DEVELOPMENT OF REACTOR SYSTEM

CODE ANALYSIS WORKBOOK:

RELAP5/ASYST

DEVELOPMENT OF REACTOR SYSTEM CODE ANALYSIS WORKBOOK: RELAP5/ASYST

By SOLOMON BELLO

A Thesis Submitted to the School of Graduate Studies in Partial Fulfillment of the Requirements for the Degree Master of Applied Science

August 2024

McMaster University

MASTER OF APPLIED SCIENCE (2024)

Hamilton, Ontario, CANADA (Department of Engineering Physics)

TITLE: Development of Reactor System Code Analysis Workbook: RELAP5/ASYST

AUTHOR: Solomon BELLO

B.Eng (Electrical/Electronics Engineering), Nigeria

M.Phil (Nuclear Science and Technology), Ghana

SUPERVISOR: Dr. David NOVOG

NUMBER OF PAGES: xxi, 228

Lay Abstract

This project aims to develop a initial training workbook for the use of RELAP5 and ASYST system codes in safety analysis. RELAP and ASYST are powerful tools used for simulating the behavior of complex thermal-hydraulic systems, particularly in nuclear power plants. But despite its widespread use in academia and industry, these codes can be challenging for new users due to their complex nature and the multitude of input options. The workbook will serve as a guide for users as it provides step-by-step instructions on problem modeling, components descriptions and nodalizations, input file development, and results analysis. Examples and analytical validation to enhance new users' understanding of the relationship of the simulations to the theoretical solutions are also included. The development of the workbook stemmed from the recognition of the need for accessible and structured educational resources for RELAP5 and ASYST.

Abstract

In the operation of nuclear reactor, proficient utilization of system code analysis tools in predicting reactor behaviour is critical for ensuring safe operation and performance optimization. However, the complexity inherent in these tools often poses a significant challenge for new users, hindering their ability to navigate and utilize these systems effectively.

This workbook aims to address this challenge by presenting a comprehensive training package specifically designed for new users delving into system code analysis. It provides consistent examples, analytical solutions and results for two major nuclear reactor system codes (RELAP5 and ASYST), different models, and analysis techniques into a single, user-friendly platform. The overarching objective is to provide prospective nuclear engineers, researchers, and operators with a versatile and efficient tool that streamlines the process of initial training for reactor system analysis.

This work provides the much needed structured and accessible training methodologies to facilitate skill acquisition among new users in nuclear power engineering. The methodology employed involves description of basic required components, theoretical background, and simulation results. Essential components of this training package include the procedural steps, input file development, practical

hands-on exercises and simulated flow cases of reactor components with analytical validation. Each example provides detailed inputs for modelling steady and transient conditions for both single, multi-channels, connecting components and systems.

The results of simulations were validated by presenting respective analytical calculations which show good agreement. Sensitivity studies are also included so new users can begin gaining intuition on the impact of some of the inputs of simulation results. Finally conclusions and recommendations for future work are also included in the final chapter.

To my family

Acknowledgement

I would like to take this opportunity to express my sincere gratitude to those who have offered me their invaluable support without which it is impossible for me to finish this thesis.

First, I am eternally grateful to my very supportive and dedicated supervisor, Professor David Novog who have been a strong pillar providing unwavering support, impartful guidance and great mentorship from the onset of this research.

To my colleagues and friends particularly Dr. Garik Patterson, Ramazan Bayram Akir, Kevin Sebastian Ruiz, thank you for the valuable discussions, support, and contributions.

I also extend my gratitude to the McMaster Nuclear Reactor (MNR) staff and colleagues, including managers, operators, and tour guides for the cooperation, discussions and excellent working relationship.

Many thanks to the staff and faculty members of Engineering Physics Department at McMaster University, for their help and valuable instructions in various courses.

Last but not the least, my family. To Mum, thank you for the fervent prayers and care, no matter the distance that separates us,you will always remain in my heart. To my ever beautiful wife and amazing kids, thank you all for the genuine love and unconditional support and patience, you have my deepest love.

Table of Contents

List of Figures

List of Tables

Nomenclatures and Abbreviations

Nomenclature

k	Thermal conductivity
h	Convection coefficient
L	Length
V	Velocity
T	Temperature
P	Pressure
D	Diameter
D_h	Hydraulic diameter
ν	Kinematic viscosity
f	Friction loss coefficient
Re	Reynolds number
Nu	Nusselt number
Pr	Prandtl number
T	Temperature
t	Time
τ	stress tensor

g	Acceleration due to gravity
σ	Stefan-Boltzman coefficient
Q	Heat transfer rate
ρ	Density, reactivity in reactor kinetics
A	Cross-sectional area,
G	Mass flux
f	Friction factor

List of Abbreviations

AECL	Atomic Energy of Canada Limited
AGR	Advanced Gas-Cooled Reactor
ANL	Argonne National Laboratory
ANS	American Nuclear Society
AMR	Adaptive Mesh Refinement
ASYST	Adaptive System Thermal hydraulics
ATHLET	Analysis of Thermal hydraulics and Transients
BEPU	Best Estimate Plus Uncertainty
BWR	Boiling Water Reactors
CATHENA	Canadian Algorithm for Thermal Hydraulic Network Analysis
CANDU	Canadian Deuterium Uranium Reactor
CNSC	Canadian Nuclear Safety Commission
CHF	Critical Heat Flux
CONDOR	Core Optimization by Differential Evolution and Reactor analysis

COP	Conference of Parties
CRP	Coordinated Research Projects
FDM	Finite difference method
FLASH-1	Fast Linear Analysis and Simulation of Hydraulics
FORTRAN	Formula Translation
GATUNA	Genetic Algorithm for Time-Dependent Optimization and Nonlinear Analysis
GCR	Gas-Cooled Reactors
HX	Heat Exchanger
HPC	High-Performance Computerssure Core Spray System
IAEA	International Atomic Energy Agency
IEA	International Energy Agency
INL	Idaho National Laboratory
IRUG	International RELAP Users Group
ISP	International Standard Problem
ISS	Innovative Systems Software
LOCA	Large Break Loss of Coolant Accident
LGR	Liquid Graphite Reactors
LMFBR	Liquid Metal Fast Breeder Reactor
MSR	Molten Salt Reactors
MELCOR	Methods for Estimation of Leakages and Consequences of Releases
MCNP	Monte Carlo N-Particle Transport Code
NRC	Nuclear Regulatory Commission
PHWR	Pressurized Heavy Water Reactors

