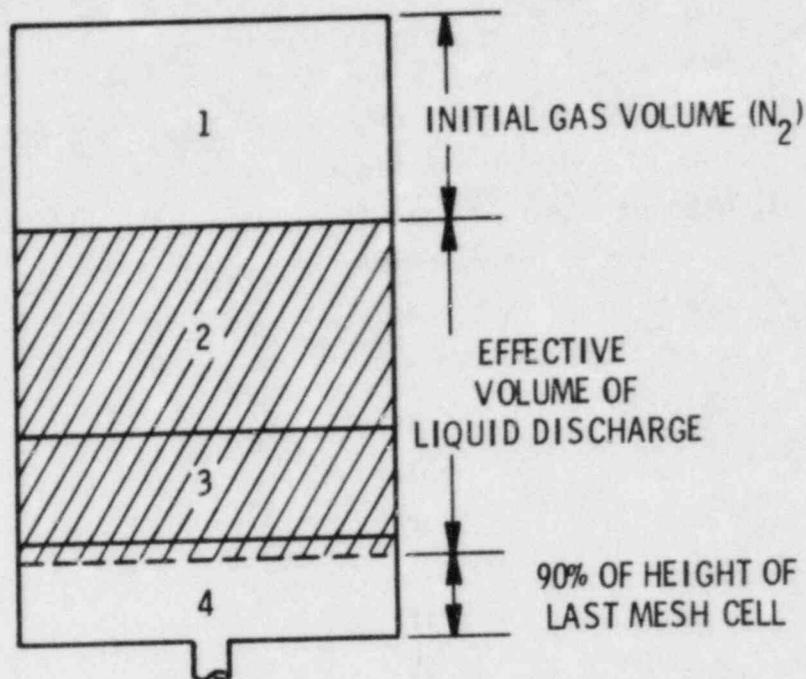


FIGURE 3.31. Accumulator Noding Diagram

identification procedures. The BREAK and FILL modules are used to impose boundary conditions at any one-dimensional component terminal junction.

The BREAK module imposes a pressure boundary condition (which may be time dependent) one cell away from its adjacent component, as shown in Figure 3.32. The void fraction and fluid temperature associated with the BREAK may be specified as a function of time. This module can be used to model any pressure boundary on the system, such as the containment in LOCA calculations or the pressure suppression tank in blowdown experiments.

The FILL module imposes a velocity boundary condition (which may be time dependent) at the junction between the FILL and its adjacent component, as shown in Figure 3.33. FILLS may be used to fix the mass flow of either vapor or liquid at a junction. ECCS injection may be modeled with a FILL. The FILL velocity can be a function of time or pressure, with or without trip control. Five control options are provided for specifying the type of FILL.

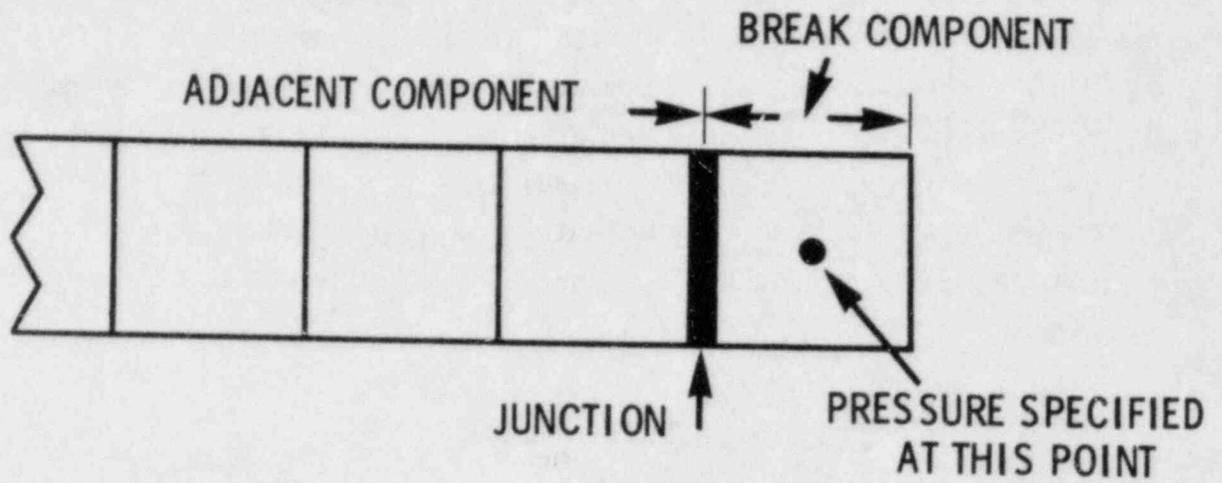


FIGURE 3.32. Break Noding Diagram

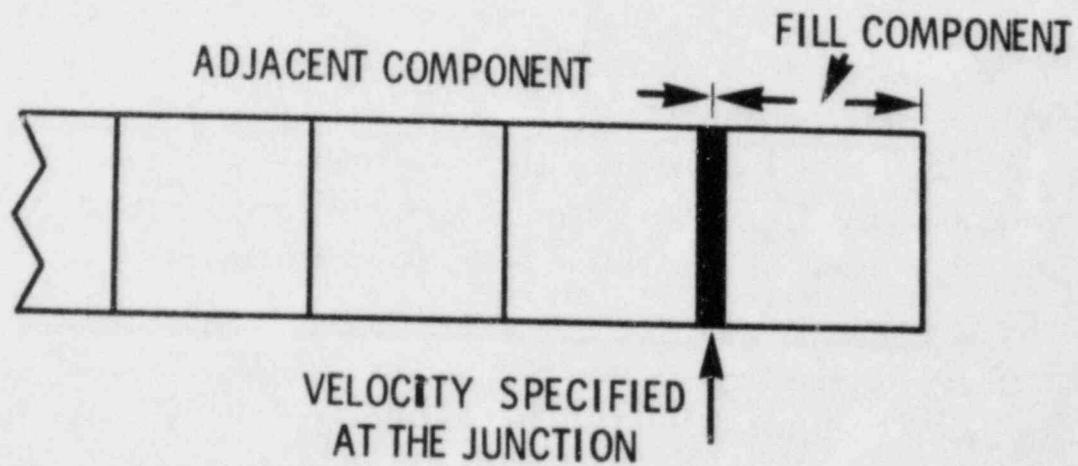


FIGURE 3.33. Fill Noding Diagram

The parameters needed for specifying BREAKs and FILLs are described in the Sections 2.2.2 and 2.2.3 of this manual. It is suggested that the cell volume (VOLIN) and length (DXIN) in FILL and BREAK components be the same as those for the neighboring cell in the adjacent component. The temperature and vapor fraction specified in FILLs and BREAKs are used to determine the properties of fluid convected into the adjacent component if an inflow condition should occur. These components may be connected only to other one-dimensional components. They may not be connected directly to the VESSEL component.

3.5.3 PIPE Module

The PIPE module is used to model thermal-hydraulic flow in a one-dimensional duct or pipe. A PIPE can be used alone in a problem or can connect other components together to model a system. Area changes, wall heat sources and heat transfer across the inner and outer wall surfaces can be modeled in the PIPE component. Material properties for stainless steel (304, 316 and 347), carbon steel A508 and Inconel 718 are available for representing the wall material in the wall conduction calculation.

Figure 3.34 shows a typical noding diagram for a PIPE containing a venturi and an abrupt area change. The numbers within the PIPE indicate cell numbers, and those above it are cell boundary numbers. The geometry is specified by providing a volume and length for each cell and a flow area and hydraulic diameter at each cell boundary. The junction variables JUN1 and JUN2 provide reference numbers for connecting this pipe to other components.

The user must specify the degree of implicitness in the numerical hydrodynamics solution method, the noding for wall heat transfer, and correlations to be used for CHF and wall friction factors in the PIPE input. Most PIPEs should be treated with the faster partially implicit numerical hydrodynamics option (IHYDRO = 0). However, when very high flow velocities are expected, as in a pipe adjacent to a break, the fully implicit hydrodynamics option (IHYDRO = 1) should be used.

The noding in a PIPE with implicit hydrodynamics modeling critical flow were investigated for a number of experiments and the results reported in

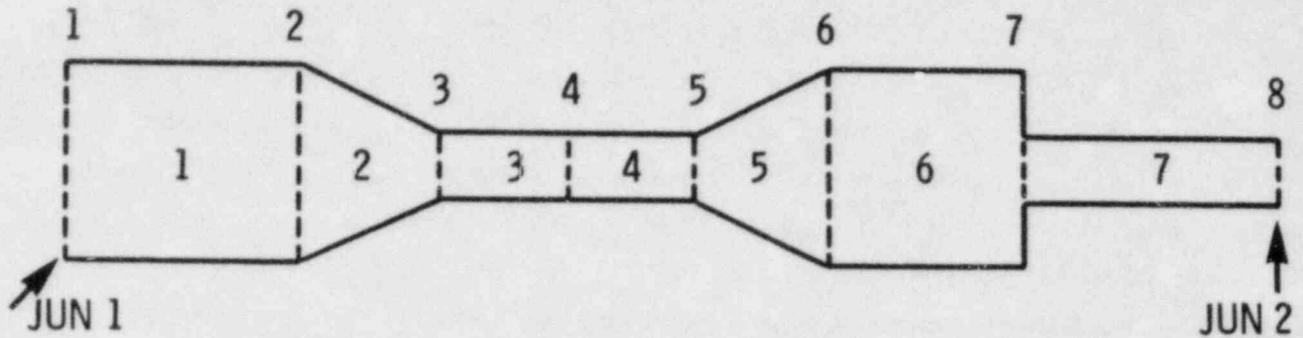


FIGURE 3.34. Pipe Noding Diagram

TRAC-P1A Development Assessment (Ref. 3). The region of a pipe experiencing critical flow must be modeled with extremely short nodes, as shown in Figure 3.35. This figure shows the PIPE noding reported in reference 3 for the Edwards blowdown experiment.

Wall heat transfer can be omitted by setting the number of heat transfer nodes (NODES) to zero. The user can choose one of three methods of calculating CHF, or this calculation can be bypassed by setting the input parameter ICHF to zero. Wall friction and losses due to abrupt area changes are chosen by setting appropriate values of the input arrays NFF and FRIC at each cell interface. The values for the NFF array flag specific friction factor correlations and the values for the FRIC array are included as local additive losses.

While a PIPE can be connected to any other component, including another PIPE, it is usually a good idea to keep the number of components to a minimum. It should also be noted that the junction between two implicit PIPEs

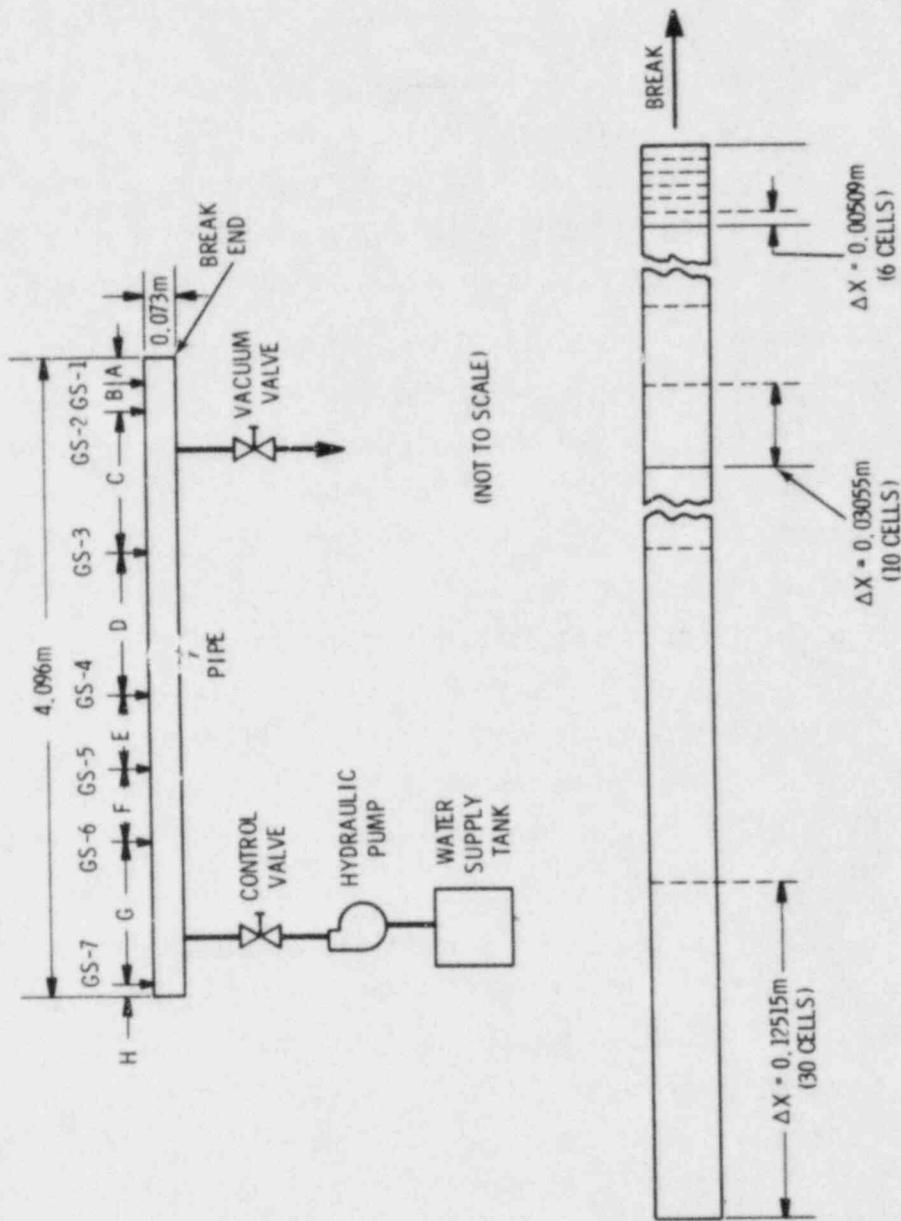


FIGURE 3.35. PIPE Noding Diagram for the Edwards Blowdown Experiment

is treated as partially implicit so that some of the advantages of using the implicit hydrodynamics are lost if a continuous pipe is modeled as two separate PIPES.

When using the semi-implicit hydrodynamics, the lengths of adjacent cells should not be exceedingly different. This can lead to calculational inaccuracies, especially when a long volume follows a short volume. It is recommended that the change in cell lengths should not be greater than a factor of 1.2 between adjacent cells or across a junction boundary.

During the execution of a problem, the solution procedure is controlled by routines PIPE1, PIPE2 and PIPE3. At the beginning of each time step, PIPE1 calls SLIP to obtain relative velocities, FWALL for wall friction and orifice loss coefficients, MPROP for wall metal properties, and HPIPE to obtain wall heat transfer coefficients. During the iterations for a time step, PIPE2 calls DF1D for the numerical hydrodynamics solution and J1D to update boundary arrays. After a time step is successfully completed, PIPE3 updates the wall temperatures with a call to CYLHT, computes new fluid properties (viscosity, heat capacity and surface tension) with a call to FPROP, and resets the boundary arrays with J1D. If the time step fails to converge, THERMO is called to restore variables to their old values and a new time step, half the size of the old one, is tried.

Output for a pipe is managed by subroutines CWPIPE and WPIPE. CWPIPE transfers data from LCM to SCM and calls WPIPE. WPIPE prints out the component number, junction numbers, iteration count, pressures, vapor fractions, saturation temperatures, liquid and vapor temperatures, liquid and vapor densities, mixture velocity, slip ratio and wall friction factor. If wall heat transfer is included (NODES>0), then information on the heat transfer regime, liquid and vapor wall heat transfer coefficients, interfacial heat transfer coefficient, heat transfer rate from the wall, wall temperature for critical heat flux and wall temperature profiles are also printed.

3.5.4 Pressurizer

The pressurizer in a PWR is a large fluid reservoir that maintains the pressure within the reactor primary coolant system and compensates for changes

in the coolant volume due to system transients. During normal operation this reservoir contains the highest energy fluid in the primary system. It is usually maintained about 50% to 60% full of saturated liquid pressurized by the saturated steam above it. The pressurizer pressure is the controlling source of the primary coolant loop pressure and is transmitted by a long surge line connected to one of the hot legs.

A pressurizer component is simulated by the PRIZER module. This module is normally used to model the pressurizer itself with the surge line represented by a PIPE or TEE component. As shown in a typical noding diagram in Figure 3.36, a PRIZER component can be connected only at one junction to other one-dimensional components. By convention the junction connector is at the highest numbered cell NCELL, with cell 1 assumed to be the closed-off end.

The procedures for data input, initialization of arrays, advancement of time-dependent variables, and editing are similar to those given for a PIPE component (see Section 3.5.3). The hydrodynamics are treated using the one-dimensional, semi-implicit drift-flux routine, DF1DS. However, drift velocities are not obtained from the SLIP routine as is the case in PIPE, but are specified in subroutine PRIZR1 to produce a sharp liquid/vapor interface during pressurizer discharge.

In a PRIZER component the walls are adiabatic, but energy transfer from a heater/sprayer can be simulated. The primary purpose of this heater/sprayer logic is not to account for added energy but rather to serve as a system pressure controller. The user specifies a desired pressure setpoint, PSET, and the pressure deviation, DPMAX, at which the heaters input their maximum power of QHEAT. The power that is input to the pressurizer fluid is directly proportional to the difference between PSET and P(1), the pressure in cell 1. That is,

$$Q_{\text{input}} = \text{QHEAT} \left[\frac{\text{PSET} - \text{P}(1)}{\text{DPMAX}} \right] \quad (3.17)$$

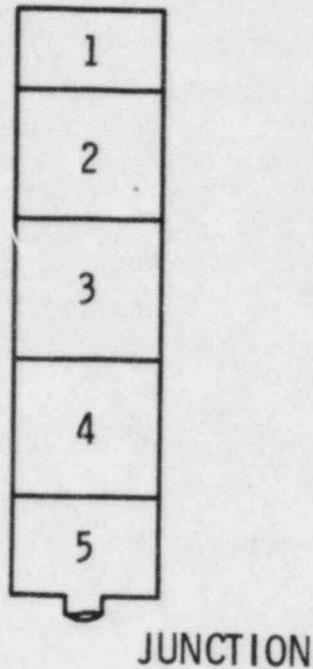


FIGURE 3.36. Pressurizer Noding Diagram

The maximum value of Q_{input} is limited to $\pm QHEAT$. This power is distributed over all fluid cells in the pressurizer with the fraction of power input to each cell equal to the fraction of total liquid mass in that cell. Power is not added if the collapsed liquid level is less than the input parameter ZHTR. If pressure control is not desired, QHEAT is set to zero.

Wall friction coefficients are calculated in routine FWALL by specifying a friction correlation option, NFF, along with additive friction factors, FRIC, for each cell edge. The homogeneous flow friction factor option (NFF = 1) is suggested for a pressurizer.

The output edit for a PRIZER component is similar to the PIPE one-dimensional component with the addition of four variables specific to the pressurizer: the discharge volumetric flow rate, total liquid volume discharged, collapsed liquid level and heater/sprayer power input to the pressurizer fluid.

3.5.5 PUMP Module

The pump module employed in COBRA/TRAC describes the interaction of the system fluid with a centrifugal pump. The model calculates the pressure differential across the pump and its angular velocity as a function of the fluid flow rate and the fluid properties. The model is designed to treat any centrifugal pump and can include two-phase effects.

The pump model is represented by a one-dimensional component with N cells, N must be greater than 1. A typical noding diagram for the pump component is shown in Figure 3.37. The pump momentum is modeled as a source (SMOM) that is included between cells 1 and 2. The source is positive for normal operation with the pressure rise occurring in going from cell 1 to cell 2, so it is necessary to number cells so that the cell number increases in the normal flow direction.

The following considerations were felt to be important in creating the pump module:

- 1) Compatibility with adjacent components should be maximized.
- 2) Choking at the pump inlet or outlet should be automatically predicted.
- 3) The calculated pressure rise across the pump should agree with that measured at steady-state conditions.

The first two criteria eliminated the use of a lumped-parameter model. Since the adjacent components are usually described by PIPE modules that are based on a one-dimensional drift-flux approximation, the pump is treated likewise. The resulting PUMP module combines the PIPE module with pump correlations. The pump model is identical to the one-dimensional pipe model except that a momentum source is included in the mixture momentum equation written between cells 1 and 2:

$$\frac{V_{1.5}^{n+1} - V_{1.5}^n}{\Delta t} = \frac{(p_1^{n+1} - p_2^{n+1})}{\rho_{1.5}^n \Delta X} - C^n - g_z - \frac{f V_{1.5}^n |V_{1.5}^n|}{D_h} + \text{SMOM} \quad (3.18)$$

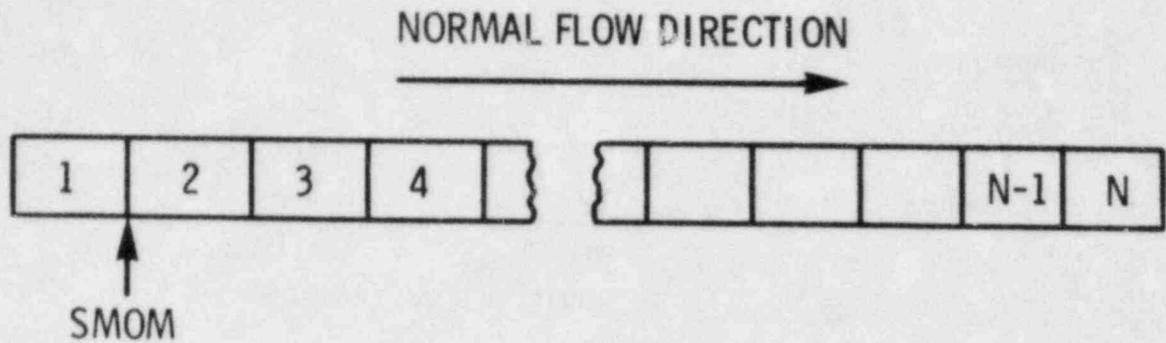


FIGURE 3.37. Pump Noding Diagram

where f is the friction factor, C^n represents the convective terms evaluated at time n and 1.5 refers to the average value between cell 1 and cell 2. The source term, SMOM is taken to be

$$SMOM = \frac{\Delta P}{\rho_{1.5}^n \Delta X} + C^n + g_z + \frac{f}{D_h} v_{1.5}^n |v_{1.5}^n| \quad (3.19)$$

where ΔP is the pressure rise through the pump, evaluated from the pump correlation at the flow velocity $v_{1.5}^n$ and density $\rho_{1.5}^n$. With this definition of the momentum source, the steady-state solution of Equation (3.18) is

$$P_2 - P_1 = \Delta P \quad (3.20)$$

which is the desired result.

It is necessary to evaluate the momentum source for a pump cell only once each time step, and the source is needed only during the explicit pass in DF1DI or DF1DS, which calculates the temporary velocity, V . Numerical results indicate that evaluating the pump source at the old time (n) is adequate and that it is not necessary to re-evaluate the source during the implicit iteration phase of DF1D. The pump source evaluation is performed by subroutine PUMPSR.

The pump characteristic curves describe the pump head and torque response as a function of fluid volumetric flow rate and pump speed. Homologous curves (one curve segment represents a family of curves) are used for this description because of their simplicity. These curves describe in a compact manner all operating states of the pump obtained by positive or negative impeller velocities with positive or negative flow rates.

To take into account two-phase effects on pump performance, the pump curves are divided into two separate regimes. Data indicate that two-phase pump performance in the vapor fraction range of 20 to 80% is significantly degraded in comparison to its performance at vapor fractions outside of this range. One set of curves describes the pump performance for single-phase fluid (0 or 100% vapor fraction), and another set describes it for two-phase fluid. The pump head at any vapor fraction is calculated from the relationship

$$H = H_1 - M(\alpha) (H_1 - H_2) \quad (3.21)$$

where H = total pump head

H_1 = pump head from single-phase homologous curves

H_2 = pump head from the fully degraded homologous curves

M = pump head degradation multiplier

α = vapor fraction

The two-phase hydraulic torque is treated similarly. The following definitions are employed in the subsequent development:

ΔP = pump differential

ρ = pump inlet density

H = pump head = $\frac{\Delta P}{\rho}$

Q = pump volumetric flow rate

Ω = pump impeller angular velocity

In order to allow one set of curves to be used for a variety of pumps, the following normalized quantities are used:

$$\begin{aligned}h &= \frac{H}{H_R} \\q &= \frac{Q}{Q_R} \\ \omega &= \frac{\Omega}{\Omega_R}\end{aligned}$$

where the subscript R denotes the rated condition. Use of the pump similarity relations (Ref. 4) shows that

$$\frac{h}{\omega^2} = f\left(\frac{q}{\omega}\right) \quad (3.22)$$

For a small ω , this correlation is not satisfactory, and the following combination is used:

$$\frac{h}{q^2} = f\left(\frac{\omega}{q}\right) \quad (3.23)$$

The first correlation is used in the range $0 < \frac{q}{\omega} < 1$, and the second is used in the range $0 < \frac{\omega}{q} < 1$. The four resulting curve segments, as well as the curve selection logic used, are shown in Table 3.2.

The dimensionless torque, β , is defined by

$$\beta = \frac{T_{hy}}{T_R}$$

where T_{hy} = hydraulic torque

T_R = rated torque.

TABLE 3.2. Definition of the Four Curve Segments Used to Describe Homologous Pump Curves

<u>Curve Segment</u>	<u>$\frac{q}{\omega}$</u>	<u>ω</u>	<u>q</u>	<u>Correlation</u>
1	$\left\{ \begin{array}{l} \leq 1 \\ \leq 1 \end{array} \right.$	> 0	$\left. \right\}$	$\frac{h}{\omega^2} = f\left(\frac{q}{\omega}\right)$
4		< 0		
3	$\left\{ \begin{array}{l} > 1 \\ > 1 \end{array} \right.$	$\left. \right\}$	< 0	$\frac{h}{q^2} = f\left(\frac{\omega}{q}\right)$
2			≥ 0	

q = normalized flow

ω = normalized angular velocity

h = normalized head

The single-phase torque, T , is dependent upon the fluid density and is calculated from

$$T = \beta T_R \frac{\rho}{\rho_R} \tag{3.24}$$

where ρ is the pump inlet density and ρ_R is its rated density. The density ratio multiplier is needed to correct for the density difference between the pumped fluid and the rated condition. For two-phase conditions the impeller torque is calculated from

$$T = T_1 - N(\alpha)(T_1 - T_2) \tag{3.25}$$

where T = total impeller torque

T_1 = impeller torque from single-phase homologous curves

T_2 = impeller torque from fully degraded homologous curves

$N(\alpha)$ = torque degradation multiplier

The homologous, normalized torque curve segments are correlated in the same manner as the head curve segments shown in Table 3.2.

In addition to the homologous head and torque curves, the head degradation multiplier and torque degradation multiplier defined in Equations (3.21) and (3.25) are needed. These functions are usually nonzero only in the vapor-fraction range where the pump head and torque are either partially or fully degraded.

The pump module treats the pump angular velocity as a constant (input) while its motor is energized. After a drive motor trip, the time rate of change of the pump motor assembly is proportional to the sum of the moments acting on it and is calculated from the equation

$$I \frac{d\Omega}{dt} = \sum_i T_i = (T + T_f + T_b) \quad (3.26)$$

where I = pump motor assembly moment of inertia

T = impeller torque

T_f = torque due to friction (constant)

T_b = bearing and windage torque

T_b is assumed to be of the form

$$T_b = C \frac{\Omega |\Omega|}{\Omega_R} \quad (3.27)$$

where C is an input constant and Ω_R is the rated impeller angular velocity. The impeller torque is evaluated from the homologous torque curves and

Equation (3.25) as a function of the fluid density and flow rate as well as pump angular velocity. For time step $n+1$, Equation 3.26 is evaluated explicitly

$$\Omega^{n+1} = \Omega^n + \frac{1}{I} \sum_i T_i (\Omega, \rho, Q)^n \Delta t^{n+1} \quad (3.28)$$

Pump Options

The wall heat transfer, wall friction, CHF calculation and implicit hydrodynamics options are the same for the PUMP module as for the PIPE module. In addition the following options are specified: pump type, motor action, reverse speed option, two-phase option and pump curve option.

If the pump motor is energized, its angular velocity is assumed to be the constant value specified. If the motor is not energized, a pump coastdown calculation is performed using the specified initial pump speed.

There are two pump options available. For pump option 1 (IPMPTY = 1) the pump speed variation is specified by input. The pump is initially energized at a constant speed specified by input (OMEGA). The pump motor may be tripped by a TRIP signal. If a pump trip has occurred, the pump speed is taken from a table of pump speed versus time-after-trip (array SPTBL).

Pump option 2 (IPMPTY = 2) is similar to option 1 except that the pump speed is calculated from Equation (3.28) after a trip has occurred rather than from an input table. The relationship between the various pump input parameters as well as the algorithm for the pump speed calculation are shown in Table 3.3. The value entered for IPMPTR is the TRIP identification number for pump trip initiation and NPMPTX is the number of pairs of points in the pump speed table (SPTBL). If IPMPTR = 0, the pump will maintain a constant speed.

If the reverse speed option is specified (IRP = 1), the pump can rotate in both the forward and reverse directions. If reverse speed is not allowed

(IRP = 0), the pump will rotate in the forward direction only. For this case, if negative rotation is calculated (after trip with option 2) the speed will be set to zero.

If the two-phase option is turned on (IPM = 1), the degraded pump head and torque will be calculated from Equation (3.21) and (3.25). If the two-phase option is turned off (IPM = 0), only the single-phase head and torque homologous curves will be used.

The user may either specify pump homologous curves in the input or use the built-in pump curves. The built-in pump curves are for the MOD-1 Semiscale system pump and are based on the data in Refs. 4 through 6. These curves, as well as the head and torque degradation multipliers, are shown in Figures 3.38 through 3.43. Since these homologous curves are dimensionless, they can be used to describe a variety of pumps by specifying as input the rated values for density, head, torque, flow and angular velocity.

There are several restrictions and limitations in the current version of the pump module. Since there is no pump motor torque-versus-speed model, the pump speed is assumed at the input value if the motor is energized. The pump momentum source must be located between cells 1 and 2 of the pump model. Finally, the head degradation multiplier, $M(\alpha)$, and the torque degradation multiplier, $N(\alpha)$, are assumed to apply to all operating states of the pump.

TABLE 3.3. Pump Control Input Parameters

<u>IPMPTY</u> Pump Option	<u>IPMPTR</u> Pump Trip I.D.	<u>NPMPTX</u> <u>SPTBL</u> Pairs of Points	<u>Pump Speed</u> <u>Speed Table</u>	<u>Algorithm</u>
1	x = pump trip desired	x	x	OMEGA before trip
	0 = no pump trip	0		SPTBL after trip
2	x = pump trip desired	x		OMEGA before trip
	0 = no pump trip	0		Code calculated after trip