PSA	Probabilistic Safety Assessment
PWR	Pressurized Water Reactors
RBMK	Reactor Bolshoy Moshchnosti Kanalniy
REHLI/GIF	Reactor-to-Environment and Hydrothermal Loop/Gravity
RELAP	Reactor Excursion and Leak Analysis Program
RETRAN	Reactor Transient Analysis
SCALE	Standardized Computer Analyses for Licensing Evaluation
SDTP	Science Data Transfer Protocol
SMR	Small Modular Reactors
SNGLJUN	Single Junction
TDJ	Time-Dependent Junction
TDV	Time-Dependent Volume
TRAC	Transient Reactor Analysis Code
TRACE	Transient Reactor Analysis Code
USNRC	United States Nuclear Regulatory Commission

Chapter 1

Introduction

To address the increasing energy demand and cutting down on carbon emission, 22 countries signed a declaration supporting tripling nuclear energy capacity by 2050 at the 28th Conference of the Parties to the original 1992 United Nations Framework Convention on Climate Change (COP28, 2024). Presidents and/or senior officials from several country including Bulgaria Canada, the Czech Republic, Finland, France, Ghana, Hungary, Japan, South Korea, Moldova, Mongolia, Morocco, the Netherlands, Poland, Romania, Slovakia, Slovenia, Sweden, Ukraine, the United Arab Emirates, the UK and the USA signed the document [1]. The Paris Agreement in 2015 promised to meet the increasing demand using clean and sustainable sources in conjunction with the Net Zero Emission commitment [2].

To meet this demand, nuclear power generation technologies and assessment requires constant improvement. Many countries such as the USA, Canada, Russia, and Japan played a pioneering role in the development of nuclear systems and the computer codes to support their development [3]. However, accidents at Three Mile Island, Chernobyl, and Fukushima have proved that continuous improvement of safety and in the tools to model such events is still needed [5][6][7][67].

These accidents led to the development of safety principles to ensure safe operation, and as part of this development the application and validation of systems codes is important to simulate the plant response to upset conditions [8]. The application of the safety analysis methods becomes a crucial part of both the plant design and operation, providing safer yet more efficient solutions to remain competitive in the market all while complying with regulatory rules [9].

The accurate analysis of system behavior and response in the realm of nuclear engineering and reactor safety is vital in ensuring the safe and efficient operation of nuclear power plants [10]. However, since nuclear accident simulations can involve very extreme conditions, complex geometries, and control system inteventions, sophisticated software involving a mixture of physical laws and empirical relationships is needed to accurately simulate such events [11].

Reactor system codes are computational tools used in nuclear engineering to simulate the behavior of nuclear reactors cooling under normal and adverse conditions and include models capable of simulating the primary heat transport system, secondary side and other supporting hydraulic systems, safety system response, and control logic under various conditions [12]. These codes play a crucial role in reactor design, safety analysis, operation, and optimization and utilize mathematical models to simulate the complex physical phenomena occurring within a reactor system, including thermal-hydraulics, neutronics, and structural mechanics. System analysis codes employ empirical correlations to close the physical formulation which often have more unknown variables equations available for the conservation of mass momentum and energy [13]. Correlations are also used to describe some physical phenomena which are too complex to model on first-principles (e.g., CHF) [10].

Reactor system codes find applications across various stages of a reactor's lifecycle, including design, licensing, operation, and decommissioning [12].

The motivation behind this research stems from the limitations and complexities inherent in the current state of reactor analysis tools and how new users can begin to understand such codes. These tools typically requiring extensive expertise and time-consuming efforts to navigate and integrate information across multiple platforms. Furthermore, the need for a more accessible and unified training framework becomes evident with the increasing complexity of reactor designs, coupled with the demand for quality assurance in safety evaluations and regulatory compliance [12][15]. While formal training courses exist, there is still a need for less formal and self-led tutotrials to get new users up and running in their work.

This thesis provides a Reactor System Code Analysis Workbook that aims to overcome these challenges by offering a cohesive platform of sample problems of gradually increasing complexity. It provides the specific inputs for two of the popular thermal hydraulics system code and models, allowing users to perform multifaceted analyses covering different aspects of reactor behavior, transient scenarios, and safety assessments.

The anticipated outcomes of this research endeavor include the creation of a versatile, user-centric, and robust platform for reactor system analysis. This workbook will not only simplify the analysis process for users but also foster better decision-making, improved safety assessments, and optimization of reactor designs. By facilitating a more holistic understanding of reactor behavior through comprehensive analysis, the Reactor System Code Workbook seeks to advance the field of nuclear engineering and contribute to safer and more efficient nuclear power generation.

1.1 Overview of RELAP5 and ASYST System Codes

1.1.1 The Reactor Excursion and Leak Analysis Program (RELAP 5)

RELAP5 (Reactor Excursion and Leak Analysis Program) is a computer code widely used in the nuclear engineering field for the simulation and analysis of nuclear reactor systems which solves the overall steady state and transient behaviour of the systems through a finite-volume approximation to the conservation of mass, momentum and energy [12]. It also includes capability to analyze the control system, point kinetic neutronic behaviour, and general heat structures (i.e., heat transfer to solid bodies). Originally developed by the United States Nuclear Regulatory Commision (NRC) is has the capability to analyze light water and heavy water nuclear reactors, including pressurized water reactors (PWRs), boiling water reactors (BWRs), and advanced reactor designs. Over its history it has undergone several modifications and the version used in this thesis is independently maintained and developed by ISS based in Idaho Falls, USA [17].