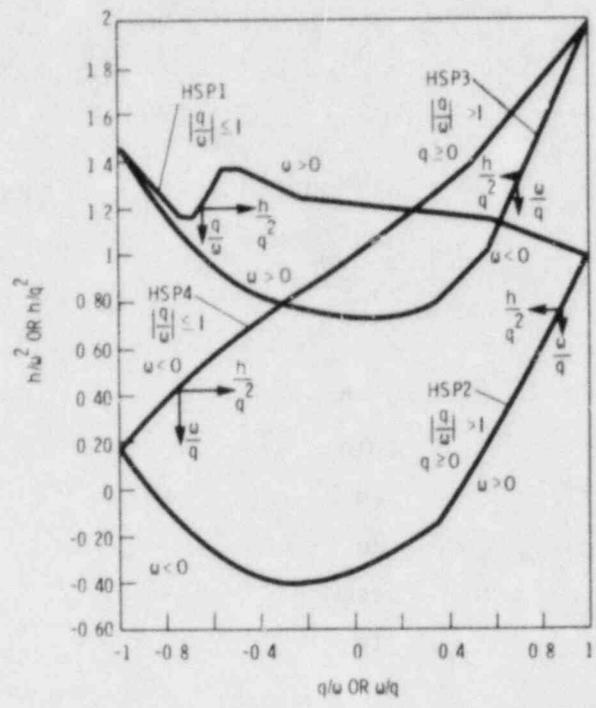


FIGURE 3.38. Single-Phase Homologous Head Curves

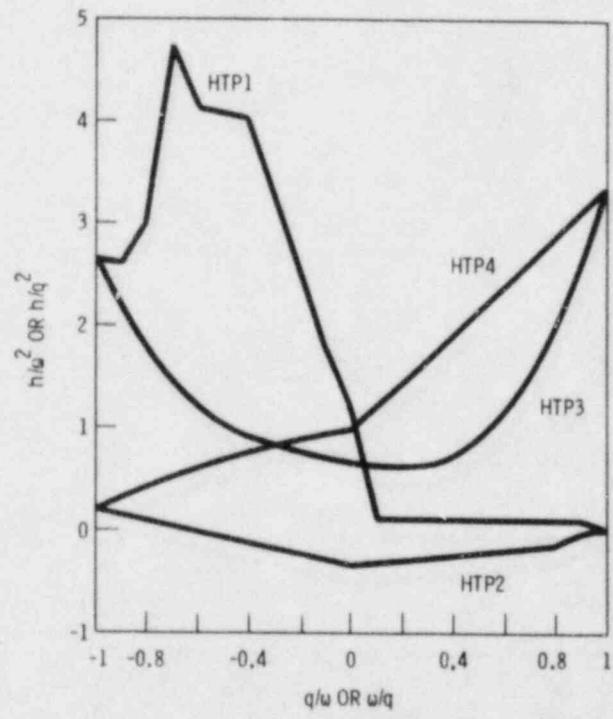


FIGURE 3.39. Fully Degraded Homologous Head Curves

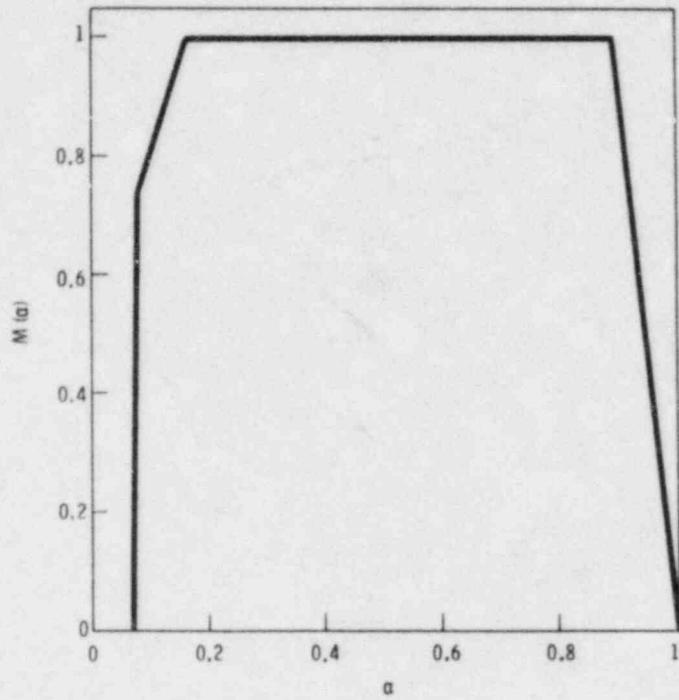


FIGURE 3.40. Head Degradation Multiplier

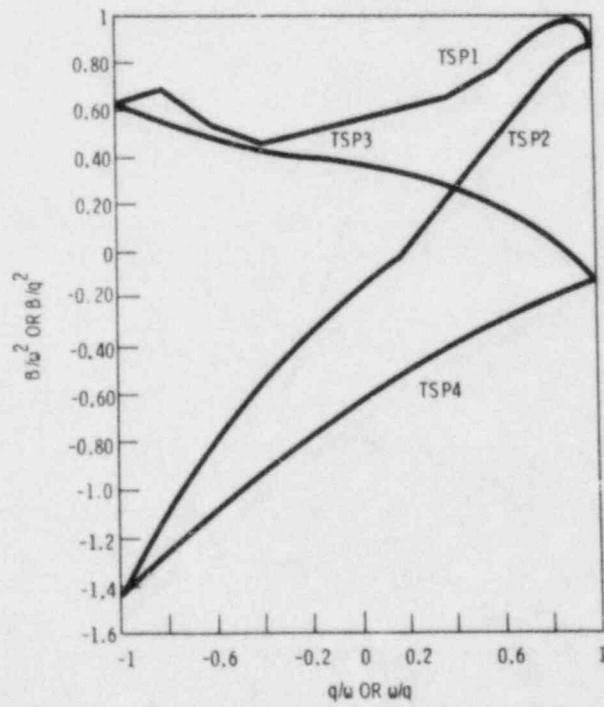


FIGURE 3.41. Single-Phase Homologous Torque Curves

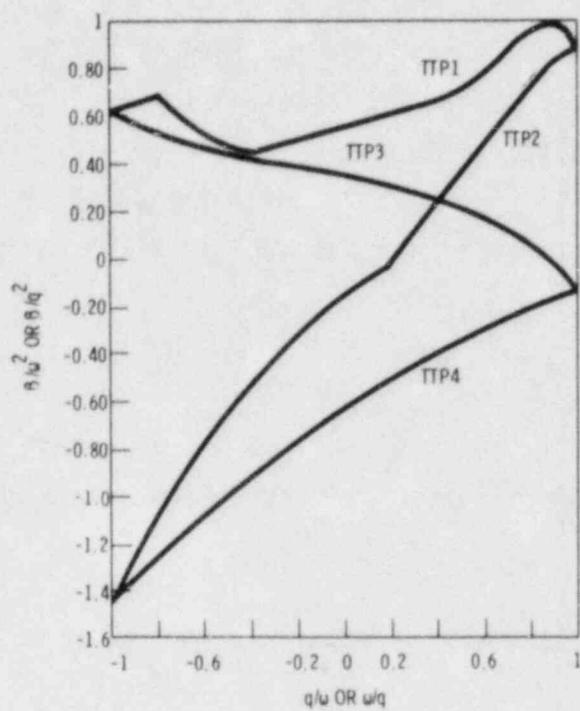


FIGURE 3.42. Fully Degraded Homologous Torque Curves

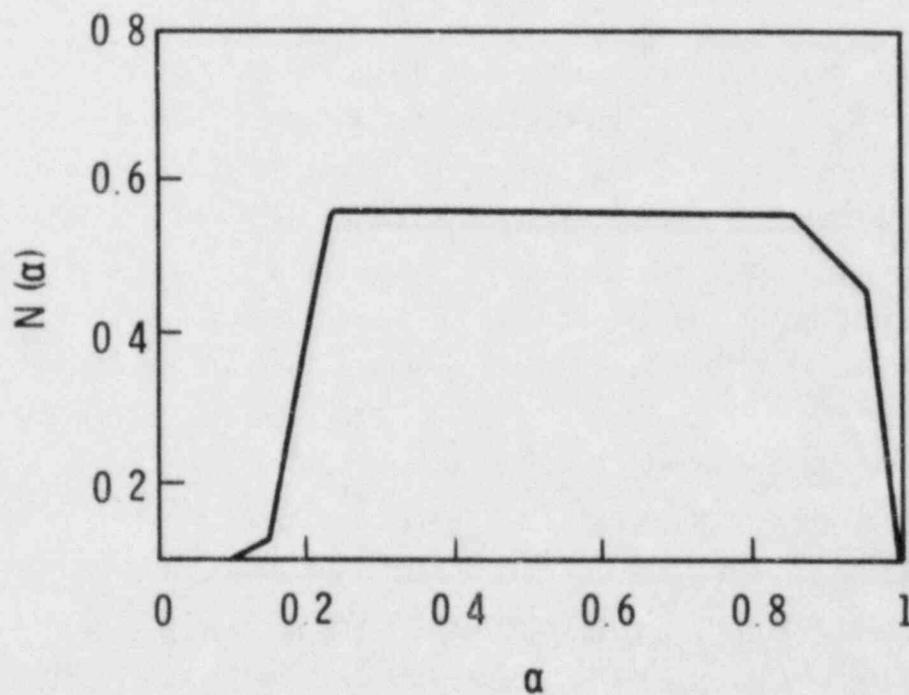


FIGURE 3.43. Torque Degradation Multiplier

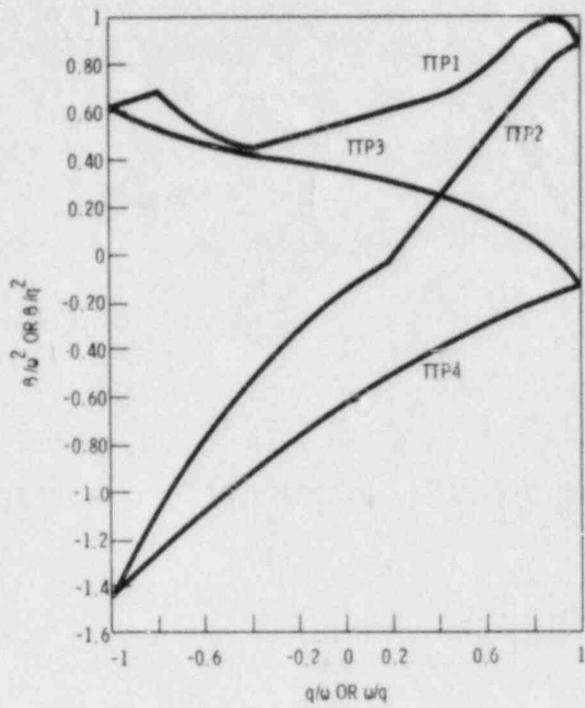


FIGURE 3.42. Fully Degraded Homologous Torque Curves

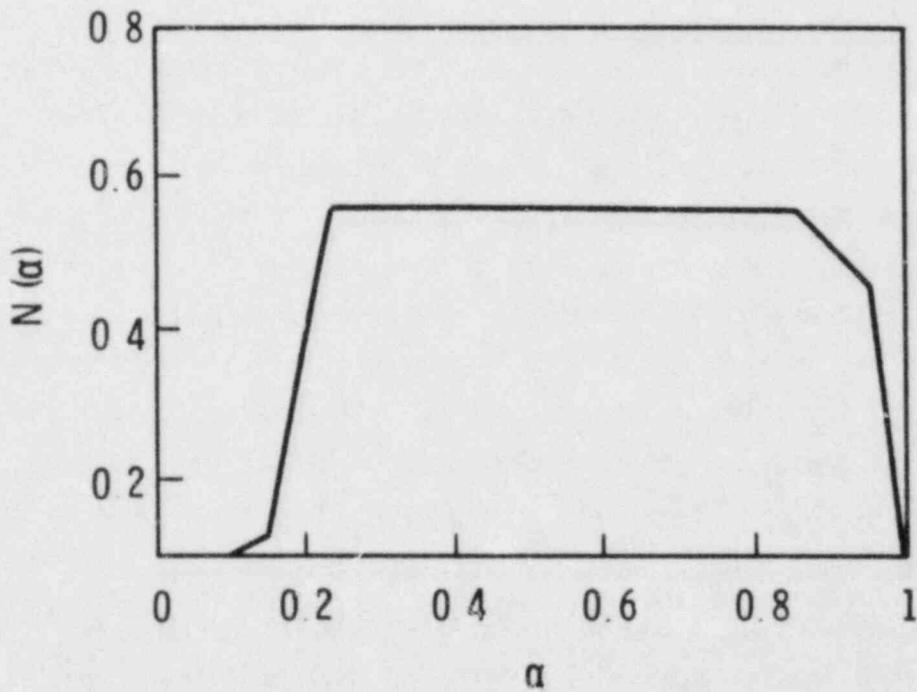


FIGURE 3.43. Torque Degradation Multiplier

The PUMP module input consists of the same geometric and hydrodynamic data and initial conditions that are required for the PIPE module. In addition, information specific to the PUMP is required, as described in the input specifications. (See Section 2.2.6.) The speed table (SPTBL) as well as the homologous pump curve arrays must be input in the following order:

$$x(1),y(1),x(2),y(2), \dots x(n),y(n).$$

Here x is the independent variable and y is the dependent variable. Furthermore the independent variables must be input in a monotonic increasing order, i.e.,

$$x(n) > x(n-1) > \dots x(2) > x(1).$$

Linear interpolation is used within the arrays.

3.5.6 Steam Generator

In a pressurized water reactor the steam generators transfer energy from the primary coolant loop to the secondary coolant to produce steam. The STGEN module can be used to model either a U-tube or a once-through type steam generator. The user specifies the type of generator through the input variable KIND: 1 = U-tube; 2 = once-through. Although there are two different steam generator designs, the basic operation is similar for both types. Primary coolant enters an inlet plenum, flows through a tube bank to exchange heat with a secondary coolant on the exterior of the tube bank, and is discharged into an outlet plenum. A typical noding diagram for a STGEN component is given in Figure 3.44. This figure illustrates the modeling conventions used for this component. On the primary side, cell 1 and cell NCELL1 are adiabatic and model the inlet and outlet plenums, respectively, of the steam generator. The tube bank is represented by a single tube with flow area, wall area and fluid volume of the entire tube bank.

In the STGEN module, the primary side and the secondary side hydrodynamics are treated separately, with the wall heat transfer the only coupling between the two sides. The hydrodynamics of the primary side are

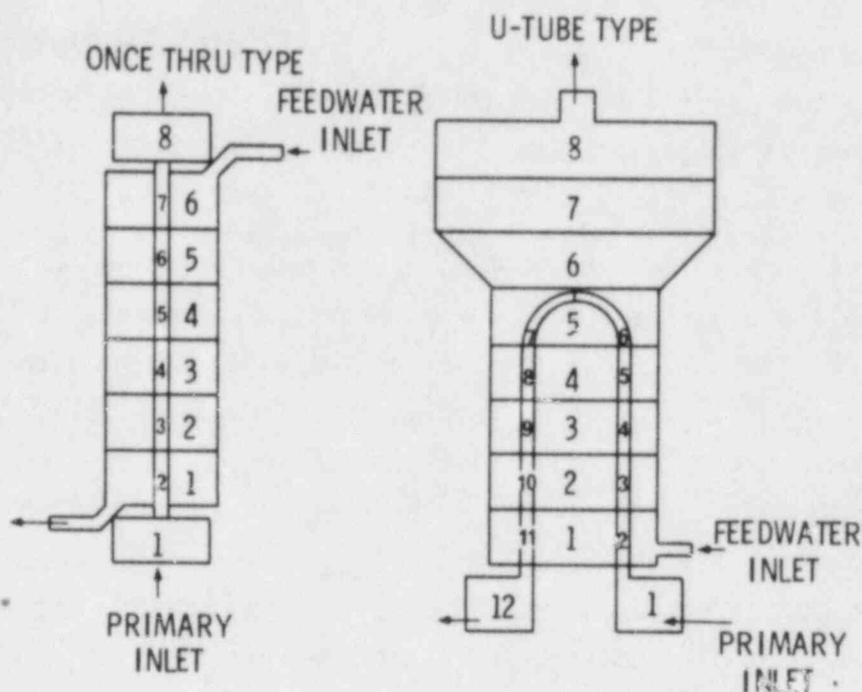


FIGURE 3.44. Noding Diagrams for Once-Through and U-Tube Steam Generators

solved by calling the one-dimensional drift-flux routine, DF1DI or DF1DS. The tube wall temperature for each mesh cell is held constant over the time step, but the vapor and liquid temperatures are treated implicitly. Next, the hydrodynamics for the secondary side are solved in the same manner. Once convergence has been reached for all system components, a final pass is made and tube wall temperatures are updated for the current time step.

The procedures for reading input, initialization of arrays, advancement of the time step, and editing are similar to those described for the PIPE module (see Section 3.5.3). The calculational sequence for the STGEN module is identical to that for a PIPE except that the sequence of calls is performed twice -- once for the primary side and once for the secondary side.

Although the procedure for reading input data is similar to a PIPE module, there are some differences. The most obvious difference is the specification of four junction numbers (see Figure 3.44): two for the primary side connections and two for the secondary side. Although it is possible to connect the secondary side junctions to any one-dimensional component, the

most common arrangement is to model the inlet as a FILL and the discharge as a BREAK. The FILL specifies the secondary side fluid inlet conditions and flow rate at the base of the tube sheet. The BREAK specifies the steam generator secondary steam dome pressure.

There is no provision for modeling the downcomer in the steam generator secondary with the one-dimensional steam generator component, so the fluid state for the boundary FILL must be that of the water entering the tube bank and not simply the feedwater inlet conditions. In some calculations, the inlet conditions at the lower tube sheet on the secondary side can be readily deduced. But in many cases, particularly transients where the behavior of the secondary system significantly influences the results, this model is not adequate. The VESSEL component can be used in such cases to model the steam generator in much greater detail, including the downcomer and carryover from the steam dome.

The number of fluid mesh cells on the primary side is specified as NCELL1, and that on the secondary side is NCELL2. There are some constraints imposed on the possible combinations of values for NCELL1 and NCELL2. For a once-through type (KIND = 2), $NCELL2 = NCELL1 - 2$ (i.e., on the primary side, cell 1 is the inlet plenum, cells 2 through $NCELL1 - 1$ model the tube bank, and cell NCELL1 is the outlet plenum.) Each secondary cell contains (and exchanges heat with) one primary cell. For a U-tube type (KIND=1), it is assumed that two active primary cells, one on the upflow side of the U-tube and the other on the downflow side, are contained in one active secondary cell (see Figure 3.44). Thus, NCELL2 must be greater than or equal to $(NCELL1 - 2)/2$. The secondary side cells that are greater than $(NCELL1 - 2)/2$ are treated as adiabatic and can be included to model the volume above the tube bank. (In Figure 3.44 these are cells 6 through 8 on the secondary side.)

There must be at least one wall temperature node (NODES) in the primary tube of a steam generator. Three nodes are suggested for most cases since this places a node at each tube surface and one at the tube wall center. The tube material is specified with the variable MAT. Available material options are stainless steel 304, 316 and 317, carbon steel A508 and Inconel 718. Two flags, ICHF1 and ICHF2, are used to determine if a CHF calculation is to be

performed on the primary and secondary sides, respectively. If CHF calculations are desired, these flags are set equal to 1; otherwise they are set equal to 0.

A CHF calculation must be flagged on the secondary side, or boiling heat transfer calculations will not be performed. It is therefore recommended that the combination $ICHF1 = 0$ and $ICHF2 = 1$ be used, since boiling occurs on the secondary side. If in a particular problem boiling and/or dryout is expected on the primary side, however, $ICHF1$ should be specified as 1 also. Either fully-implicit or semi-implicit hydrodynamics may be selected for the STGEN component.

There are some special considerations in setting up the geometric input data in order to model the entire tube bundle of a steam generator with a single tube. The inner tube radius, $RADIN$, tube wall thickness, TH , and hydraulic diameter, HD , should be those of an actual single tube in the bundle, but the flow area and cell volumes must correspond to the values for the total tube bank. The user must also specify the heat transfer surface area in each cell for both the primary and secondary sides. This is the effective heat transfer surface area for all the tubes in each mesh cell and can be used to account for tube fouling factors or enhanced heat transfer from fins. Steam generator cells can be made adiabatic by specifying zero heat transfer areas. (The inlet and outlet plenums, i.e., the first and last cells on the primary side, should be adiabatic.)

Specifying the heat transfer area for the "once-through" type is straightforward; it is the total effective heat transfer area for the steam generator, multiplied by the fraction of total tube length in each mesh cell. For the "U-tube" type, however, the convention is a little obscure. On the primary side the heat transfer area is the effective interior area for all the tubes in each mesh cell. The total heat transfer to a secondary side fluid cell is the sum of the heat transfer from the up-tubes and the down-tubes in the cell, but the code assumes that the wall heat transfer area specified in the secondary cells includes only that of the up-tubes. In the calculation, the effective heat transfer areas for the up-tubes and down-tubes are assumed to be equal, and the input value for wall surface area on the

secondary side is doubled. Therefore, the user should not specify WA2 as the effective surface area of both the up and the down tubes in a secondary side fluid cell. The value entered for WA2 should be the area of only the up-tubes in a secondary side fluid cell.

The volumes and flow areas on the primary side are those determined by considering all the tubes in the bank. However, the hydraulic diameter is that for a single tube. The volumes and flow areas on the secondary side are the actual geometric values for each mesh cell. The secondary hydraulic diameter is determined by standard methods used in heat transfer over tube bundles.

Tube wall initial temperatures must also be specified for each node in each cell; an array with NCELL1*NODES elements. Even though cell 1 and cell NCELL1 are adiabatic, tube wall temperatures must be given for both of these cells. (They are defined only to simplify indexing.) In the numbering convention used, temperatures begin with cell 1 and are specified from interior (primary side) to exterior (secondary side) for each mesh cell. (That is, the temperatures are specified in the order $T_{1,1}$, $T_{1,2}$, $T_{1,3}$, ... $T_{1, \text{NODES}}$, $T_{2,1}$, $T_{2,2}$, $T_{2,3}$, ..., $T_{2, \text{NODES}}$, etc. The first index is the cell number, the second is the node number.)

Friction factor correlation options (NFF) and additive friction losses (FRIC) are given separately for the two sides. The possible options for NFF are described with the STGEN component input instructions in Section 2.2.7. The homogeneous option (NFF = 1) is suggested for both the primary and secondary sides.

The output edit for a steam generator component is similar to that given for a PIPE component, with primary side variables given first and then secondary side variables. Also, heat transfer variables are always given. Tube wall temperatures are printed for each node of each active mesh cell.

3.5.7 TEE Module

The TEE module models the thermalhydraulics of three piping branches, two of which lie along a common line with the third entering at some angle β from the main axis of the other two (see Figure 3.45). From the standpoint

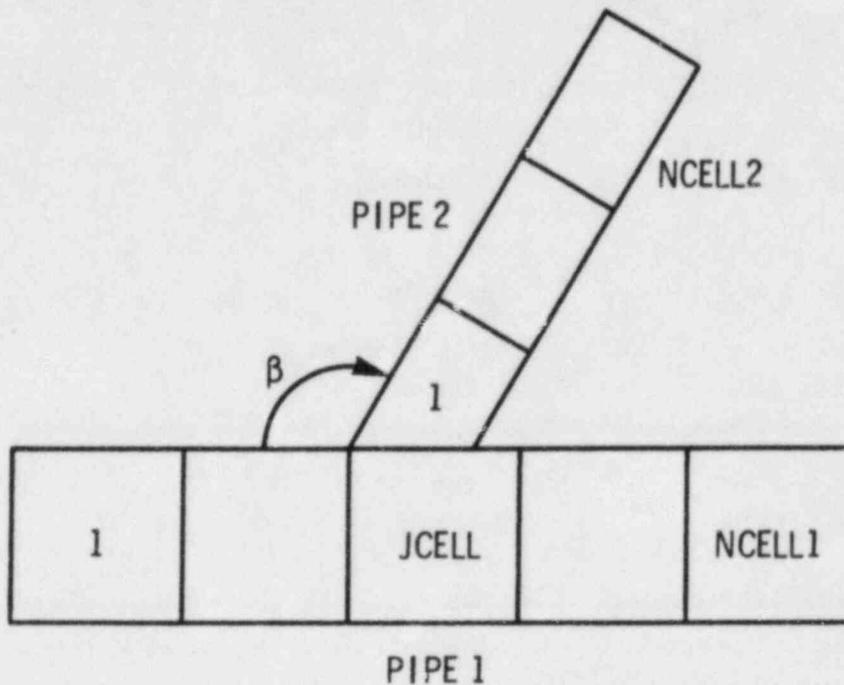


FIGURE 3.45. Tee Noding Diagram

of the code, the tee is treated as two PIPES, as indicated in Figure 3.45. The angle β is from the low-numbered end of PIPE 1 to PIPE 2. The low-numbered end of PIPE 2 always connects to PIPE 1. The straight PIPE segment is numbered from cell 1 to NCELL1, with the connection to PIPE 2 at cell JCELL. The branch PIPE segment is numbered from the cell immediately adjacent to JCELL, beginning with cell 1 and ending with cell NCELL2.

The connection to PIPE 1 from PIPE 2 is treated with mass, momentum and energy source terms. For PIPE 2 the conditions in cell JCELL of PIPE1 form the inlet boundary conditions. The time differencing and iteration procedure are such that the scalar quantities are conserved (within a convergence tolerance) and the level of implicitness at the connection ensures that no additional stability limitations apply at a TEE. The levels of implicitness for the finite-difference equations applied to PIPE 1 and PIPE 2 can be specified independently using the input variables IHYD1 and IHYD2. Since the junction between PIPE 1 and PIPE 2 is always treated partially implicitly, the velocity at that point is always included in the computation of the time step stability limit. Phase separation at the junction is calculated if the flag

ISEP is set to one. Phase separation is computed if the void fraction in the junction cell JCELL exceeds the user-specified value ALSEP.

Since the TEE is modeled as essentially two connected PIPEs, the PIPE model description in Section 3.5.3 should be referenced for additional information on the calculational sequence. The sequence for a TEE includes separate calculations of the primary and secondary sides.

Input and output information for a TEE component is very similar to that for a PIPE component except that two PIPEs are involved in a TEE component.

3.5.8 VALVE Module

The VALVE module is used to model the thermal-hydraulic flow in a valve. A valve is modeled as a one-dimensional component with at least two fluid cells, as shown in Figure 3.46. The treatment of heat transfer and fluid dynamics in a valve is identical to that of a pipe, as described in Section 3.5.3.