Although developed for light water reactors (PWR/BWR), the code is a flexible tool for large reactor nuclear system simulations, SMRs, and also for small scale test facilities [17]. It provides advanced thermal hydraulic modeling capabilities to simulate the behavior of coolant flow, heat transfer, phase changes and transient conditions such as reactor startups, shutdowns, power maneuvers, loss-of-coolant accidents (LOCAs), and other abnormal operating scenarios within reactor system [12]. In addition, RELAP has the capability to model complex phenomena such as two-phase flow, thermal stratification, boiling, condensation, and heat exchange in various reactor components, including reactor cores, steam generators, pumps, and heat exchangers. Users can model entire nuclear reactor systems, including primary and secondary coolant loops, reactor cores, steam turbines, and balance of plant components.

Computational method in RELALP employ advanced numerical analysis to solve the governing equations of fluid dynamics and heat transfer, including finite difference methods (FDM) for discretizing the solid domain and implicit time integration schemes for solving transient problems. It uses a two-fluid, six-equation formulation to model two-phase flow and phase change phenomena accurately (i.e. mass-momentum-energy are conserved for each of the fluid phases with additional equations to handle phase-to-phase exchange)[12].

1.1.2 RELAP for CANDU Reactor

The design peculiarities of CANDU type reactors, especially the horizontal fuel channel and the moderator separated from the coolant do not allow a straightforward application of the advanced core degradation models. But the analysis of design basis accidents and the modelling of experiments in specially designed facilities can be successfully perform [18, 19]. The major thermal hydraulic phenomena for the key CANDU events and the modeling limitation of the RELAP5 has been improved and developed as RELAP/CANDU for CANDU application in the (1) Critical Flow Model: This was adapted to better represent the flow characteristics in CANDU reactor systems (2) Nuclear Kinetics Model: The decay heat model was updated to include the ANS94-4 model, which accounts for the Pu241 contribution, making it more suitable for CANDU reactors (3) Critical Heat Flux Model: This was improved to better represent the heat transfer in CANDU's horizontal fuel channels (4) Reactor

Core Control Model: A digital sampling model was implemented in the control function to enhance reactor core control simulations (5) Valve and Spray Model: The motor-operated valve model was improved, and a pressurizer spray model was implemented (6) Improvement of Horizontal Flow Regime Map: This was specifically enhanced to model the unique horizontal fuel channels in CANDU reactors(7) Heat Transfer Model in Horizontal Channel: A CANDU-specific 37-element fuel bundle heat transfer model was implemented to better represent heat transfer in the horizontal fuel channels [20][21]. Additionally, the channel model was improved, which enhanced the applicablity of the code for CANDU fuel channel analysis where flow stratification phenomena become important [20]. These adaptations allow for more accurate modeling of CANDU reactor behavior, particularly in accident scenarios and safety analyses [19]. This code is based on the REIAP5/MOD3.2.2 gamma version, presented in FORTRAN90 language [20] which offers more advanced programming features compared to earlier FORTRAN versions, potentially leading to improved code organization and readability.

1.1.3 Adaptive SYStem Thermal hydraulics (ASYST)

Similar to the RELAP5, Adaptive SYStem Thermal hydraulics (ASYST) is a computer code developed for the simulation and analysis of thermal hydraulic behavior in nuclear reactor systems. ASYST is a new code that combines the capabilities of RELAP for design basis transients with the tools SCDAPSIM and SAMPSON for beyond design basis analysis[22][23]. The ASYST-THA thermal hydraulic module has replaced the previously used US NRC-developed RELAP5 code in RELAP/SCDAPSIM/MOD3.x and the THA code in SAMPSON. It introduces new system-level hydrodynamic options, including advanced multi-dimensional and multi-fluid models initially developed by the Institute of Safety and Security (ISS) and the Institute of Applied Energy (IAE). ASYST provides reactor-specific modeling options that include modules for (a) core and fuel assembly structures, (b) late-phase debris and melt relocation, (c) containment systems including melt spreading and interactions between molten core and concrete, and (d) fission product release and transport[22]. The module for core/fuel assembly behavior is based on derivatives of SCDAPSIM/MOD3.x models and correlations. The late-phase debris/melt relocation module combines 2D models from SCDAPSIM/MOD3.x with 3D models from SAMPSON MCRA, DCA, and DSA.

The fission product release and transport module integrates models from both SCDAPSIM/MOD3.x and SAMPSON. The core-concrete interaction module combines a SCDAPSIM-based porous media model with SAMPSON's Debris-Concrete Interaction (DCRA) models and correlations. ASYST-THA describes the thermal hydraulic behavior of the reactor vessel, coolant system, and containment, incorporating SAMPSON's modules for hydrogen combustion (HYNA), hydrogen detonation (DDOC), and steam explosions (VESUVIUS). The development of ASYST is supported by funding and technical contributions from the international SCDAP Development and Training Program (SDTP). This tool is designed to offer advanced modeling capabilities for analyzing both transient and steady-state conditions across various reactor types, including pressurized water reactors (PWRs), boiling water reactors (BWRs), heavy water moderated reactors, and advanced reactor designs.

The development of ASYST is supported by the international SCDAP Development and Training Program (SDTP)[23][24] and aims to offer advanced modeling capabilities for both transient and steady-state analyses across various reactor types, including PWRs, BWRs, heavy water moderated reactors, and advanced reactor designs. It is worth noting that ASYST is described as a detailed integral BEPU (Best Estimate Plus Uncertainty) accident analysis code, which aligns with the SDTP's focus on advanced safety analysis methods including uncertainty assessment capabilities. ASYST incorporates advanced thermal hydraulic modeling capabilities to simulate the behavior of coolant flow, heat transfer, and phase changes within nuclear reactor systems. Its models complex phenomena similar to RELAP such as two-phase flow, thermal stratification, boiling, condensation, and heat exchange in reactor components.