The valve action is modeled by controlling the flow area and hydraulic diameter at a given cell boundary in the component. The following expressions are used for this purpose:

$$\text{Flow Area} = \text{AVLVE} * \text{FRACT}$$

$$\text{Hydraulic Diameter} = \text{HVLVE} * \text{FRACT}$$

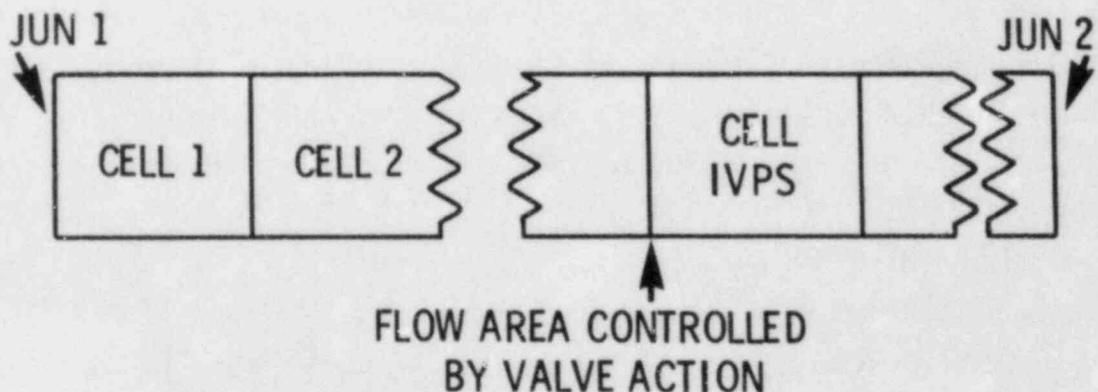


FIGURE 3.46. Valve Noding Diagram

where AVLVE and HVLVE are the fully open valve flow area and hydraulic diameter, respectively, and FRACT is the fraction the valve is open.

Five options are provided for controlling the valve action. Options 1 through 4 open or close the valve with a trip. The action can be instantaneous or a function of time. Option 5 models a check valve with the open or closed condition determined by a pressure differential between the specified cells and a setpoint. (For this option, the valve opening and closing is damped to prevent pressure oscillations.) The valve option is specified by the value of the input parameter IVTY. The possible values of IVTY and corresponding options are shown in Table 3.4.

TABLE 3.4. VALVE Activation Options

IVTY	OPTION
1	Valve is normally open and is closed instantly on a trip signal.
2	Valve is normally closed and is opened instantly on a trip signal.
3	Valve is normally open and is closed on a trip signal according to a time-dependent valve table.
4	Valve is normally closed and is opened on a trip signal according to a time-dependent valve table.
5	Check valve is controlled by a static pressure gradient. If IVPG = 1, then $DP = P(IVPS - 1) - P(IVPS)$; if IVPG = 2) then $DP = P(IVPS) - P(IVPS - 1)$.

If $DP + PVS > 0$, the valve opens.
 If $DP + PVS < 0$, the valve closes.

The valve opening and closing is damped so that flow area and hydraulic diameter can change by only 1% per time step. This damps pressure oscillations in the system caused by the valve action.

3.6 STEADY-STATE CALCULATIONS

The current version of COBRA/TRAC does not have a steady-state implicit solution algorithm. In developing the COBRA-TF code, and later COBRA/TRAC, the main focus was on the ability to calculate transients, and the problem of obtaining the steady state from which the transients were initiated was largely ignored. As a result, problems that begin with a stable steady state must be initialized by running a transient calculation to obtain a solution for the steady-state conditions of the system. The dump/restart capability (discussed in Section 3.7) makes it possible to change the boundary conditions and characteristics of the problem with relative ease. Volume 4, COBRA/TRAC Developmental Assessment and Data Comparisons, describes a number of calculations that were initialized to steady-state conditions by running a transient calculation with constant boundary conditions and then restarted with transient boundary conditions to simulate a blowdown, small break or similar de-stabilizing occurrence within the system.

Obtaining a steady-state solution with COBRA/TRAC is relatively straightforward. Essentially, the procedure does not differ from that for any other calculation. There are a few points, however, that the user should keep in mind in order to waste as little computer time as possible in establishing the steady state (from which, it is presumed, the transient of real interest will be initiated.)

- 1) Turn the pumps on before adding heat to the system. The flow through the system should be established before any significant amount of energy is added.
- 2) Ramp the power up to the steady-state value at a rate that will make boiling in the VESSEL unlikely. (This assumes the steady state is completely subcooled.) Boiling and condensing in the transient to steady state can result in pressure oscillations that will slow convergence to the steady state.
- 3) Trip set points that might be activated in the course of the transient to steady-state calculation should not be specified until the desired steady state has been reached. (New trips can be added on a restart.)

- 4) Forcing functions on boundary conditions in the VESSEL must be specified in the initial input as constant for a specific period of simulation time. The time at which these functions begin varying becomes, by definition, the point of initiation of the transient of interest.
- 5) On the restart that initiates the transient from the steady-state conditions, the minimum time step should be specified at a value (on the order of $1.0E-6$) and a trip identifier number of 1000 should be specified to force the code to use the minimum time step size.
- 6) PIPE or TEE components modeling broken pipes that are expected to experience choked flow in the transient should be converted to the implicit solution by setting the flag IHYDRO to 1. (In the transient to establish steady-state conditions, these components are not yet broken, so it is more efficient to use the partially implicit hydrodynamics (IHYDRO = 0) in that part of the calculation.)
- 7) PIPE or TEE components that model broken pipes with choked flow should be renoded (on the restart that initiates the blowdown transient) to reflect the noding patterns recommended for choked flow in TRAC-PIA Development Assessment (Ref. 3).

Point 4 deserves some additional comment since it determines how long the transient to steady state will be. It must be chosen correctly the first time, since the user cannot alter the VESSEL input data on restarts. As a rule of thumb, the time-to-steady-state should be on the order of the time it takes one particle of fluid to circulate twice through the main flow path of the system. This assumes that the temperatures, both fluid and material, in the system have been initialized close to their steady-state values. If an appreciable mass of solid material must be heated up to steady-state temperatures, two or three passes through the loop might be required before stable wall temperatures are achieved. (The convergence of wall temperatures may be accelerated using the RTWFP parameter on the time step control cards.)

The time step adjustment recommended in point 5 is needed because the code calculates the new time step based on the behavior of the system in the

previous time step. If the system is at a stable steady state and a transient is begun, or if the system is changing extremely rapidly because of a fast transient, the code can calculate a time step that is too large for stability. The code has the capability to "back up" the solution numerically when the time step is too large for convergence, but this is expensive in terms of wasted calculation time and should be avoided where possible.

The modeling of PIPEs or TEEs that will experience choked flow is very important for blowdown transients and must not be omitted. Reference 3 contains detailed recommendations on how to model pipes in choked flow. Basically, the length of the pipe in choked flow is modeled by a large number of extremely small nodes, decreasing from the nominal pipe node length gradually (by ~50% in each length reduction) to 0.1, 0.05, 0.025, and 0.0083 meters and, in some extreme cases, even smaller. But these very small nodes should not be used in the transient to steady state with the pipe partially implicit. They will cause the time-step size to be unnecessarily Courant-limited, making the transient to steady-state conditions use more computer time (and hence be more expensive) than it needs to be.

3.7 USING THE DUMP/RESTART OPTION

The DUMP/RESTART option in COBRA/TRAC makes it possible to run long transient calculations efficiently. Long transients modeling large systems can require many hours of computer time to execute. The DUMP/RESTART capability allows the user to run the transient as a series of shorter runs, each restarting at the point the previous run left off. The user can then monitor the calculation as it progresses and minimize wasted computer time resulting from input errors and computer system failures.

The DUMP/RESTART option is relatively easy to implement. This section discusses the three types of restarts a user might want to execute; a simple restart with no changes in the input, a restart with changes in the input for one or more components, and a restart with new components added to the system.

3.7.1 Simple Restart

Every COBRA/TRAC run generates two DUMP files. Data for the one-dimensional components are written to a file named TRCDMP, and data for the VESSEL component are written to a file named TAPE9. (If the graphics option is being used, a file named TAPE11 is generated also. This has nothing to do with the restart per se, but TAPE11 must also be accessible for restarts if the graphics output option is being used.) These files, TRCDMP, TAPE9 and TAPE11, are local files, and the user must do whatever is appropriate on his system to save them permanently for later access.

The dump files TRCDMP and TAPE9 contain data describing the state of the system for the one-dimensional components and the VESSEL, respectively, at specific time steps. The line-printer output for the calculation contains, for each time step written to the dump file, the following message:

```
SAVED A RESTART DUMP AT NNNN XX.XXX
```

where NNNN is an integer number identifying the time step and XX.XXX is a real number identifying the corresponding simulation time. Dumps are saved at the time intervals specified by input on the time domain cards (see Section 2.4) every 3600 CPU seconds and at the end of each run. The user may specify any interval he deems appropriate for his problem, but since these files can become quite large, excessive dumps should be avoided. The code automatically writes a time step to the dump files every thirty minutes of CP time in a given run and at the end of the run. For most long transients this is quite adequate. The calculation may be restarted from any time step written to the dump file, but in most cases the last time step on the file will be the one used to restart the simulation.

The input file for a restarted run is in general much simpler than that for the initial run, since most of the required information is stored on the dump files. But some data must be repeated or changed. An example of a simple restart file is shown in Table 3.5. The main problem control data (see Section 2.1) is repeated in the restart input file. On a simple restart this data is the same as on the initial run, except for the restart time data. In

TABLE 3.5. Simple Restart File

```

1
semiscale mod3 - steady state(s-07-6)
3749      18.77200
21      22
1.      10      20
2
1      2      3      4      5
6      7      8      9      10
11     12     28     29     30
31     16     21     37     38
40
1      0      20.
4
2      0      20.
8
vessel      40      40
4 *** semiscale mod 3 *test s-07-6* cobra/trac ***

5
25 3 0 6
23 3 0 7
18 3 0 1
19 3 0 9
35 6 -1 40
end

10 300 60
40 6 41 11 12 1 2 9 16 5
1 27 5 1 19 2 37 17 2 37 9 9 37 9 2
37 33 7 37 31 5 37 18 2 37 7 12 37 8 12
37 7 6 37 8 6 37 5 2 37 3 2 35 1 54.
35 1 78. 35 1 114. 35 1 126. 35 8 54. 35 8 78.
35 8 126. 35 8 150. 35 -3 162. 35 -4 306. 35 -5 258.
5 17 3 5 9 8 5 9 3 5 1 3 5 8 2
5 8 6 5 7 8 5 8 8 5 8 13 5 30 5
6 17 3 6 9 8 6 9 3 6 1 3 6 8 2
6 8 6 6 7 8 6 8 8 6 8 13 6 30 5
35 -7 3. 1 5 2 1 10 2 1 24 3 1 1 2
37 7 9 37 8 9 1 12 3 1 19 2 1 10 1
1 10 3 37 7 4 37 8 4 5 35 3 6 35 3
0.1e-3      .05      19.99999      400.
50.      10.0      10.5      10.5
-1.0

```

the example, line 3 shows the restart time step as 3749 and the restart time as 18.772 seconds. The one-dimensional component data has been omitted entirely since all necessary data is on the dump file. The VESSEL input consists of the VESSEL component card and one line to set the restart flag and reinitialize the VESSEL title array. (See Section 2.3.) A blank line must be inserted at this point so the code will read the VESSEL data from the restart file TAPE8 rather than the input file. The input for the VESSEL connections must be repeated on the restart, since it is not saved on the restart file. The component input is terminated with an END card at the end of the connection input. The code will then look for the data for the remaining undefined components on the one-dimensional component restart file, TRCRST. The graphics input data must also be repeated in the restart input file since it is not saved to TAPE9. It must be the same as that specified in the initial run, except that new components can be added in elements of the IGRFIT array that were initially zero. New time domain cards are also specified in the restart input file.

During execution of a restarted run, the dump tapes read for the restart must be available in local files. The VESSEL restart data is read from file TAPE8, so the appropriate dump file TAPE9 must be specified as TAPE8 for the restart. Similarly, the one-dimensional component restart data is read from file TRCRST, which must correspond to the appropriate dump file TRCDMP. If a graphics file is being created, the current TAPE11 must be available locally also, and must be accessible for both reads and writes.

3.7.2 Restart with Changes to Components

Only the one-dimensional components may be changed on a restart. The VESSEL component input cannot be revised. The input file for a restart with changes to components is essentially identical to that for a simple restart. The only difference is that the input data for the one-dimensional component (or components) being changed is specified in the input file. The new input replaces the data for those components stored in the dump file.

An example of a restart with changes to a component is shown in Table 3.6. The most common reason for changing a component is the belated discovery of an error in the original input specifications. In this example,

it was discovered that the flow rate on the secondary side of the steam generator component was incorrect, so the appropriate FILL components were re-initialized with new values for the inlet velocity.

In this case, the modeling change was relatively insignificant. But in cases where geometry changes will drastically change the flow solution in a component, the solution arrays should be re-initialized with values compatible with the new geometry. Estimates of the new values can be obtained by hand calculations with the conditions at the component's junctions as boundary conditions.

It is possible to replace a component in a system with a different component on a restart. A BREAK could be replaced by a FILL, for example, if it were expedient to change at some point in a calculation from a pressure boundary to a flow boundary condition.

3.7.3 Restart with New Components

In addition to redefining or replacing components in a system on a restart, it is possible to add new components. The restart input file must, in this case redefine the control data, increasing the total number of components and junctions and adding the new component numbers to the ORDER array. The input for the new components is specified in the restart input file before the VESSEL restart data. (Only one-dimensional components may be added. It is not possible to add a second VESSEL component in this way.)

An example of a restart file with new components is shown in Table 3.7. The number of components in the system has been increased from 21 to 29 and the number of junctions from 22 to 29. In this example, the new components model the ECC injection system and the break assembly in the Semiscale test facility. They are added to the system model when the calculation has reached steady state (see Section 3.6) on the restart where the transient is initiated.

TABLE 3.6. Restart with Changes to Components

1					
semiscale mod3 - steady state(s-07-6)					
3749	18.77200				
21	22				
1.	10	20			
2					
1	2	3	4	5	
6	7	3	9	10	
11	12	28	29	30	
31	16	21	37	38	
40					
1	0		20.		
4					
2	0		20.		
8					
f111	28	28			
21	1				
1.0	1.0	0.0	1.1037e-1	495.00e0	
5.91e6					
f111	30	30			
10	1				
.26	.01	0.	1.3588e-2	495.00e0	
59.10e+5					

TABLE 3.6. (continued)

```

vessel          40          40
4 *** semiscale mod 3 *test s-07-6* cobra/trac ***

5
25  3  0  6
23  3  0  7
18  3  0  1
19  3  0  9
35  6 -1 40
end

10 300 60
40  6 41 11 12  1  2  9 16  5
  1 27  5  1 19  2 37 17  2 37  9  9 37  9  2
37 33  7 37 31  5 37 18  2 37  7 12 37  8 12
37  7  6 37  8  6 37  5  2 37  3  2 35  1 54.
35  1 78. 35  1 114. 35  1 126. 35  8 54. 35  8 78.
35  8 126. 35  8 150. 35 -3 162. 35 -4 306. 35 -5 258.
  5 17  3  5  9  8  5  9  3  5  1  3  5  8  2
  5  8  6  5  7  8  5  8  8  5  8 13  5 30  5
  6 17  3  6  9  8  6  9  3  6  1  3  6  8  2
  6  8  6  6  7  8  6  8  8  6  8 13  6 30  5
35 -7  3.  1  5  2  1 10  2  1 24  3  1  1  2
37  7  9 37  8  9  1 12  3  1 19  2  1 10  1
  1 10  3 37  7  4 37  8  4  5 35  3  6 35  3
      0.1e-3          .05      19.99999          400.
      50.           10.0       10.5           10.5
      -1.0

```

TABLE 3.7. Restart with Added Components

1					
semiscale mod3 - blowdown (s-07-6)					
3838	20.014				
29	29				
.00100	10	20			
2					
1	2	3	4	5	
6	7	8	9	10	
11	12	28	29	30	
31	16	21	37	38	
40	41	42	17	18	
32	33	34	26		
1	0		20.		
4					
2	0		20.		
8					
pipe	6	6			
21	1	12	7	7	
0	1				
1.69926e-2	7.1374e-3				
295.					
r 9	.00833r 3	.025r 3	.05r 3	.1r 2	.25741
	.51481e				
r 9	2.563e-6r 3	.7689e-5r 3	1.5377e-5r 3	3.0753e-5r 2	1.9486e-4
	4.67e-4e				
r19	2.7540e-4s				
r 3	9.0713e-4e				
r19	0.r 3	3.5e-02e			
f	0. e				
r19	2.41750e-2s				
r 3	3.39852e-2e				
r18	4	-4s			

TABLE 3.7. (continued)

r 3	4e				
f	0. e				
f	3.2e-4e				
f	0.00e				
f	561.8 e				
i18	1.64736e+7	1.64789e+07	1.64843e+7e		
f	561.50 e				
pipe		41	41		
	19	1	13	14	7
	0	1			
	1.69926e-2	7.1374e-3			
	295.				
	.51481r 2	.21481r 3	.1r 3	.05r 4	.025
r 6	.00833e				
	4.67e-4r 2	1.9486e-4r 3	3.0753e-5r 3	1.5377e-5r 4	0.7689e-5
r 6	2.5630e-6e				
r 3	9.0713e-4s				
r17	9.6953e-5e				
r 3	3.5e-2r17	0.0e			
f	0.e				
r 3	3.39852e-2s				
r17	1.11111e-2e				
r 3	4	-4s			
r16	4e				
f	0.e				
f	3.00e-4e				
r 3	3.300s				
r17	30.78 e				
f	561.7e				
	1.64683e+7i16	1.646290e7	1.645760e+7e		
f	561.50e				

TABLE 3.7. (continued)

break		26	26		
	12	0	0		
	.00833	2.5630e-6	1.	400.	.24527e+6
break		42	42		
	14	0	0		
	.00833	2.5630e-6	1.	400.	.24527e+6
fill		33	33		
	34	2	0	17	
	1.	1.2668e-4	0.	0.	312.
	1.0047e+6				
	0.	.01727	51.0428	.00493	51.5646
	.125800	52.0865	1.19956	52.6080	1.4528
	53.1298	1.4898	54.6949	1.6641	56.2601
	1.66498	57.3036	1.7620	60.4340	1.8360
	67.8261	1.8776	78.2609	1.8364	83.4782
	1.7309	109.5652	1.8286	120.	1.8221
	224.3478	1.89793	460.	1.89e	
fill		34	34		
	35	2	0	16	
	1.	1.2668e-4	0.	0.	312.
	7.6378e+6				
	0.	.005780	22.3477	.0057800	22.8694
	.08941	23.3911	.56617	23.91290	.51067
	24.4346	.46712	25.4781	.50258	27.5650
	.46636	34.8692	.50181	40.0865	.51222
	49.9994	.52070	78.2609	.51449	88.6956
	.50019	172.1739	.49711	224.3478	.50582
	469.5651	.50952e			

TABLE 3.7. (continued)

accum		18		18	
	4	24			
	.3799	.4354	.29564	.29564e	
	.0249	.028565	.019394	.019394e	
	.0656	.0656	.0656	.0656	.03280e
f	0. e				
f	-1. e				
	.2889	.2889	.2889	.2889	.204358e
f	le				
	1.	0.	0.	0.e	
f	0. e				
f	312. e				
f	312. e				
f	4.313e+06e				
valve		32	32		
	15	0	24	26	7
	.0127	.0064			
	295.				
	5	0	0	1	8
	5.0671e-4	.0254	0.		
r 6	.1503	.3006r 2	.5418	.6474	1.2948
	2.5896	1.2948	1.0823	.7874e	
	4.92984e-3	2.46492e-3	1.23246e-3	6.1623e-4	3.08115e-4
	1.54058e-4	1.52317e-4r 2	2.74535e-4	3.28044e-4	6.56088e-4
	1.31218e-3	6.56088e-4	5.48412e-4	3.98983e-4e	
	.0328	.0164	.0082	.0041	.00205
	.001025r10	5.0671e-4e			
f	0.e				
r 4	0.r 5	-1.r 5	0.	1.	0.e
	.204358	.1445	.102173	.07225	.05072
	.03613r10	.0254e			

TABLE 3.7. (continued)

f	4e			
f	0.e			
f	0.e			
f	0.e			
r 7	312.	r 8	558.8e	
r 7	4.313e+6r	8	1.64695e7e	
tee			16	16
	2		0	7
	0		3	26
	.0127		.0064	0.
	295.			33
	0		1	
	.006350		.0064	
	295.			
f	.4064e			
f	1. e			
f	2.0593e-4e			
f	1.2668e-4e			
	5.0671e-4r	2	5.0671e-4	4.6400e-4e
f	1.2668e-4e			
r 2	0.		.153	0.e
f	0. e			
f	0.e			
f	0. e			
r 3	.0254	.0243078	e	
f	.0127e			
f	4e			
f	4e			
f	0. e			
f	0. e			
f	0. e			
f	0. e			

TABLE 3.7. (continued)

f	0. e				
f	0. e				
f	558.8 e				
f	558.8 e				
f	1.64695e+7e				
f	1.64695e+7e				
tee		17	17		
	2	0	7	G.	0
	0	3	35	28	
	.00635	.0064			
	295.				
	0	1	34		
	.00635	.0064			
	295.				
f	.3333e				
f	1. e				
f	4.223e-05e				
f	1.2668e-4e				
f	1.2668e-4e				
f	1.2668e-4e				
f	0.e				
f	0.e				
f	0.e				
f	0.e				
f	.0127e				
f	.0127e				
f	4e				
f	4e				
f	0.e				
f	0.e				
f	0.e				
f	0.e				

TABLE 3.7. (continued)

f	0.e							
f	0.e							
f	558.8 e							
f	558.8 e							
f	1.64695e+7e							
f	1.64695e+7e							
fill		28		28				jp 201
	21	1						jp 202
	1.0	1.0		0.0	1.1037e-1		546.	
	5.91e6							jp 204
break		29		29				jp 205
	22	1		8		3		
	1.0	1.0		1.0	546.00e0		6.60e6	
	0.	6.60e6		30.	6.60e6		50.	
	6.50e6	60.		6.45e6	120.		6.42e6	
	220.	6.18e6		320.	6.10e6		420.	
	6.00e6e							
fill		30		30				jp 862
	10	1						jp 863
	.26	.01		0.	1.3588e-2		546.	
	59.10e+5							jp 865
break		31		31				jp 866
	11	1		14		3		
	1.0	1.0		1.0	545.00e0		57.50e5	jp 868
	0.	5.75e6		20.	5.75e6		22.	
	6.35e6	24.		6.50e6	26.		6.60e6	
	30.	6.52e6		34.	6.38e6		40.	
	6.25e6	50.		6.10e6	60.		6.00e6	
	120.	5.70e6		220.	5.45e6		320.	
	5.30e6	405.		5.10e6e				

TABLE 3.7. (continued)

```

prizer          5          5
                5          8
                8.00e3    152.10e5    1.00e5    10.0e+0
                7214.41e-4  3932.34e-4  1932.34e-4r 2    0.10e0e
                16.42e-3    8.95e-3    4398.01e-6r 2    22.76e-4e
r 5 2.2760 e-2    6.94e-5e
r 5 0.            7.91220e-5e
f -1.0          e
r 5 .17         9.40e-3e
r 5 +4          +1e
                .9095    .03538    .0119    7.322e-3    4.313e-3e
                0.    -1.535e-3    -1.789e-3    -1.916e-3    -1.971e-3
                -6.620e-1e
                621.7    618.2    614.4    609.2    600.1e
f 621.7e
                1.62397e+7    1.62414e+7    1.62431e+7    1.62440e+7    1.62447e+7e
vessel          40          40

```

4 *** semiscale mod 3 *test s-07-6* cobra/trac, cycle 12 ***

```

0
5
25 3 0 6
23 3 0 7
18 3 0 1
19 3 0 9
35 6 -1 40
end

10 300 60
40 6 41 11 12 1 2 9 16 5
1 27 5 1 19 2 37 17 2 37 9 9 37 9 2
37 33 7 37 31 5 37 18 2 37 7 12 37 8 12
37 7 6 37 8 6 37 5 2 37 3 2 35 1 54.

```

TABLE 3.7. (continued)

35	1	78.	35	1	114.	35	1	126.	35	8	54.	35	8	78.
35	8	126.	35	8	150.	35	-3	162.	35	-4	306.	35	-5	258.
5	17	3	5	9	8	5	9	3	5	1	3	5	8	2
5	8	6	5	7	8	5	8	8	5	8	13	5	30	5
6	17	3	6	9	8	6	9	3	6	1	3	6	8	2
6	8	6	6	7	8	6	8	8	6	8	13	6	30	5
35	-7	3.	1	5	2	1	10	2	1	24	3	1	1	2
37	7	9	37	8	9	1	12	3	1	19	2	1	10	1
1	10	3	37	7	4	37	8	4	5	35	3	6	35	3
	0.1e-7			1.e-6			20.01800						600.	
	55.0			.40			50.0		5000.000					
	0.1e-7			0.05			40.0					600.		
	55.0			.40			50.0		5000.0					
	-1.0													

4.0 SAMPLE PROBLEM

COBRA/TRAC is a complex code that requires some skill and experience to use correctly. The user guidelines in Section 3 are intended as an aid when learning to apply the code. Additionally, Volume 4, the Applications Manual, contains descriptions and calculational results of a large number of problems that have been run on COBRA/TRAC.

This section provides an example illustrating the process of constructing a COBRA/TRAC input file for a moderately complex system using the PKL facility for test K9 (Ref. 7). The COBRA/TRAC model of the system uses most of the one-dimensional components and exercises many of the modeling features of the VESSEL component. It does not, of course, exercise every feature and option available in COBRA/TRAC^(a), but it uses enough of the code's capabilities to serve as a useful example.

This section follows the logic generally employed in setting up an input deck for a COBRA/TRAC simulation. It begins with a description of the system to be modeled and the test conditions to be simulated. The data are presented in the form in which they appear in the original reference. (It should be pointed out, however, that test reports do not appear to be written with the convenience of the code user in mind, and the needed data will seldom be as neatly summarized or as concisely listed as shown here.) The next step is to decide on an appropriate set of COBRA/TRAC components to model the system and construct the geometric input from the system data. The test parameters can then be converted to boundary conditions on the problem, and the simulation is ready to run.

(a) The PWR vessel with UHI, described in Volume 6 of this report, comes as close as humanly possible to achieving this. The full vessel model will be a useful guide to anyone wishing to model a large complex system.

4.1 PKL SYSTEM DESCRIPTION FOR TEST K9

The PKL test facility was designed to model the thermal-hydraulic behavior of a 1300 MW PWR. A diagram of the facility is shown in Figure 4.1. The system consists of a pressure vessel, external downcomer, and three primary coolant loops. One loop models two loops of a 4-loop plant, and the other two each model a single loop. Each coolant loop contained a U-tube steam generator and an assembly simulating the hydraulic resistance of a shut-down coolant pump. Details describing the physical geometry of the system, which are needed to construct the COBRA/TRAC input model, are given in Tables 4.1 and 4.2.

The pressure vessel contains 340 simulated fuel rods (337 electrically heated). The upper and lower plenums of the vessel are scaled to PWR dimensions and contain appropriate internals. Figure 4.2 shows a horizontal cross-section of the core and Figure 4.3 shows a vertical cross-section of the pressure vessel. A diagram of the external downcomer is shown in Figure 4.4. Relevant dimensions and elevations are included. Table 4.3 summarizes the geometric characteristics of the pressure vessel that are relevant to the COBRA/TRAC input model.

The material properties of the components of the system are summarized in Table 4.4. The pressure vessel and primary system piping were CrNi-steel. The pipe wall thickness was 4.05 mm in the single loops and 4.0 mm in the double loop. The heater rods were composed of NiCr 80/20 tubing (1.0 mm thick) and were filled with powdered magnesia (MgO). A helical coil of nichrome wire (0.2 mm thick, coiled on a 6.0 mm diameter) served as the heat source. The axial power profile was produced by varying the pitch of the heater coil. The seven-step profile is shown in Figure 4.5.

Test K9 starts at the refill phase of a LOCA; i.e., after the blowdown has occurred, with pressure equalized between containment and the reactor primary system. Prior to test initiation, the primary system was purged with

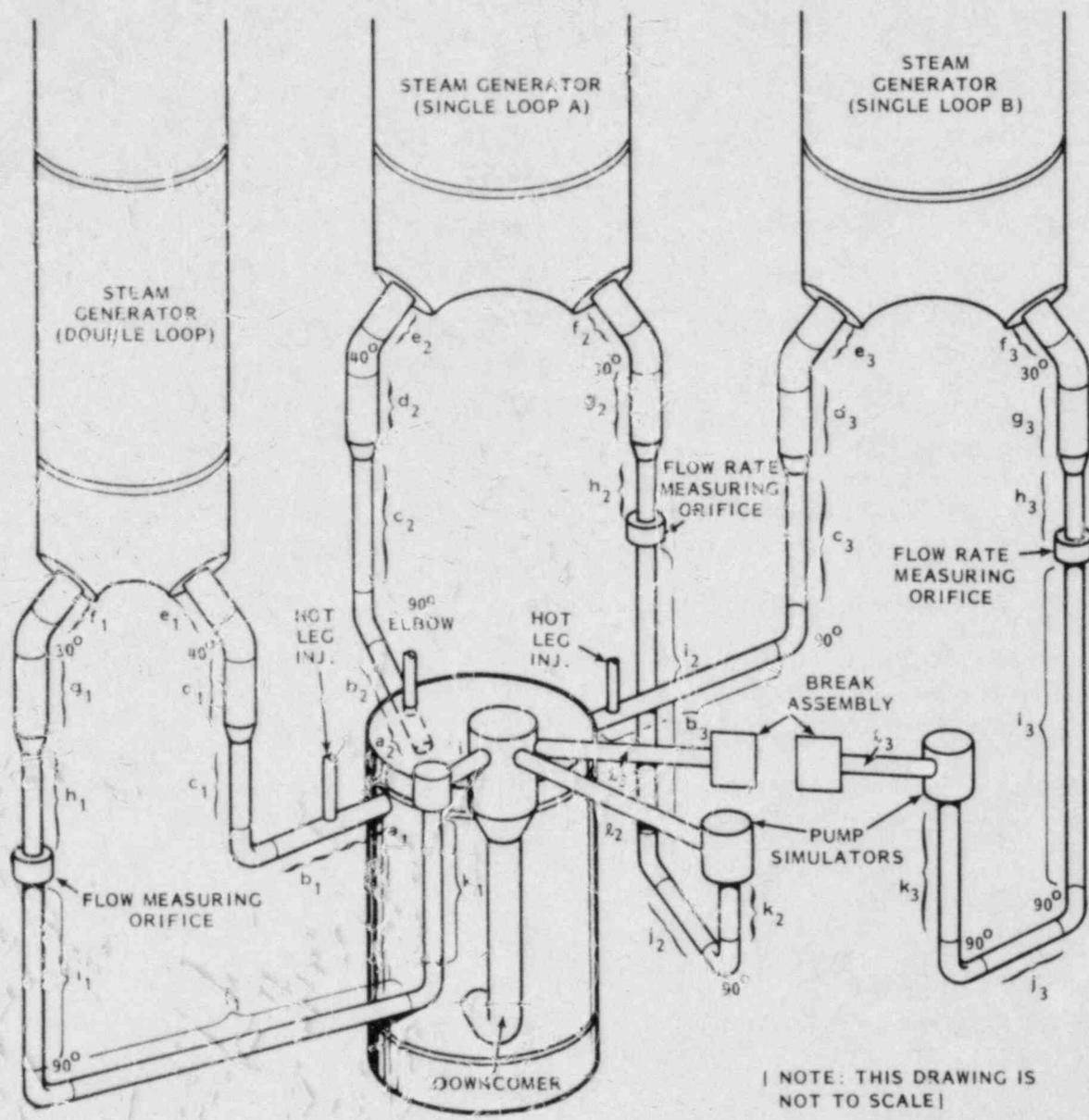


FIGURE 4.1. Diagram of PKL Test Facility as Configured for Test K9

TABLE 4.1. Geometry of PKL System

Component	Loss ^(a) Coefficient	Length(m)	Diameter(m)	Cross-Sectional Area (m ²)
Double Loop				
hot leg				
plenum outlet	0.5		0.113	0.01
pipe segment(a)	0.015	0.122	0.113	0.01
hot leg injection line	0.233	0.448	0.088	0.0084
pipe segment (b)	0.27	2.265	0.113	0.01
90° bend, (R=C.17m)	0.231	0.267	0.113	0.01
pipe segment (c)	0.025	0.21	0.113	0.01
pipe enlargement	0.155	0.22	0.113/0.175	0.01/0.024
pipe segment (d)	0.015	1.2	0.175	0.024
40° bend (R=0.262 m)	0.021	0.184	0.175	0.024
pipe segment (e)	0.004	0.327	0.175	0.024
steam generator				
inlet plenum		0.087		
entrance	0.091	0.275	0.024	
tube inlet	0.124		0.019	16.98(10 ⁻³)
tube bundle (60 tubes)	4.15	17.49	0.019	16.98(10 ⁻³) 2.84(10 ⁻⁴) per tube
outlet plenum				0.087
tube exit	0.349		0.019	16.98(10 ⁻³)
exit	0.067		0.075	0.024
cold leg				
pipe segment (f)	0.004	0.327	0.175	0.024
30° bend, (R=0.262 m)	0.016	0.138	0.175	0.024
pipe segment (g)	0.015	1.2	0.175	0.024
pipe contraction	0.01	0.22	0.175/0.113	0.024/0.01
pipe segment (h)	0.159	1.33	0.113	0.01
flowrate measuring orifice	1.5	0.002	0.08752	0.00601
pipe segment (i)	0.31	2.58	0.113	0.01
90° bend, (R=0.12 m)	0.231	0.267	0.113	0.01
pipe segment (j)	0.119	1.0	0.113	0.01
90° bend, (R=0.12 m)	0.231	0.267	0.113	0.01
pipe segment (k)	0.38	3.18	0.113	0.01
pump simulator	1.43	0.25	0.113	0.01
pipe segment (l)	0.348	2.916	0.113	0.01
inlet to downcomer nozzle	1.0		0.113/0.199	0.01/0.0188

(a) For single loops, loss coefficient is computed based on D=0.0808; for double loop, loss coefficient is computed based on D=0.113.

TABLE 4.1. (continued)

Component	Loss Coefficient	Length(m)	Diameter(m)	Cross-Sectional Area (m ²)
Single Loop A -				
hot leg				
plenum outlet	0.5		0.0808	0.0051
pipe segment (a)	0.014	0.08	0.0808	0.0051
hot leg injection line	0.302	0.342	0.061	0.0042
pipe segment (b)	0.271	1.563	0.0808	0.0051
90° bend, (R=0.12 m)	0.245	0.188	0.0808	0.0051
pipe segment (c)	0.057	0.33	0.0808	0.0051
pipe enlargement	0.132	0.15	0.0808/0.125	0.0051/0.0123
pipe segment (d)	0.022	1.2	0.125	0.0123
40° bend, (R=0.188 m)	0.022	0.13	0.125	0.0123
pipe segment (e)	0.005	0.29	0.125	0.0123
steam generator				
inlet plenum				0.056
entrance	0.106		0.125	0.0123
tube inlet	0.131		0.019	8.5(10 ⁻³)
tube bundle (30 tubes)	4.36	17.31	0.019	8.5(10 ⁻³)
outlet plenum				0.056
tube exit	0.364		0.019	8.5(10 ⁻³)
exit	0.070		0.125	0.0123
cold leg				
pipe segment (f)	0.005	0.29	0.125	0.0123
30° bend, (R=0.188 m)	0.017	0.098	0.125	0.0123
pipe segment (g)	0.0218	1.2	0.125	0.0123
pipe contraction	0.01	1.15	0.125/0.0808	0.0123/0.0051
pipe segment (h)	0.242	1.4	0.0808	0.0051
flowrate measuring orifice	1.5	0.0016	0.0626	0.00308
pipe segment (i)	0.455	2.63	0.0808	0.0051
90° bend, (R=0.12 m)	0.245	0.188	0.0808	0.0051
pipe segment (j)	0.242	1.4	0.0808	0.0051
90° bend (R=0.12 m)	0.245	0.188	0.0808	0.0051
pipe segment (k)	0.570	3.265	0.0808	0.0051
pump simulator	1.448	0.215	0.0808	0.0051
pipe segment (l)	0.356	2.056	0.0808	0.0051
inlet to downcomer nozzle	1.0		0.0808/0.489	0.0051/0.188

TABLE 4.1. (continued)

Component	Loss Coefficient	Length(m)	Diameter(m)	Cross-Sectional Area (m ²)
Single Loop B -				
hot leg				
plenum outlet	0.5		0.0808	0.0051
pipe segment (a)	0.014	0.08	0.0808	0.0051
hot leg injection line	0.302	0.342	0.061	0.0042
pipe segment (b)	0.271	1.563	0.0808	0.0051
90° bend, (R=0.12 m)	0.245	0.188	0.0808	0.0051
pipe segment (c)	0.057	0.33	0.0808	0.0051
pipe enlargement	0.132	0.15	0.0808/0.125	0.0051/0.0123
pipe segment (d)	0.022	1.2	0.125	0.0123
40° bend, (R=0.188 m)	0.022	0.13	0.125	0.0123
pipe segment (e)	0.005	0.29	0.125	0.0123
steam generator				
inlet plenum				0.056
entrance	0.106		0.125	0.0123
tube inlet	0.131		0.019	8.5(10 ⁻³)
tube bundle (30 tubes)	4.36		0.019	8.5(10 ⁻³)
outlet plenum				0.056
tube exit	0.364		0.019	8.5(10 ⁻³)
exit	0.070		0.125	0.0123
cold leg				
pipe segment (f)	0.005	0.29	0.125	0.0123
30° bend, (R=0.188 m)	0.017	0.098	0.125	0.0123
pipe segment (g)	0.0218	1.2	0.125	0.0123
pipe contraction	0.01	0.15	0.125/0.0808	0.0123/0.0051
pipe segment (h)	0.242	1.4	0.0808	0.0051
flowrate measuring orifice	1.5	0.0016	0.0626	0.00308
pipe segment (i)	0.455	2.63	0.0808	0.0051
90° bend, (R=0.12 m)	0.245	0.188	0.0808	0.0051
pipe segment (j)	0.242	1.4	0.0808	0.0051
90° bend (R=0.12 m)	0.245	0.188	0.0808	0.0051
pipe segment (k)	0.570	3.265	0.0808	0.0051
pump simulator	1.448	0.215	0.0808	0.0051
pipe segment (l)	0.356	2.056	0.0808	0.0051
inlet to downcomer nozzle	1.0			

TABLE 4.2. Geometry of PKL Steam Generators

<u>Loop Type</u>	<u>Nb. of Tubes</u>	<u>Tube Length^(a) (mm)</u>	<u>Height (mm)</u>	<u>Center Pos. Width (mm)</u>	<u>Thickness (mm)</u>	<u>Secondary Tube Diameter (mm)</u>
Single	30	8430	8215	350	100	383
Double	60	8430	8215	350	100	475