The adaptive mesh refinement (AMR) techniques aid to dynamically refine and coarsen the computational mesh based on the local flow and thermal conditions. AMR allows for increased spatial resolution in regions of interest, improving the accuracy and efficiency of simulations by focusing computational resources where they are most needed. Supports multi-physics coupling, allowing for the simultaneous simulation of fluid dynamics, heat transfer, neutronics, and other relevant physical phenomena[22]. This capability enables comprehensive analysis of reactor behavior under coupled thermal hydraulic and neutron kinetics effects, facilitating more realistic and integrated simulations.

Similar to RELAP5, ASYST is typically used in conjunction with a graphical user interface (GUI) that provides a user-friendly environment for defining input data, running simulations, and visualizing results undergoes rigorous verification and validation (V&V) against experimental data, benchmark problems, and operational experience to ensure its reliability and accuracy. The code's predictive capabilities are validated for a wide range of reactor designs, operating conditions, and transient scenarios, providing confidence in its use for safety analysis and design assessment [27]. It is employed by nuclear power plant operators, regulatory agencies, research institutions, and engineering firms for analyzing reactor behavior under normal and abnormal operating conditions, assessing safety margins, and optimizing reactor performance.

1.2 Objectives

The motivation behind developing a workbook for RELAP5 and ASYST stems from the need for structured and accessible educational resources. The primary objective is to provide a comprehensive guide for new users on their initial training in the use of RELAP5 for thermal hydraulic analysis and serve as reference material for existing users. The workbook aims to model and simulate several cases of fluid flow and heat transfer starting with a simple 1-D pipe flow to a more complex multiple connected pipe and loop with heated and non-heated structures and branches. It is our objective to ensure new user are equiped with the basic knowledge of components in the system code. A step by step approach to develop input file for simulation will be outline and simulation results presented while sensitivity analysis of each cases model and simulated are carried out to assure the workbooks fidelity and to provide new users with an opportunity to gain intuition on the importance of some of the key input parameters..

1.3 Thesis Structure

The thesis is divided into 6 chapters, which will be briefly described below:

- Chapter 1: This chapter serves as the introduction to the thesis. It briefly outlines the structure of the thesis, emphasizing the motivation behind the work and the proposed objectives. It provides a short overview of RELAP5 and ASYST System Codes which will be expatiated on in the remaining part of the thesis..
- Chapter 2: In this chapter the historical background of system codes early development are presented. The system codes are then classified in a more practical approach.
- Chapter 3: An extensive literature review is presented in this chapter starting with a historical development as well as a general overview of nuclear reactor types, components, and classification while outlining their respective distinct technologies and safety. This chapter concludes with the need for a consolidated workbook for reactor system codes analysis.
- Chapter 4: This chapter provides general approach to the development of the system code workbook beginning with the theoretical concepts essential for the subsequent analysis in the thesis. Understanding the fundamental theory and governing principle (principle of fluid flow and heat transfer) employed by these codes is necessary to ensure users are qualified. The hydrodynamic and heat equations were solved analytically to verify the sample simulated cases results. This is followed by description of the essential components of the systems code that's is adopted in the thesis.

- Chapter 5: This section presents the components modeling of several cases presenting each cases nodalizations and detailed line-by-line input file development for each case modeled. Cases include simple pipe to a more complex multiple pipe connections, and loops with heat structures. Efforts are directed towards designing a guide that allows for customization according to user-specific needs.
- Chapter 6: Presentation of the simulation results are present for the 25 cases modeled coupled with the sensitivity analysis. This covers both steady and transient simulations, heated and non-heated structures, pump simulation, and loops powered by both pump and TDJ.
- Chapter 7: This chapter presents the conclusions drawn from the research and recommendation for future studies.

The Reactor System Code Analysis Workbook aspires to serve as an invaluable asset in enhancing safety, and advancing the field of nuclear engineering by promoting efficient and self-taught usage of the system codes.

Chapter 2

Background

2.1 Early Development

The initial development and use of reactor system analysis codes, known as the first generation of reactor analysis tools, marked significant milestones in the history of nuclear engineering. These early achievements laid the foundation for the sophisticated computational tools we now use for reactor design, safety analysis, and operation. Below are some key developmental milestones from the early history of reactor system analysis [28].

Significant developments took place across the 1960s in the growing ability to simulate reactor system behaviour using computer-based simulations – paving the way for future generations of reactor system codes. FLASH-1 (Fast Linear Analysis and Simulation of Hydraulics), the direct predecessor of RELAP, was the first in this series of codes, and was only able to use a three-volume simplified model (three zones with mixing) due to the limited computational power available at the time [28]. FLASH-1 was developed at the Idaho National Laboratory (INL) and the NRC, the aim being to create a program to manage these fluid dynamics and heat transfer processes in reactor system behaviour, especially those under transient and accident conditions.

RELAP Development: RELAP (Reactor Excursion and Leak Analysis Program, RELAP) code was developed in the US late 1960s by researchers at the INL to simulate reactor transients and accidents in pressurised water reactor (PWR). It was one of the earliest generalised thermal hydraulic system codes for PWRs, containing two-phase flow models, and state-of-the-art numerical methods (at the time) for simulating reactor behaviour for a variety of operating conditions.

RELAPSE-1 (RELAP1) was written in FORTRAN IV in 1966 on IBM-7040 and CDC-660 machines; RELAP2 was developed next in 1968, and was one of the first dedicated reactor analysis codes (including point kinetics and thermalhydraulics, but largely using the same three-volume system as RELAP1) [28]

In the 1970s, advanced numerical methods, more representative physical models and faster computers contributed to system codes maturity. RELAP4 development included one-dimensional fluid flow modelling and modelling of the cross-flows important to NPP systems. The Level Swell experiment series of General Electric (GE), started in the same period, became of high importance for code validation. Three main thermal hydraulic system codes were developed in the 1980s namely RELAP5, CATHARE and ATHLET [29][30][31], and CATHENA in Canada, covering all reactor types and many transient run scenarios.