(a) from tube sheet inlet to top of U-bend

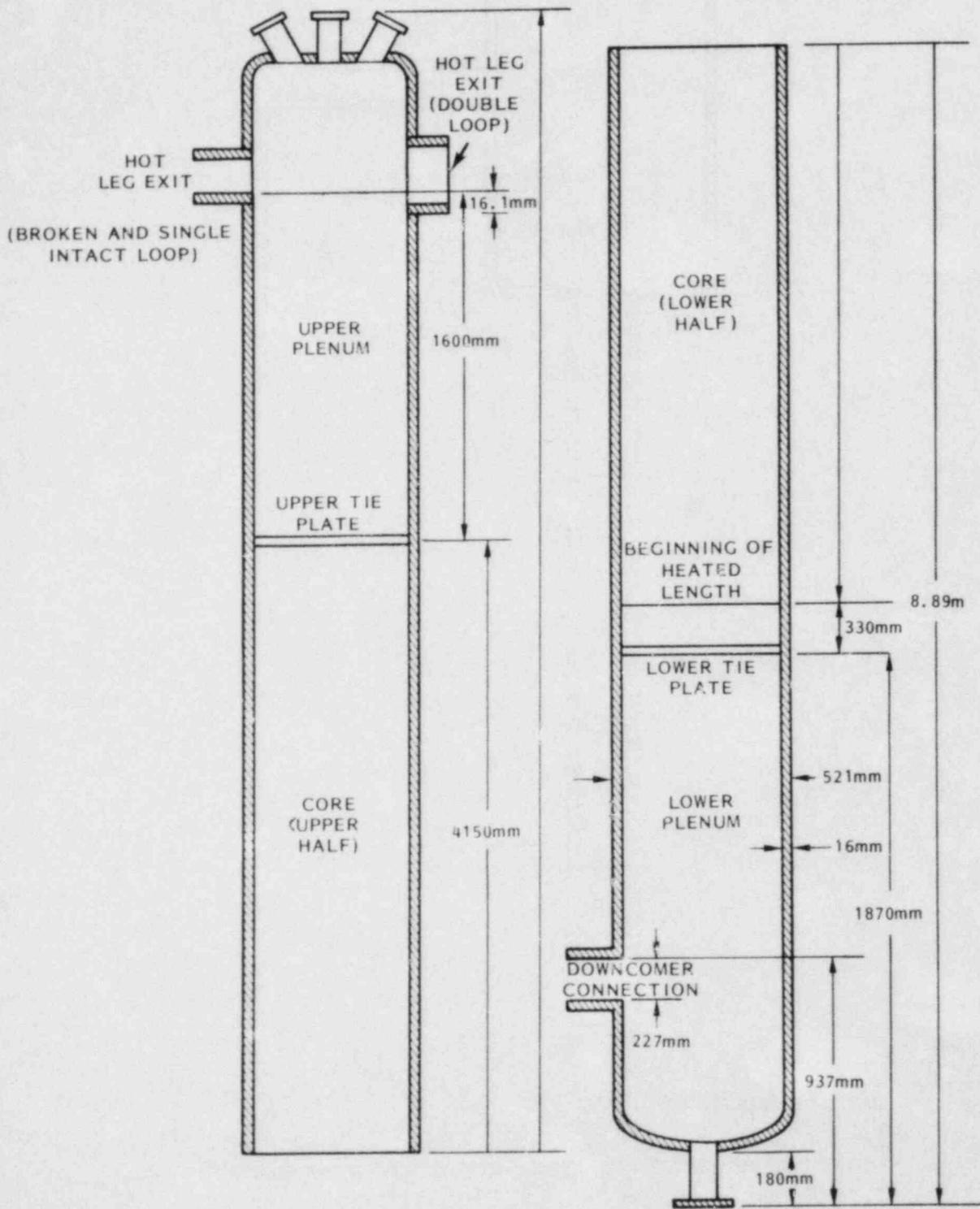


FIGURE 4.3. Vertical Cross Section of PKL Pressure Vessel

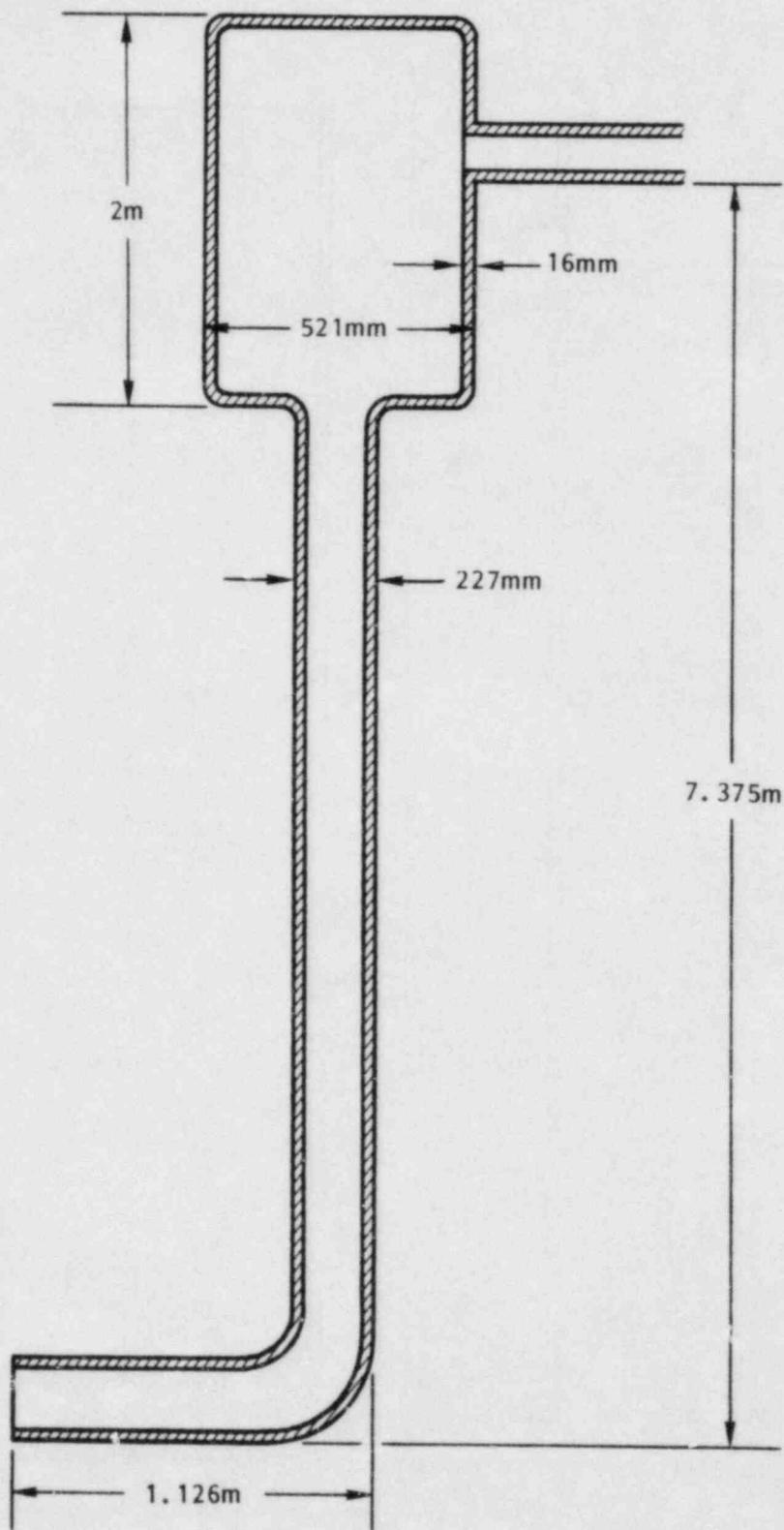


FIGURE 4.4. Vertical Cross Section of PKL External Downcomer

TABLE 4.3. Geometry of PKL Pressure Vessel

Component	Loss Coefficient	Diameter(m)	Volume (m ³)	Length(m)
Pressure Vessel		0.489	0.544	8.89
lower plenum		0.489	0.305 ^(a)	1.87
core		0.04747	0.099	4.48 ^(b)
lower tie plate-	8.68 (upflow)			
	8.90 (downflow)			
upper tie plate-	3.98 (upflow)			
grid spacers	2.99 (downflow)			
upper plenum		0.489	0.186	2.54
upper plenum internals-				
8 tubes, 35 mm x 1.5 mm				
16 tubes, 42.4 x 2.0 mm				

TABLE 4.4. Material Properties for Heater Rods in PKL Test Facility

magnesia -							
Temperature (°C):		100	300	500	1000	1300	
specific heat (W-S/kg K):		1039	1164	1220	1298	1333	
density (kg/m ³):	3150						
NiCr 80/20							
Temperature (°C):		20	100	250	550	800	1000
thermal conductivity (W/m-K):		12.1		16.3	21.4	25.6	33.8
specific heat (W-S/kg-K):			502			502	502
density (kg/m ³):	8300						

(a) Includes volume of horizontal section of downcomer pipe.
 (b) Measured from lower tie plate to upper tie plate.

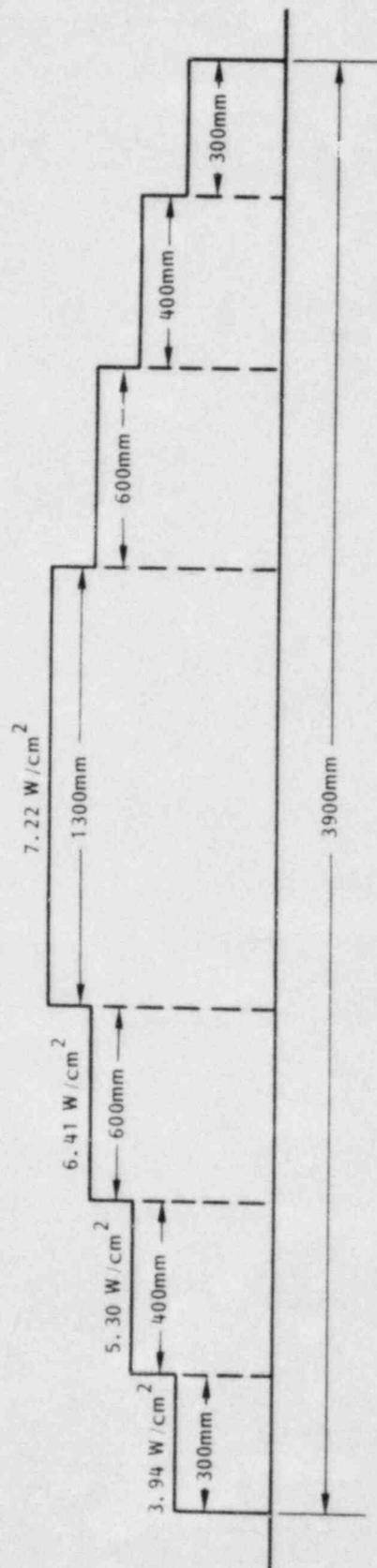


FIGURE 4.5. Seven-Step Axial Power Profile for PKL Test K9

steam and thus heated to the desired initial temperatures. The power to the heater rods was controlled to produce the desired initial radial and axial temperature profile in the bundle.

Test K9 simulated a double-ended cold leg break with only cold leg injection of ECC water for the refill and reflood. Table 4.5 lists the initial conditions for the test, and Table 4.6 gives the test parameters and boundary conditions.

4.2 COBRA/TRAC MODEL OF PKL SYSTEM

The PKL system consists of three loops, but because the system is essentially one-dimensional (even in the pressure vessel) it can be looked upon as having effectively only two loops--an intact side and a broken side. This characteristic makes it possible to greatly reduce the number of computational cells in the COBRA/TRAC model by using only two loops--one modeling both the double and single intact loops, the other the single broken loop. The system can be modeled, then, by seven main components: a pipe for the intact loop hot legs, a steam generator for the intact loop, a pipe for the intact cold legs, a pipe for the broken loop hot leg, a steam generator for the broken loop, a pipe for the broken cold leg upstream of the break assembly, a pipe for the broken cold leg downstream of the break, and a vessel module for the pressure vessel and downcomer. A diagram of the system model is shown in Figure 4.6.

The COBRA/TRAC model of the PKL system can, for convenience, be divided into three categories. These are the loop piping, the steam generators, and the pressure vessel and downcomer. The following subsections describe how the data for the components in each category are converted to COBRA/TRAC input. In some cases, the conversion is relatively self-evident, and is explained only briefly. Some input, however, involves modeling assumptions and conventions and are discussed in greater detail.

4.2.1 Loop Piping Components

The PIPE components shown in Figure 4.6 model the hot legs and cold legs of the intact and broken loops of the system. PIPE2 and PIPE9 each model a

TABLE 4.5. Initial Conditions for Test K9

System Pressure	4.5 bar	
Peak Heater Rod Temperature		
inner zone	571°C	(initial power 3.98 kW/rod)
middle zone	599°C	(initial power 3.93 kW/rod)
outer zone	600°C	(initial power 3.92 kW/rod)
Bundle Initial Power	1327 MW	
Steam Generator Secondary (broken loop)		
initial temperature	267.5°C	
initial pressure	53 bar	
initial water level	7.5 m	
Steam Generator Secondary (single intact loop)		
initial temperature	270.5	
initial pressure	55.5 bar	
initial water level	7.8 m	
Steam Generator Secondary (double loop)		
initial temperature	270.5	
initial pressure	55.5	
initial water level	7.6 m	

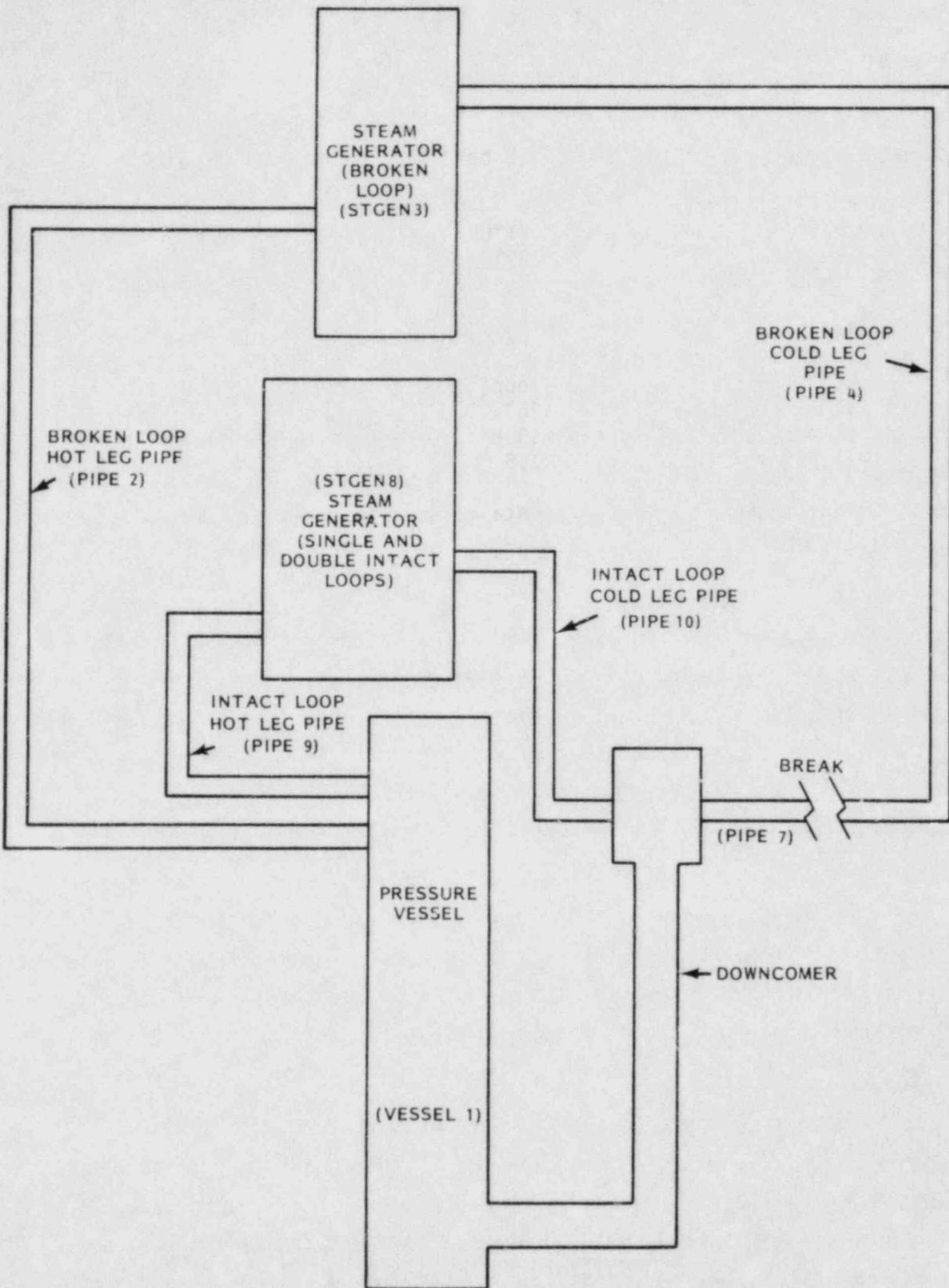


FIGURE 4.6. Diagram of COBRA/TRAC Model of PKL System

hot leg, from the upper plenum exit to the entrance to the steam generator inlet plenum. PIPE10 models the intact loop cold leg from the exit of the steam generator outlet plenum to the downcomer inlet nozzle. PIPE4 models the segment of the cold leg between the break assembly and downcomer inlet nozzle.

The pipes of the intact loop (PIPE9 and PIPE10) have the nominal length of a single loop, but the volume, flow areas and hydraulic diameters of the cells add up to that of the double loop plus the single loop. The COBRA/TRAC input for the PKL model is listed in Table 4.7, and the values for the VOL, FA and HD arrays show the doubling of the loop model. The cell lengths are chosen to accommodate the various bends and angles in the piping, plus area changes and other obstructions. Figures 4.7 and 4.8 show diagrams of the PIPE cells in the intact loop hot and cold legs, respectively. The corresponding DX, GRAV and FRIC array entries are also given.

The pipes of the broken loop (PIPE2, PIPE4 and PIPE7) have geometry array values corresponding to a single loop. The FA array contains values 0.0051 and 0.0123; the HD array contains 0.0808 and 0.125--numbers that correspond exactly with the values given in Table 4.1 for the single loops. The volume array (VOL) contains values that correspond to segments of a single loop. Figures 4.9 and 4.10 show diagrams of the PIPE cells of the broken loop hot and cold legs, with the corresponding DX, GRAV and FRIC array entries.

The fluid in the loop piping (both intact and broken loops) was initialized with stagnant steam at the system operating pressure. That is, the pressure array (P) was set to $4.6(10^5)$ N/m², the velocity array (VM) was set to zero, and the void fraction array (ALP) was set to 1.0. The liquid temperature array, TL, was initialized at the saturation value for the pressure.

The intact loop piping (PIPE9 and PIPE10) connects in a closed loop from the vessel upper plenum, through the steam generator (STGEN8), to the downcomer inlet nozzle. The broken loop piping (PIPE2, PIPE4 and PIPE7) is terminated by BREAK components simulating the containment tank. These components are BREAKs #5 and #6, and supply time-dependent pressure boundary conditions on the primary system corresponding to the break pressure table given in Table 4.6.

TABLE 4.6. Test Parameters and Boundary Conditions

Containment Back Pressure at Break Position -

time (seconds)	0	10	20	30	40	50	75	100	125
pressure (bar)	4.6	4.36	4.18	3.93	3.78	3.71	3.64	3.6	3.60
	150	200	300	350					
	3.50	3.39	3.14	3.0					

Power Per Rod -

time (seconds)	0	1.8	63	75	100	125	150	200	250	300	350
power (kW)	0.	4.0	4.0	3.91	3.75	3.62	3.51	3.34	3.18	3.05	2.87

Cold Leg ECC Injection Rate -

time (seconds)	0	1.5	5.0	10.2	17.6	24.9	26.0
flowrate (kg/sec)	22.0	22.3	20.3	17.5	14.8	13.0	1.9

Temperature of ECC Injection Water -

time (seconds)	1	25	50	75	100	125	150	175	200
temperature (°C)	50.49	50.49	51.47	52.34	52.42	52.73	53.81	53.36	53.52
	225	250	275	300	325	350	375	400	425
	53.52	53.60	53.28	53.36	53.28	53.36	53.13	53.68	53.44
	450	475	500						
	53.52	53.36	53.44						

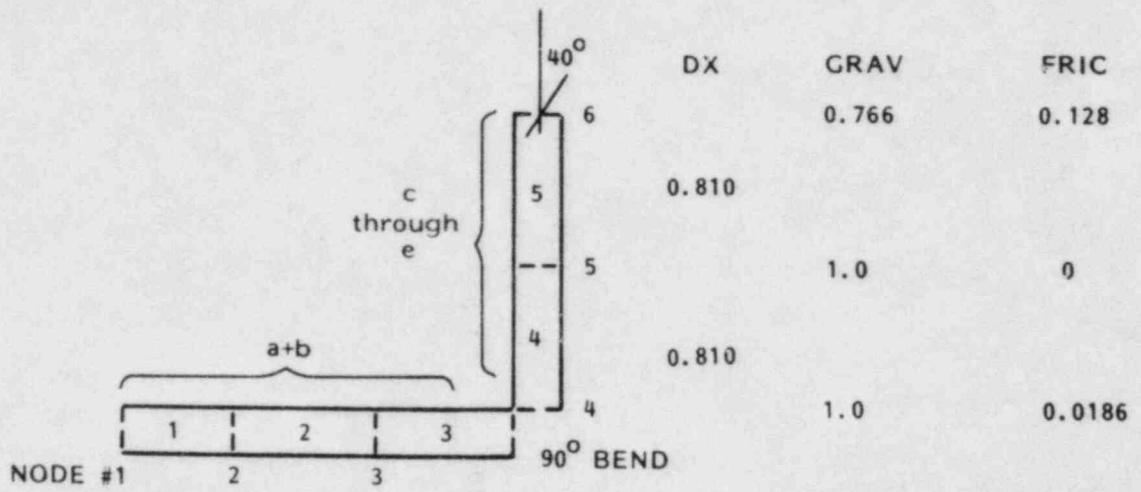


FIGURE 4.7. COBRA/TRAC Model of Intact Loop Hot Leg Piping (PIPE9)

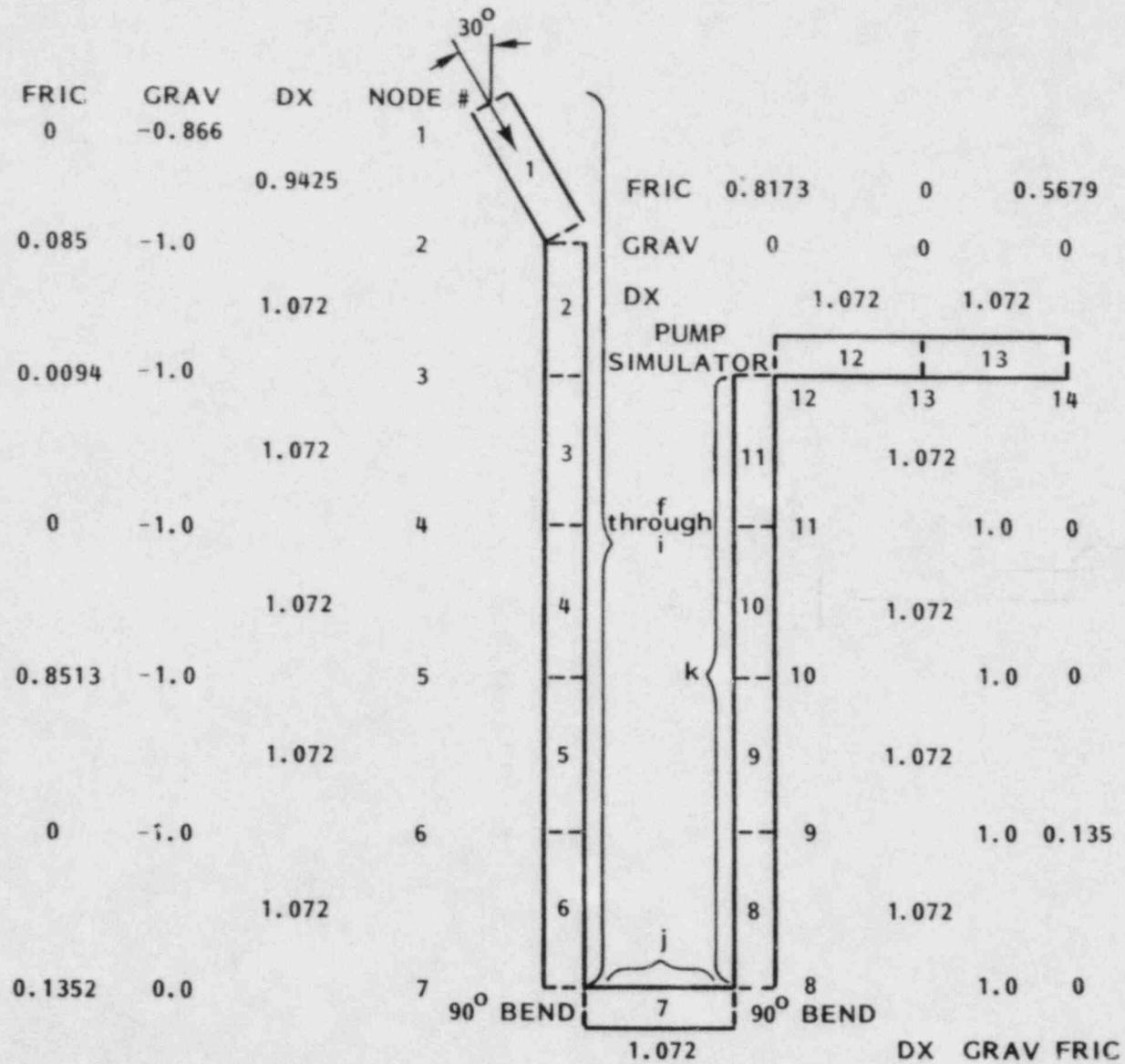


FIGURE 4.8. COBRA/TRAC Model of Intact Loop Cold Leg Piping (PIPE10)

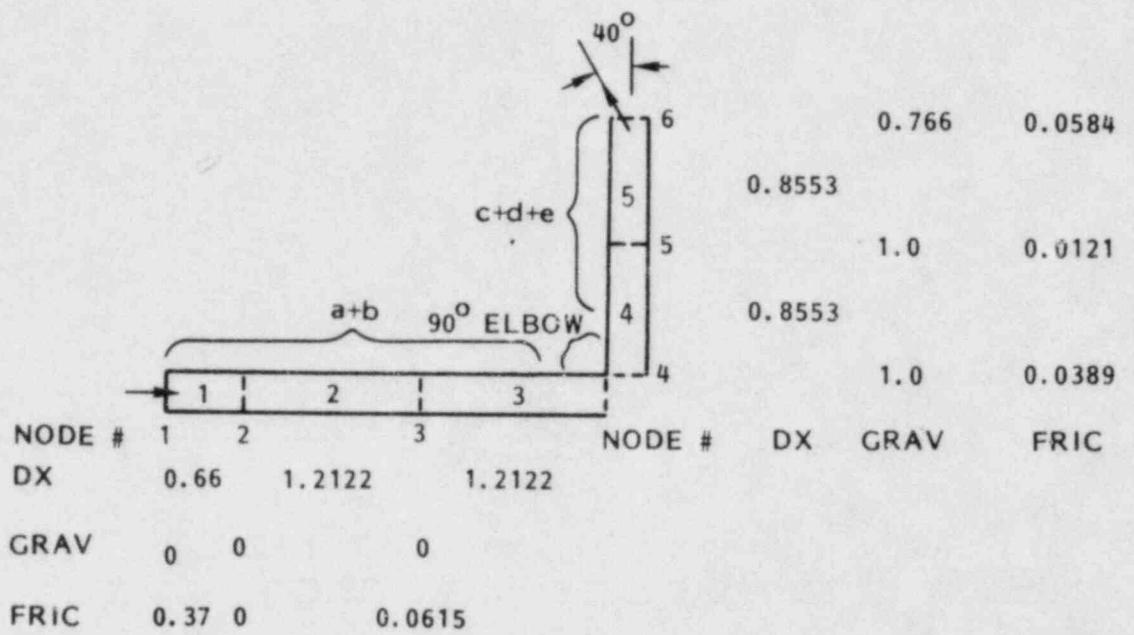


FIGURE 4.9. COBRA/TRAC Model of Broken Loop Hot Leg Piping (PIPE2)

4.21

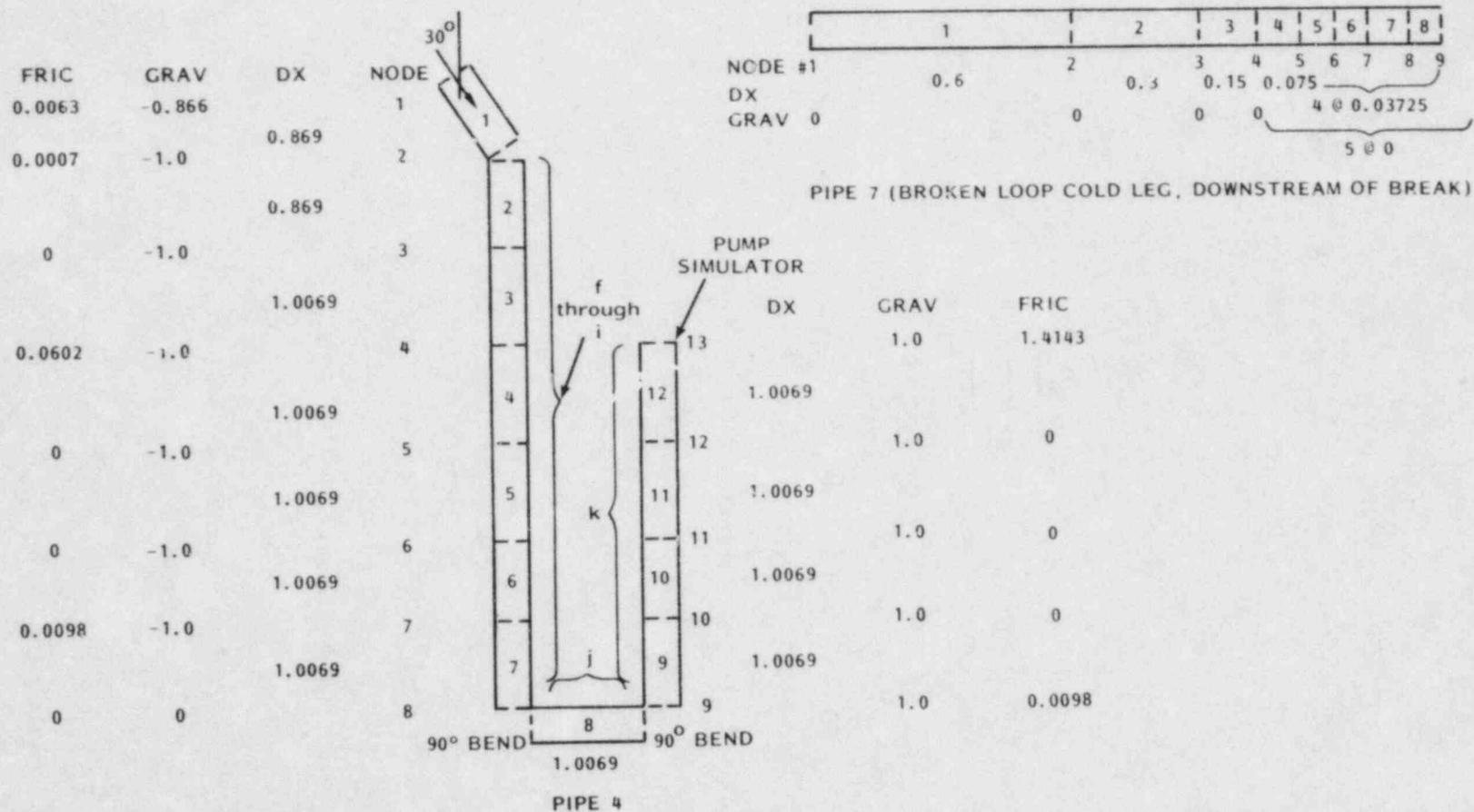


FIGURE 4.10. COBRA/TRAC Model of Broken Loop Cold Leg Piping (PIPE4 and PIPE7)

4.2.2 Steam Generator Components

The three steam generators in the PKL facility are modeled with two steam generator components, STGEN8 and STGEN3. STGEN3 models the steam generator in the broken loop, and STGEN8 models the two steam generators from the double and single intact loops. The steam generators are U-tube type, with 30 tubes each in the single loop components and 60 tubes in the double loop component. So STGEN8 models 90 tubes on the primary side, and STGEN3 models 30 tubes.

The input for the steam generator components is listed in Table 4.7. It is important to note in the geometry input for STGEN3 that the flow area and hydraulic diameter of each cell are that of a single tube, while the volume corresponds to the volume of all 30 tubes. For STGEN8, the flow area is that of one double-loop tube plus one single-loop tube, while the volume corresponds to the volume of the 60 double-loop tubes plus the 30 single-loop tubes on the intact side.

The heat transfer areas entered in arrays WA1 and WA2 for each steam generator component reflect the total area available within each node length in all tubes. For STGEN3, the WA1 array entries are 30 times the inside surface area of one tube. For STGEN8, the WA1 array entries are 60 times the inside surface area of a double-loop tube, plus 30 times the inside surface area of a single-loop tube. The entries in the WA2 arrays are calculated similarly, using the tube outside diameters. (Note that although the primary tubes pass twice through each cell of the secondary side, the entries in WA2 reflect only the surface area of the "up side" of the tubes. The code automatically doubles the WA2 array values to account for the "down side" half of the tubes.)

The steam generators are initialized with stagnant steam on their primary sides. Material temperatures are set to the saturation temperature at the system pressure. The secondary sides were given an initial liquid level and fluid temperature corresponding to the data listed in Table 4.5, and an initial pressure of 5.3MPa.

The steam generator components connect to the hot and cold leg piping on their primary sides. The secondary sides, however, must be supplied with boundary conditions. The exit from a secondary steam dome can be modeled quite simply with a pressure boundary condition that corresponds to the steam dome pressure. Break component #11 is used on the secondary of STGEN3, and Break component #13 is used on STGEN8. The input for both of these components is listed in Table 4.7.

The boundary condition on the steam generator secondary inlet is not so simple as that for the outlet. The COBRA/TRAC steam generator component models the secondary only from the tube sheet inlet to the steam dome. It does not model the downcomer or carryover flow paths. So the inlet boundary condition on the secondary must be a flow boundary that includes the effects of recirculation. Even in cases where the feedwater flow to the secondary is zero, there can be significant flow at the tube sheet inlet, as a result of boiling and recirculation. For this example, the recirculation flow can be neglected, and the secondary side boundary condition at the tube sheet inlet is modeled as a zero flow condition corresponding to actual test conditions. The FILL components #12 and #14 model these boundaries for STGEN3 and STGEN8, respectively.

4.2.3 VESSEL Component

The VESSEL component is used to model the pressure vessel and external downcomer in the PKL system. The channel-splitting capability is utilized to construct the input, which consists of 14 channels in 5 sections. Channels 1 through 8 model the pressure vessel, and 9 through 14 model the downcomer. A diagram of the vessel mesh is shown in Figure 4.11, and the vessel component input is listed in Table 4.7.

The lower plenum is modeled by channel 1 up to the level of the downcomer pipe penetration, and by channel 2 (which connects axially to channel 1) from the upper edge of the downcomer pipe penetration to the lower tie plate. The core from the lower to the upper tie plate is modeled by channels 3 and 4 (connected axially). The upper plenum is modeled by channels 5 and 8 (connected laterally), with channels 6 and 7 modeling the volume directly above the holes in the upper tie plate. The areas for channels 6 and 7 equal the total flow area through the upper tie plate, and they have hydraulic

TABLE 4.7. COBRA/TRAC Input File for PKL Test K9

2

pk1 refill reflood test k-9 cold leg 2 ended break cold leg injection
second pass june 1981

	14	14	1		
	1.0e-03	5	40		
	2	4	9	10	7
	3	8	11	12	13
	14	5	6	1	
pipe		2	2		
	5	0	1	2	6
	0	0			
	.04	.00635			
	295.				
r 1	.66r 2	.825r 2	.810e		
r 1	.003366r 2	.00421r 2	.00996e		
r 4	.0051r 2	.0123e			
r 1	.0306r 1	.0164r 1	0.0r 1	.0186r 1	0.0
r 1	.128e				
r 4	0.r 1	1.r 1	.766e		
r 4	.0808r 2	.125e			
f	4e				
f	0.e				
f	1.e				
f	0.e				
f	421.35e				
f	460000.e				

TABLE 4.7. (continued)

pipe	4	4	4	15	6
	12	0	10		
	0	0			
	.0404	.00635			
	295.				
r 2	.869r10	1.00688e			
r 2	.010689r10	.005135e			
r 1	.017r 2	.0123r10	.0051e		
r 1	.0063r 1	.0007r 1	0.0r 1	.0602r 2	0.0
r 1	.0098r 1	0.0r 1	.0098r 3	0.0r 1	1.4143e
r 1	-.866r 6	-1.r 1	0.0r 5	1.0e	
r 3	.125r10	.0808e			
f	4e				
f	0.0e				
f	1.0e				
f	0.0e				
f	421.35e				
f	460000.e				
pipe	9	9	9	6	6
	5	0	3		
	0	0			
	.074	.00635			
	295.				
r 1	.66r 2	1.2122r 2	.8553e		
r 1	.010032r 1	.018425r 1	.014322r 1	.031069r 1	.302815e
r 2	.015156r 1	.01181r 1	.9449r 1	.036325r 1	.0354e
r 1	.37r 1	0.0r 1	.0615r 1	.0389r 1	.0121
r 1	.0584e				
r 3	0.0r 2	1.0r 1	.766e		
r 4	.609r 2	.942e			
f	4e				
f	0.e				

TABLE 4.7. (continued)

f	1.e				
f	0.0e				
f	421.35e				
f	460000.e				
pipe		10	10		
	13	0	8	4	6
	0	0			
	.0404	.00635			
	295.				
r 2	.9425r11	1.072e			
r 1	.03424r 1	.03243r10	.01625r 1	.01394e	
r 1	.0354r 1	.036325r 1	.03025r10	.01516r 1	.0131e
r 1	0.0r 1	.085r 1	.0094r 1	0.0r 1	.8513
r 1	0.0r 1	.1352r 1	0.0r 1	.1352r 2	0.0
r 1	.8173r 1	0.0r 1	.5679e		
r 1	-.866r 5	-1.0r 1	0.0r 4	1.0r 3	0.0e
r 3	.94248r11	.50884e			
f	4e				
f	0.0e				
f	1.0e				
f	0.0e				
f	421.35e				
f	460000.e				

TABLE 4.7. (continued)

pipe		7	7		
	8		5	7	6
	0	1			
	.0404	.00635			
	295.				
	.6	.3	.15	.075f	.03725e
	.00306	.00153	.000765	.000383f	.00019e
f	.0051e				
f	0.e				
f	0.e				
f	.0808e				
f	4e				
f	0.e				
f	1.e				
f	0.e				
f	421.35e				
f	460000.e				
stgen		3	3		
	20	2	2	10	6
	1	0	0	1	
	.019	.0015			
	9	12	11		
	.435r18	.9617	.435e		
	.0074r18	.008180	.0074e		
r 1	.0123r 1	.017r18	.008506	.017e	
r 1	.0188r19	0.0r 1	.0523e		
r10	1.0	.707	-.707r 9	-1.0e	
r 1	.125r19	.019	.125e		
f	1e				
r 1	.1152r18	1.7674	.1152e		
f	1.0e				
f	0.0e				

TABLE 4.7. (continued)

f	421.35e				
f	421.35e				
f	460000.e				
r 8	.9617	.9867e			
r 9	.0622e				
r 9	.0647	.0614e			
f	0.0e				
f	1.0e				
r 1	.125r 8	.0624	.0592e		
f	1e				
f	1.994e				
r 8	0.0r 1	1.0e			
f	0.0e				
f	540.65e				
f	540.65e				
f	5300000.e				
f	543.65e				
stgen		8	8		
	18	2	6	8	6
	1	0	0	1	
	.0095	.0015			
	8	14	13		
r 1	.1915r16	1.093r 1	.1915e		
r 1	.0107r16	.02789r 1	.0107e		
r 1	.0354r17	.025518r 1	.0354e		
r 1	.2399r17	0.0r 1	.5926e		
r 9	1.0r 1	.707r 1	-.707r 8	1.0e	
	.942r17	.019r 1	.9425e		
f	1e				
r 1	.1152r16	6.0063r 1	.1152e		
f	1.0e				
f	0.0e				

TABLE 4.7. (continued)

f	421.35e				
f	421.35e				
f	460000.e				
f	1.093e				
f	.1795e				
f	.16424e				
f	0.0e				
f	1.0e				
f	.0528e				
f	1e				
f	6.7763e				
r 7	1.0r 1	0.0e			
f	0.0e				
f	543.65e				
f	43.65e				
f	5 ² .0000.e				
f	543.65e				
break		11	11		
	11	0			
	.9867	.0622	1.	540.65	5300000.
break		12	12		
	13	0			
	1.093	.1795	1.	543.65	5550000.
f111		13	13		
	12	1			
	.9867	.0622	1.	0.0	295.
	5300000.				
f111		14	14		
	14	1			
	1.093	.1795	1.	0.0	295.
	5550000.				