RELAP5 series began with the original RELAP5/MOD0 in 1979, continued through the 1980s and the 1990s, with modified versions (MOD1, MOD1.5, MOD2, etc.) that were released during those years and included ever-expanding capabilities. Some of the most advanced versions of the RELAP5 series, such as the

RELAP5/MOD3, emerged in the 1990s and represented a major innovations in geometric flexibility. For instance, in 1996 a three-dimensional three-phase hydrodynamics code RELAP5-3D was released — it is a fully three-dimensional hydrodynamics code but was only available through the INL version of RELAP5 and also included inegrated neutron kinetic modelling capabilities characterises the Reactor-to-Environment and Hydrothermal LOop/Gravity Inclined Flow (REHLI/GIF) code and includes features such as full three dimensional hydrodynamics, including piecewise constant cross-sectioning of the control volume — resulting in rectangular, cylindrical and spherical geometries; variable gravity to model moving systems; integration with the NESTLE neutron kinetics code that provides volumetric energy deposition and heating source formulation in the full recirculation transient, and consistent in terms of grid density and time step lengths; and, the ability to model alternative boundary conditions for the control volume surfaces at the outlet, where the energy and mass balances are solved. In the late 1990s, the International RELAP Users Group (IRUG) was formed to ensure the development of RELAP5-3D provide license support and distribute the code. [28][31]

Starting in the 2000s, new multi-physics neutron kinetics, thermal hydraulics, fuel behaviour and structural mechanical models started to become available for holistic simulations. RELAP-7 (Reactor Excursion and Leak Assessment Program) code developed by Idaho National Laboratory (INL) for systems safety analysis was one of the pioneering codes for multiphysics modelling for the next-generation reactor systems. The Consortium for Advanced Simulation of Light Water Reactors (CASL) tools also began developing multiphysics models for next-generation reactor safety analysis code [32].

In the 2010s and beyond, we see a large increase in the use of High-Performance Computing (HPC) resources for system code analysis[32], especially for applications where the problem is too large and complex to perform a simulation of sufficient accuracy in an acceptable amount of time. The remarkable pace of computational developments in the early years of nuclear engineering provided the bases for modern approaches, some of which still have value for education and research today.

2.2 Key Milestones Achieved by Reactor System Codes

Advanced codes like RELAP-7, developed by Idaho National Laboratory, represent a new generation of nuclear reactor system safety analysis tools. These codes provide more accurate and reliable simulations of reactor behavior under both normal and accident conditions[33]. However a vast majority of safety and operational analysis are still performed with the RELAP5 variant, or other codes of the same vintage (e.g., CATHENA).

These codes all incorporate conservation equations, flow regime models, and advanced numerical methods to predict reactor system performance. A rigorous verification and validation process has been developed for each reactor analysis code including comparisons to relevant experiments and code-to-code validation. To ensure code accuracy and dependability across a range of physical phenomena and accident scenario, there is the need to performing sensitivity analysis, validation of experiments, and the creation of validation matrices [34].

Modern codes combine thermal-hydraulics, neutronics, and fuel behavior models to provide a more comprehensive analysis of reactor systems. This integration allows for more realistic simulations of complex reactor phenomena. They have been adapted and applied to various reactor designs, from low-power research reactors to high-flux reactors, demonstrating their versatility and importance in reactor safety analysis [34]. The International Atomic Energy Agency (IAEA) has outlined specific milestones for nuclear power program development, including the establishment of regulatory frameworks. System codes play a crucial role in meeting these regulatory requirements and ensuring reactor safety[35][36].

2.3 Classification of Reactor System Codes

According to the NRC[37], the nuclear analysis codes can be categorized based on discipline such as: Safety Analysis codes, Fuel Behavior Codes, Reactor Kinetic Codes, Thermal Hydraulics Codes, Severe Accident Codes, Containment Analysis Codes, Structural Analysis Codes, Probabilistic Safety Assessment (PSA) Codes and Multiphysics Codes. Expanding on the above classification, this project has provided an approach to classify reactor system codes based on the following factors;

2.3.1 Based on Physics Model

(a) Neutronics Codes: Employ for the calculation of neutron transport, nuclear reactions and neutron flux distribution in the reactor core (e.g., MCNP [Monte Carlo N-Particle Transport Code]; Serpent [68], SCALE [Standardized Computer Analyses for Licensing Evaluation][38], NESTLE< and PARCS.

(b) Thermal Hydraulic Codes: Thermal hydraulic codes simulate the thermal hydraulic transients of nuclear reactor plants, ie, they calculate single and two-phase flow using a 1-dimensional approach to solve the governing equations, the heat transfer and the entire thermodynamic processes. Examples include: RELAP5

(Reactor Excursion and Leak Analysis Program); TRACE (Transient Reactor Analysis Code); CATHARE [30].

(c) **Fuel Performance Codes:** These codes model the behavior of nuclear fuel during reactor operation, including thermal, mechanical, and chemical processes. Examples include FRAPCON (FRAPCON Nuclear Fuel Performance Code), FRAPTRAN (Fuel Rod Analysis Program – Transient), and BISON [33].

(d) **Multi-physics Codes**: Integrate multiple physics phenomena, such as neutronics, thermal hydraulics, fuel performance, and structural mechanics, into comprehensive simulations. Examples include RELAP-7, coupled neutronics thermal hydraulics codes within CASL (Consortium for Advanced Simulation of Light Water Reactors), and OpenFOAM-based codes for multi-physics simulations. In addition flexible finite-volume codes such as GOTHIC are used extensively for containment analysis where steam flows, temperature, flow paths, and hydrogen migration are all considered [40].

2.3.2 Based on Computational Methods

(a) **Deterministic Codes:** Deterministic codes solve the governing equations of reactor behavior using deterministic methods, which rely on predefined mathematical models, discretization techniques, and closure equations (if needed). These codes typically use finite difference, finite volume, or finite element methods to discretize the spatial domain and explicit or implicit time integration schemes to solve transient problems. This includes the RELAP, Dynamic 3-Dimension Core Model (DYNED), Coolant Boiling in Rod Arrays - Two Fluids (COBRA-TF), and Purdue Advanced Reactor Core Simulator(PARCS) [41].