TABLE 4.7. (continued)

break		5			
15		1	13	3	
1.00688	.005135		1.	548.	320000.
0.0	460000.		10.	436000.	20.
418000.	30.	393000.		40.	378000.
50.	371000.		75.	364000.	100.
364000.	125	360000.		150.	350000.
200.	339000.		300.	314000.	350.
300000.e					
break		6			
7		1	13	3	
.03725	.0051		1.	548.	320000.
0.0	460000.		10.	436000.	20.
418000.	30.	393000.		40.	378000.
50.	371000.		75.	364000.	100.
364000.	125.	360000.		150.	350000.
200.	339000.		300.	314000.	350.
300000.e					
versel		1			
		1			
	pk) k9 reflood and refill test				
1	28				
350.					
	66.7	1180.08	0.0	.3126	
2	14	2			
	1256.548.24		0		
	2255.048.24	56.98	0		
	948.6924.74		0		
3	79.6484.556.98		0		
	1048.6924.74		0		
4	79.6484.5	56.98	0		
	1148.6924.74		0		
	565.31214.2		0		

TABLE 4.7. (continued)

628.49114.																				0
728.49 114.																				0
865.31214.2																				0
12291.160.4848.69																				0
1348.6924.74																				
14107.224.74																				
3	1																			
4	1																			
3	5																			
1	5	624.127.607		0.0	0.0	0	0	1.	-2	2										
1																				
2	6	715.356.517		0.0	0.0	0	0	1.	1	3										
1																				
3	7	824.127.607		0.0	0.0	0	0	1.	2	-2										
1																				
4	1	14 7.87 30.		0.	1.0	0	0	1.	-1	5										
1																				
5	14	13 7.8724.31		0.	1.0	0	0	1.	4	-1										
1																				
0																				
4	5	2	1																	
1	3	2	18.50																	
1	2																			1
13	9																			13
14	14																			14
2	2	2	18.50																	
2	3																			1
9	10																			13
3	2	8	10.62990																	
3	4																			2
10	11																			9
4	2	8	11.55017																	

TABLE 4.7. (continued)

4	6	7						3
11	12							10
5	5	8	12.32					
5	5							5
6	6							4
7	7							4
8	8							8
12	12							11
5								
42	82							
5	1							
4								
1.1673		2.3346		3.6691		4	1.	
6	1							
1	1	0	12					
7	11							
1.0	2	2						
0.5	2	3						
0.5	4	3						
0.5	5	3						
0.5	8	3						
0.5	1	4						
0.5	2	4						
0.5	5	4						
0.5	6	4						
1.0	1	6						
1.0	1	7						
8	1		2	1				
1	1	1	2		.05	337.	1.	440.
1	1		12					
1								

TABLE 4.7. (continued)

74.016	298.8	93.976	672.8	107.756	831.2	127.441	1020.2				
139.842	1056.2	164.842	1119.2	189.645	1117.4	212.086	1041.8				
221.929	898.6	228.818	858.2	235.708	789.8	251.52	298.8				
9	1										
1 hrod	.42323	0.0	4	2	2						
3	1.1102	0.	1	2.0079	1.	2	1.0541	0.	2	2.0394	0.
10	2										
1	12	196.67									
200.	.2385	11.4	400.	.2625	9.5						
600.	.2800	7.8	800.	.2900	5.6						
1000.	.3000	4.3	1200.	.3050	3.5						
1400.	.3080	3.0	1600.	.3100	2.7						
1800.	.311	2.5	2000.	.3125	2.5						
2200.	.3150	2.5	2400.	.3170	2.5						
2	12	518.22									
200.	.1199	7.77	400.	.1199	8.94						
600.	.1199	10.06	800.	.1199	11.15						
1000.	.1199	12.24	1200.	.1199	13.32						
1400.	.1199	14.40	1600.	.1199	15.46						
1800.	.1199	16.52	2000.	.1199	17.57						
2200.	.1199	18.62	2400.	.1199	19.68						
11	1	11	0								
1	19	153.542									
74.016	0.0	85.088	0.0	88.070	0.0	88.071	.6497				
99.881	.6497	99.882	.8725	115.62	.8726	115.63	1.0554				
139.251	1.0554	139.252	1.1888	190.432	1.1888	190.433	1.0554				
214.054	1.0554	214.055	.8726	229.802	.8726	229.803	.6497				
241.613	.6497	241.614	0.0	251.456	0.0						
0.0	0.0	1.8	1.0	63.	1.0	75.	.9775				
100.	.9375	125.	.9050	150.	.8775	200.	.8350				
250.	.7950	300.	.7625	350.	.7175						

TABLE 4.7. (continued)

13	14	2	1	1						
9										
0.		0.	.001		1.0	1.5	1.0136	5.0	.9227 10.2	.7955
17.6		.6727	24.9		.5909	26.	.0864	350.	.0864	
1	1	3			0.0					
5	1	3			0.0					
5	9	3			0.0					
6	9	3			0.0					
7	9	3			0.0					
8	1	3			0.0					
8	9	3			0.0					
12	5	4	1		48.51		90.			
12	9	3			0.0					
13	1	3			0.0					
13	2	3			0.0					
14	1	3			0.0					
14	2	3			0.0					
14	3	3			0.0					
4	1	2								
14	5	0	0	0	0	0	1	1		
4										
5	5	1	1							
8	5	3	3							
12	6	0	5							
12	4	0	4							
0										
6	350	0								
1	2	3	4	7	9					
	.0001			.1	400.		1.		7000.	
	10.			1.	1800.		3700.			
	-.001									

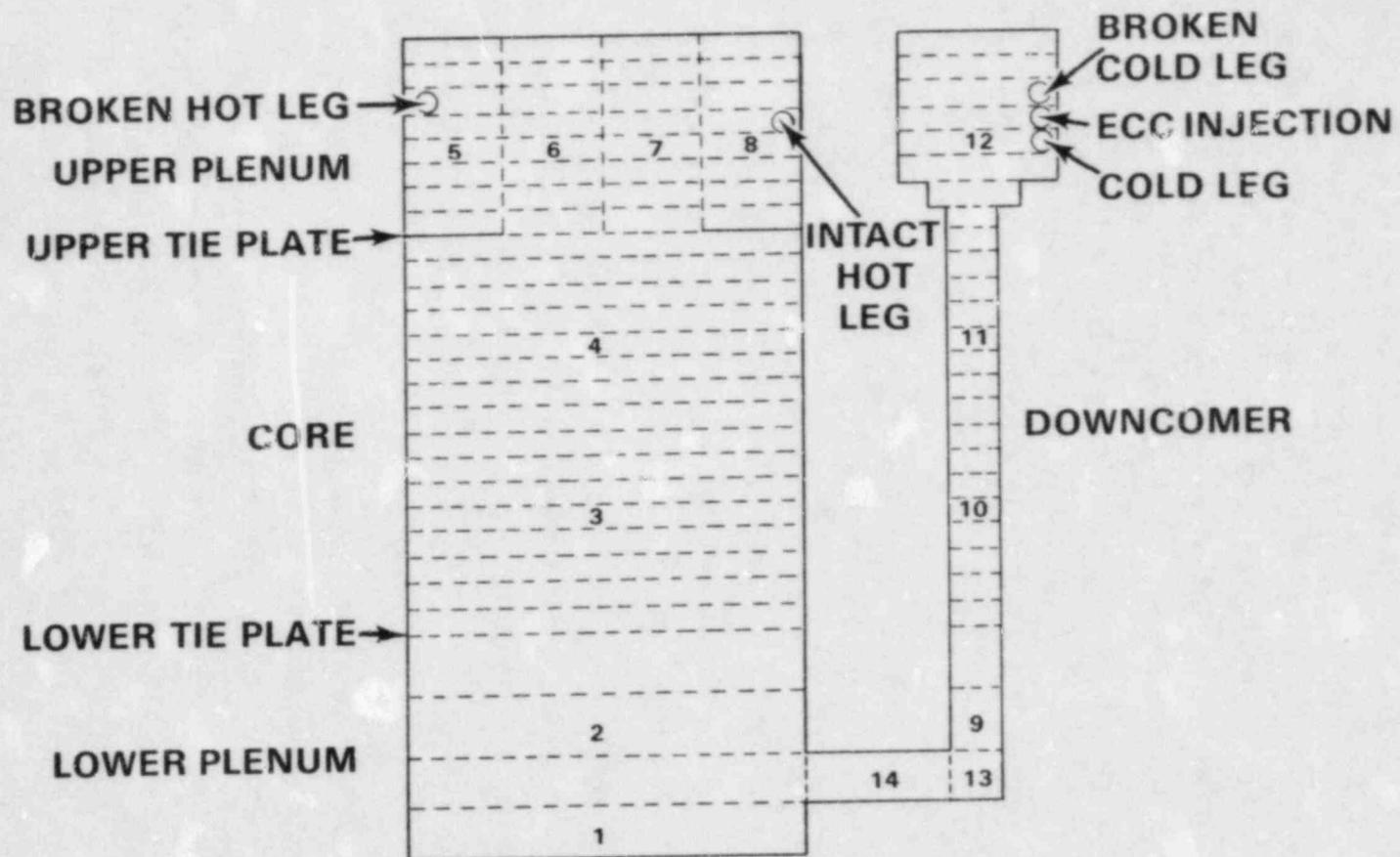


FIGURE 4.11. COBRA/TRAC Model of PKL Pressure Vessel and Downcomer

diameters equal to the hydraulic diameter of a single tie plate hole. This modeling convention is required to model effects such as counter-current flow limiting at the upper tie plate. Only channels 6 and 7 have axial connections to the core (channel 4 below); channels 5 and 8 have zero-flow boundaries at their inlet, modeling the solid portions of the lower tie plate. Channels 5, 6, 7 and 8 have gap connections that permit transverse flow in the upper plenum. The junction connection with the intact loop hot leg (PIPE9) is located in channel 8, and the junction connection with the broken loop hot leg is in channel 5. The flow boundaries at the top of the upper plenum channels are set to zero, so the only flow paths out of the vessel are the hot leg junctions.

The horizontal segment of the downcomer pipe is modeled by channels 13 and 14. Local flow boundary conditions (supplied in group 13) are used to zero the axial flow in channel 14 and permit only transverse flow into or out of channel 14 through gap connections to channel 1 and to channel 13. Flow boundary conditions are also imposed on channel 13 to permit axial flow only at the top of the channel.

The downcomer inlet structure is modeled with channel 12. A zero flow boundary condition is used at the top of channel 12. The junction connections to the intact and broken loop cold leg pipes (PIPE10 and PIPE7, respectively) are located in channel 12. There is also a special flow source boundary condition in channel 12 to model the ECCS injection into the downcomer.

The material structure of the pressure vessel and downcomer can be modeled with unheated conductors (see input group 8), but since the system was preheated with steam to a stable initial temperature, thermal inertia effects were assumed to be small. Only the segment of the pressure vessel enclosing the core was modeled for this simulation. This required two unheated conductors: one connected to channel 3, and one connected to channel 4. The rods in the core are modeled by a single average rod with the average power generation rate of the core. It is given an initial temperature profile calculated from the rod temperatures reported in the test data at time zero.

The input data for the VESSEL also includes loss coefficients to model the grid spacers and tie plates in the core. The graphics input is set up to

ask for data on the VESSEL, the broken loop hot leg (PIPE2), steam generator (STGEN3), and cold leg piping (PIPE4 and PIPE7), and the intact loop hot leg. Table 4.7 lists the vessel input. The correspondence between the PKL system description for test K9 (see Tables 4.2 and 4.5) and the COBRA/TRAC input is fairly obvious.

REFERENCES

1. D. R. Liles, and others, "TRAC-PD2: An Advanced Best-Estimate Computer Program for Pressurized Water Reactor Loss-of-Coolant Accident Analysis," NUREG/CR-2054, April 1981.
2. D. R. Liles, and others, "TRAC-P1A: An Advanced Best-Estimate Computer Program for PWR LOCA Analysis," USNRC Report NUREG/CR-0065, Los Alamos National Laboratory, 1979.
3. TRAC-P1A Development Assessment," NUREG/CR-1059, LA-8056-MS, Informal Report F-4, Energy Division, Los Alamos National Laboratory, Los Alamos, New Mexico, 1979.
4. D. J. Olson. "Experimental Data Report for Single- and Two-Phase Steady-State Tests of the 1 Loop MOD-1 Semiscale System Pump," Aerojet Nuclear Company, Report ANCR-1150, May 1974.
5. G. G. Loomis. "Intact Loop Pump Performance During the Semiscale MOD-1 Isothermal Test Series," Aerojet Nuclear Company, Report ANCR-1240, October 1974.
6. D. J. Olson. "Single- and Two-Phase Performance Characteristics of the MOD-1 Semiscale Pump Under Steady-State and Transient Fluid Conditions," Aerojet Nuclear Company, Report ANCR-1165, October 1974.
7. B. Brand, and others, "Refill and Reflood Experiment in a Simulated PWR Primary System (PKL)," Specification, 1979, OECD-CSN1, LOCA Standard Problem No. 10, Kraftwerk Union Report R513.
8. P. E. MacDonald, and others, "MATPRO-Version 9: A Handbook of Materials Properties for Use in the Analysis of Light Water Reactor Fuel Rod Behavior," Idaho National Engineering Laboratories, TREE-NUREG-1005, 1976.

DISTRIBUTION

No. of
Copies

No. of
Copies

OFFSITE

ONSITE

335 U.S. Nuclear Regulatory
Commission
Division of Technical
Information and Document
Control
7920 Norfolk Avenue
Bethesda, MD 20014

50 Pacific Northwest Laboratory

MJ Thurgood (43)
Publishing Coordination (2)
Technical Information ED (5)

5 James Han
U.S. Nuclear Regulatory
Commission
7915 Eastern Ave.
M/S 1130-SS
Silver Spring, MD 20910

NRC FORM 335 <small>(11-81)</small> U.S. NUCLEAR REGULATORY COMMISSION BIBLIOGRAPHIC DATA SHEET		1. REPORT NUMBER (Assigned by DDC) NUREG/CR-3046, Vol. 3 PNL-4385	
4. TITLE AND SUBTITLE (Add Volume No., if appropriate) COBRA/TRAC - A Thermal Hydraulics Code for Transient Analysis of Nuclear Reactor Vessels and Primary Coolant Systems - Volume 3: Users' Manual		2. (Leave blank)	
7. AUTHOR(S) M. J. Thurgood, J. M. Cuta, A. S. Koontz, J. M. Kelly		3. RECIPIENT'S ACCESSION NO.	
9. PERFORMING ORGANIZATION NAME AND MAILING ADDRESS (Include Zip Code) Pacific Northwest Laboratory P.O. Box 999 Richland, WA 99352		5. DATE REPORT COMPLETED MONTH YEAR November 1982	
12. SPONSORING ORGANIZATION NAME AND MAILING ADDRESS (Include Zip Code) Division of Accident Evaluation Office of Nuclear Regulatory Research U.S. Nuclear Regulatory Commission Washington, DC 20555		DATE REPORT ISSUED MONTH YEAR March 1983	
13. TYPE OF REPORT Computer Code Manual		PERIOD COVERED (Inclusive dates)	
15. SUPPLEMENTARY NOTES		10. PROJECT/TASK/WORK UNIT NO.	
16. ABSTRACT (200 words or less) The COBRA/TRAC computer program has been developed to predict the thermal-hydraulic response of nuclear reactor primary coolant systems to small and large break loss-of-coolant accidents and other anticipated transients. The code solves the compressible three-dimensional, two-fluid, three-field equations for two-phase flow in the reactor vessel. The three fields are the vapor field, the continuous liquid field, and the liquid drop field. A five-equation drift flux model is used to model fluid flow in the primary system piping, pressurizer, pumps, and accumulators. The heat generation rate of the core is specified by input and no reactor kinetics calculations are included in the solution. This volume is the Users' Manual. It contains the input instructions for COBRA/TRAC and its auxiliary programs, SPECSET and GRAFIX. It also includes a users' guide to the code and is intended to aid the new user in becoming familiar with the capabilities and modeling conventions of the code.		11. FIN NO. FIN B2391	
17. KEY WORDS AND DOCUMENT ANALYSIS		14. (Leave blank)	
17a. DESCRIPTORS		17b. IDENTIFIERS: OPEN-ENDED TERMS	
18. AVAILABILITY STATEMENT Unlimited		19. SECURITY CLASS (This report) Unclassified	
20. SECURITY CLASS (This page) Unclassified		21. NO. OF PAGES 22. PRICE \$	

UNITED STATES
NUCLEAR REGULATORY COMMISSION
WASHINGTON, D.C. 20555

OFFICIAL BUSINESS
PENALTY FOR PRIVATE USE, \$300

FOURTH CLASS MAIL
POSTAGE & FEES PAID
USNRC
WASH D C
PERMIT No. 567

120555078877 1 ANR4
US NRC
ADM DIV OF TIDC
POLICY & PUB MGT BR-PDR NUREG
W-501
WASHINGTON DC 20555