(b) Stochastic Codes: Stochastic codes employ probabilistic methods, such as Monte Carlo simulations, for solving neutron transport and other stochastic processes in nuclear reactors. These codes use random sampling techniques to simulate the behavior of neutrons and other particles, providing accurate solutions to complex reactor problems involving neutron transport. Examples include MCNP (Monte Carlo N-Particle Transport Code), OpenMC, and Serpent [42].

2.3.3 Based on Application Area

(a) Safety Analysis Codes: The Safety analysis codes are computational tools specifically designed to assess the safety of nuclear reactors under various operating conditions and transient scenarios. These codes play a critical role in assessing nuclear reactor system performance, in evaluating design safety margins, and in meeting licensing requirements. In safety analysis codes, one evaluates the behaviour of a nuclear reactor system in accident and abnormal conditions scenarios by following all features, including changes, of the reactor response and safety systems during normal operating conditions and in accident and abnormal conditions. Trace Operational Support Codes TRACE (Transient Reactor Analysis Code): A thermal hydraulic system code maintained by the U.S. Nuclear Regulatory Commission (NRC) for safety analysis and associated operational support and licensing of light water reactors (LWRs)[35][39].

(b) Serve Accident or Beyond Design Basis Codes: These codes are used to model and analyse beyond the normal design and operational conditions of a nuclear reactors. They are vital for understanding and mitigating the impact of extreme accidents that can lead to a core melt down, containment failure, and release of significantly radioactive materials. These codes are integral part of the ongoing efforts in enhancing nuclear reactor safety. Examples of these are MELCOR (Methods for Estimation of Leakages and Consequences of Releases), ASYST, ATHLET-CD (Analysis of Thermal-hydraulics of LEaks and Transients with Core Degradation) and MAAP (Modular Accident Analysis Program) [43].

(c) Design Optimization Codes: These are computational tools used in nuclear engineering to optimize the design of nuclear reactors for enhanced performance, safety, and efficiency. These codes minimize (or maximise) an objective function that represents some performance metric or engineering objective function of reactor design conditioned on some set of inputs that is a function of some set of inputs (reactor design) called a design vector (the design variables: parameters or variables reflecting the design of the reactor, for example, arrangements of fuel assembly, positions of the control rods, flow rates of coolant, dimensions of the reactor core, enrichment of fissile fuel atoms) which are usually continuous but can be discrete.[44] Some commonly used algorithms for optimisation problems in nuclear reactor design include gradient-based methods (for example, gradient descent and Newton's method), evolutionary algorithms (such as genetic algorithms and particle swarm optimisation) and stochastic optimisation methods. Examples includes, Dakota, GATUNA (Genetic Algorithm for Time-Dependent Optimization and Nonlinear Analysis) and CONDOR (Core Optimization by Differential Evolution and Reactor analysis) [45]

(d) Operational Support Codes: Operational support codes provide real-time or near-real-time monitoring and analysis of reactor performance during normal operation, assisting operators in optimizing reactor operation and diagnosing operational issues. They are computer simulation codes used by nuclear power plant operators and engineers to assist them during reactor operation. Various operational scenarios are simulated with their codes, which provide real-time or near-real-time monitoring of reactor behaviour and data analysis at the operational site during normal operation to aid operators in optimising reactor operation and diagnosing operational problems. RETRAN, a simple 1D approach to system analysis using drift flux models, is often used in the USA to perform operational assessments and optimization.

(e) Research and Development Codes: Research and development (R&D) codes in nuclear engineering are computational tools used for advanced studies, innovation, and technology development in the field of nuclear energy supporting the development of new reactor technologies and methodologies. These codes are designed to investigate fundamental physics phenomena, explore novel reactor designs, develop advanced fuel cycles, and evaluate innovative safety features. Examples are SCALE (Standardized Computer Analyses for Licensing Evaluation) and ORIGEN developed by Oak Ridge National Laboratory for nuclear reactor physics analysis and isotopic inventory and decay heat calculations in nuclear fuel and materials [46] are examples of codes used in R&D, but also have licensing and operational support uses as does RELAP, TRACE and other thermal hydraulic codes.

(f) Commercial Codes: Commercial reactor system codes are proprietary software packages developed and sold by commercial vendors for use in the nuclear industry. Commercial and research codes in nuclear engineering serve different purposes and are developed with distinct objectives, target users, and characteristics. The primarily purpose are such as licensing support, engineering design, operational support, and safety analysis in the nuclear industry. These codes often have a wide range of features, capabilities, and modules to support various applications in nuclear reactor analysis, thermal hydraulics, neutronics, fuel performance, and safety analysis as discussed in previous sections. These may include codes like FLUENT and StarCCM in the CFD realm, and RELAP5-3D from Idaho National Labs in the system thermal hydraulic area.

2.3.4 Based on Specific Reactor Types

The CANDU Codes are computational tools specially developed for the analysis, simulation and optimisation of CANDU-type nuclear reactors. CANDU's utilize natural uranium fuel, heavy water moderator and coolant, and have on-power refuelling capability. They also offer exceptional flexibility in terms of fuel cycles with options to burn natural uranium, spent fuel from LWR reactors, MOX fuels and even thorium based fuels. The flexibility emanates from CANDU's reactor design, its control and instrumentation system, the fuel management scheme and the operating strategy. CANDU Industry Standard Toolset (IST) codes is a collection of codes combined to form a computerised design and analysis system for reactor physics, thermal hydraulics, fuel management and safety analysis in CANDU reactors. Some examples are: CATHENA (Canadian Algorithm for Thermal Hydraulic Evaluation of Nuclear Applications): AECL's thermal hydraulic code for safety analysis and thermal hydraulic evaluation of CANDU reactors [47], WIMS-AECL for neutronic lattice calculations, and ELOCA for CANDU fuel behaviour modelling.

PWR-specific codes are targeted at simulating pressurized water reactors behaviour which include PWR core models, primary and secondary coolant loops, and safety systems for a wide range of steady-state and transient scenarios. BWR codes centered on modeling the characteristics of BWRs, such as two-phase flow dynamics, void fraction effects, and recirculation loop behavior (e;.g. TRAC-G from General Electric). Advanced reactor codes are tools used to simulate the behaviour of the next generation of reactor designs – small modular reactors (SMRs), high-temperature gas-cooled reactors (HTGRs) and molten salt reactors (MSRs)[61][63].

2.4 Significance of Reactor System Codes Analysis

Nuclear reactors are producing essential electrical power to the world, is reliable, and does not release significant CO2. So the analysis of nuclear reactor system codes is playing an important role in nuclear engineering field for reliability and safety of the complex reactor systems. The following are some of the significant uses of reactor system codes.

2.4.1 Design Optimization

Reactor system analysis is performed early on as part of the design of a nuclear reactor. At this stage, the reactor engineer performs thermal hydraulic (flow of heat) simulations at the detailed level to understand how different reactor core configurations, coolant flow paths and distributions of heat across the reactor can be optimised to generate the most energy with the least amount of thermal stresses and adequate safety margins. Neutronics simulations are also performed to optimise the design of the reactor core, the arrangement of nuclear fuel, the power profile, and various control mechanisms so as to produce the required power level with the required fuel burnup. These codes are used during the design phase of nuclear reactors to optimize system performance, safety, and efficiency [48]. By simulating different design configurations, operating conditions, and control strategies, engineers can identify design improvements, enhance thermal hydraulic performance, and optimize reactor operation for maximum efficiency.

2.4.2 Reactor Operation

Reactor system analysis is involved in the effective operation of nuclear reactors by analysing data from online monitoring systems, performing diagnostics and predictive analysis. These systems integrate the reactor instrumentation-data with the predictive models and analyse them to evaluate the reactor's performance, diagnose anomalies, and optimise the operational parameters to get the maximum output. Advanced control systems utilize reactor system models to regulate reactor power levels, control reactivity, and maintain reactor stability under varying load conditions [49]. Training simulators for licensed operating staff also utilize the modelling capabilities in order to virtually assess operators performance to anomalous conditions. Finally, power plant data can be used for integral-level validation of various aspects of the TH code being assessed. Operational simulations can also assists the operators in optimising the reactor conditions like fuel burnup, diagnosing operational problems and creating operational procedures for safe and efficient reactor operation (50).

2.4.4 Safety Assessment

Reactor system codes analysis is essential for evaluating the safety of nuclear reactor designs and operations under various transient and accident conditions, and are an important part in the reactor licensing process. It allows engineers to simulate and analyze potential scenarios such as loss-of-coolant accidents (LOCAs), control rod ejections, reactor startups, and shutdowns to assess safety margins and identify potential safety vulnerabilities [52]. Severe accident analysis simulations assess the consequences of core meltdown scenarios, containment integrity, and radioactive release pathways to inform emergency planning and mitigate radiological hazards [51]. Licensing applications of the codes are numerous and also include validation steps to ensure code accuracies are well quantified.

2.4.5 Accident Management

Reactor system codes (RSC) development is also aimed at improving accident management strategies and for responding to abnormal and emergency conditions in real time for reactors. In other words, engineers can improve accident mitigation strategies and emergency response procedures by simulating the accident under various scenarios and analysing the possible consequences [53].

2.4.6 Licensing Support

Notably, reactor system codes analyses are part of licensing efforts for new reactor designs and for proposed changes to operating reactors. The rationale is straightforward: as stated by the US Nuclear Regulatory Commission, for safety analyses of reactors to constitute an adequate basis for licensing, 'it must be shown that the analyses were performed using validated codes and methods'. The Agency's safety requirements 'were established to satisfy nuclear safety standards for the protection of the public health and safety' [54].

2.4.7 Training, Research and Development

Reactor system analysis drives innovation and advancements in nuclear reactor technology through training, research and development activities. Training and education to familiarize nuclear engineers, operators, and regulators with reactor behavior and safety principles. This provides a valuable tool for conducting virtual simulations and training exercises to improve understanding of reactor dynamics, transient behavior, and safety protocols. The analysis supports efforts in nuclear engineering by providing a platform for studying new reactor designs, advanced fuel cycles [55], and innovative safety features. This facilitates the evaluation of new technologies, materials, and operational strategies to enhance reactor safety, performance, and sustainability [56].

Advanced modeling techniques, computational methods, and simulation tools enable researchers to explore novel reactor concepts, advanced fuel designs, and innovative safety systems to improve reactor performance and sustainability. Experimental validation studies, benchmarking exercises, and collaborative research initiatives facilitate the validation and verification of reactor system models and promote knowledge sharing within the nuclear engineering community.

Chapter 3

Literature Review

3.1 State of the Art on System Code Analysis

System code analysis for nuclear reactor simulation is characterized by the continuous progress in computational capabilities, modelling methods and validation approaches [54]. This progress is the outcome of the increasing requirement for more accurate, efficient and higher reliable analyses for the design, safety assessment, and operational decision-making for nuclear installations. In line with the changing application requirements, the high data volume generated and associated computational time for complex simulations calls for advanced resource facilities such as high-performance computing to obtain results on acceptable timescales.

In recent years, conventional single computer simulation models have been enhanced and replaced by parallel programming approaches that enable simulation runs with comparatively high accuracy, large spatial and temporal resolution by enabling scalability and distribution of code modules on a large number of computing devices [55]. Modern HPC platforms using parallel programming frameworks, are able to increases the computation performance by leveraging the work sharing approach using suitably optimized algorithms. The enhanced computational power allows engineers to explore wider ranges of operating conditions, scenarios and quicker responses to accidents [56].

Over the last 2 decades, more emphasis is put on uncertainty quantification and sensitivity analysis in system code analysis to calculate how uncertainty and sensitivity in input data and model assumptions propagate into simulation results [55]. Probabilistic methods are also used, such as Monte Carlo simulations and Bayesian inference, to quantify uncertainties and conduct uncertainty quantification of reactor performance and safety.

Rigorous validation and benchmarking processes are carried out, to verify the performance of system codes against test-case problems, experimental data, operational data of the nuclear power plants, and analytical solutions [53] In particular, international benchmark exercises are carried out, bringing the developers and users of various codes together – the OECD/NEA International Standard Problem (ISP) series, and various IAEA sponsored Coordinated Research Projects (CRPs), provide opportunities to compare the predictions and to find the gaps, weaknesses and encourage the code developers and users to further validate modelling capabilities.

3.2 Overview of Nuclear Reactor Types, Components, and Operational Principles

A nuclear reactor is designed to exploit controlled nuclear reactions in order to produce heat energy, which is subsequently converted to useful electrical energy based on the energy available at the sub-atomic electromagnetic-interaction scale. They are an excellent base load electricity generation option and have very low carbon emissions per MWh of energy produced [56, 57]. As a consequence, a combination of large reactor designs (Monark, AP1000,) with some set of Small Modular Reactors might be able to succeed to provide the demand of low-carbon energy in the future. They also have applications in assisting electricity generation, propulsion systems and research. [58].

3.2.1 Nuclear Reactor Technology

Since the first appearance of nuclear reactors, many reactors have been designed both for peaceful and military purposes [59] . Besides electric power production purposes, nuclear reactors have been employed for the propulsion of vessels, rockets, and satellites and for medical use and research [60]. During the WWII era, nuclear projects were conducted under top-secret conditions which created different designs in different nations [61]. Some examples of these reactors are the popular PWRs which stand for Pressurized Water Reactor, the BWRs which stands for Boiling Water Reactors (BWRs) both are referto Light Water Reactors (LWRs), Others are the CANDUs (Canadian Deuterium Uraniu), The RBMK (Soviet Reactor Bolshoy Moshchnosti Kanalniy), and the russian VVERs (Voda Vodyanoi Energetichesky Reactor (VVERs)..

There are numerous methods to classify the nuclear power plants. A classification might be made by the type of fuel used, the generation of reactor, kind of moderator material, coolant, reactor scale and so on. According to [61][62][63], reactors can be studied according to five groups: Pressurized Water Reactors (PWRs), Boiling Water Reactors (BWRs), Pressurized Heavy Water Reactors (PHWRs), Gas-Cooled Reactors (GCRs), Liquid-Cooled Graphite Reactors (LGRs), Liquid Metal Fast Breeder Reactor (LMFBRs).

(a) Pressurized Water Reactors (PWRs)

Pressurized water reactors (PWRs) are the most common type of nuclear reactors used in the nuclear power stations across the globe specially for production of electricity. They are part of the larger group of LWR (light water reactor). 'LWRs', as their name states, use an ordinary kind of water as coolant and moderator. PWRs are nuclear reactor where pressurized water is employed both as coolant and moderator.

The heat from the chain reaction in the reactor core is passed to the secondary coolant loop in the steam generator for driving the turbines to produce electricity.The schematic of the this type of reactor is displayed in Figure 3.1a

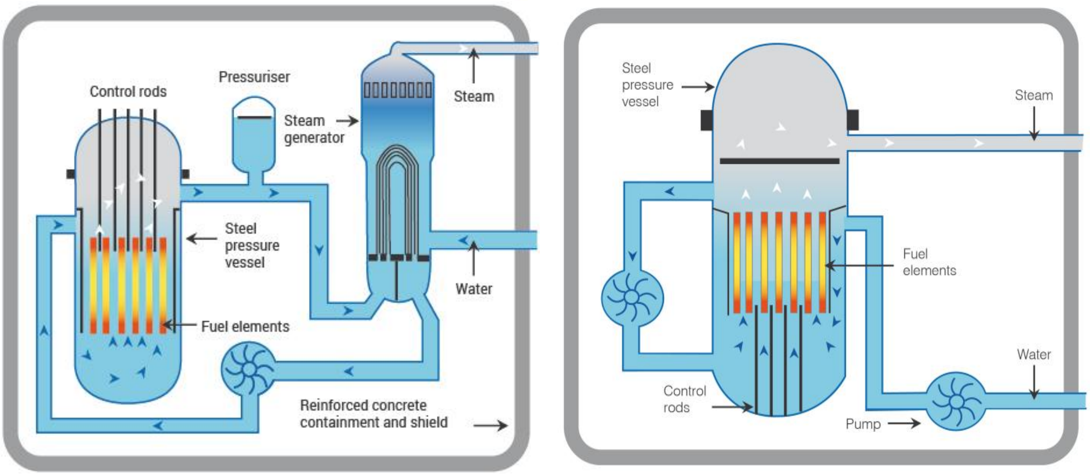


Figure 3.1 Schematic of (a) Pressurized Water (b) Boiling Water Reactors adopted from [61][63]

(b) Boiling Water Reactors (BWRs)

BWRs are like PWRs, but use the single-loop design where coolant water is allowed to boil in the reactor core as depicted in Figure 3.1. Steam thus formed directly in the reactor core drives turbines. There was an attempt to reduce the number and complexity of the components (envisioning a simplification of ‘piping, pumps and purification arrangements’) and a considerable cost savings expected as there is no cost of the boilers. Because of the void fraction in the core and its affect on neutronics, BWRs can undergo unique transients such as oscillatory behaviour which are not as common as in PWRs.

(c) Pressurized Heavy Water Reactors (PHWRs)/CANDU Reactors

PHWRs are the third most common type of nuclear reactor in the World [64] and typically used in Canada where they are called CANDU (CANada Deuterium Uranium) reactors. It is also used in India, Argentina, Romania, India, Pakistan and China. PHWRs enjoy the advantage of having online refueling that enables more flexible and efficient operation. In this design, deuterium oxide (D2O), or heavy water, serves as the coolant as well as a moderator. This arrangement permits the use of natural uranium in fuel, hence they are cost effective in countries that possess abundant deposits of natural uranium and where fuel enrichment is not desired.

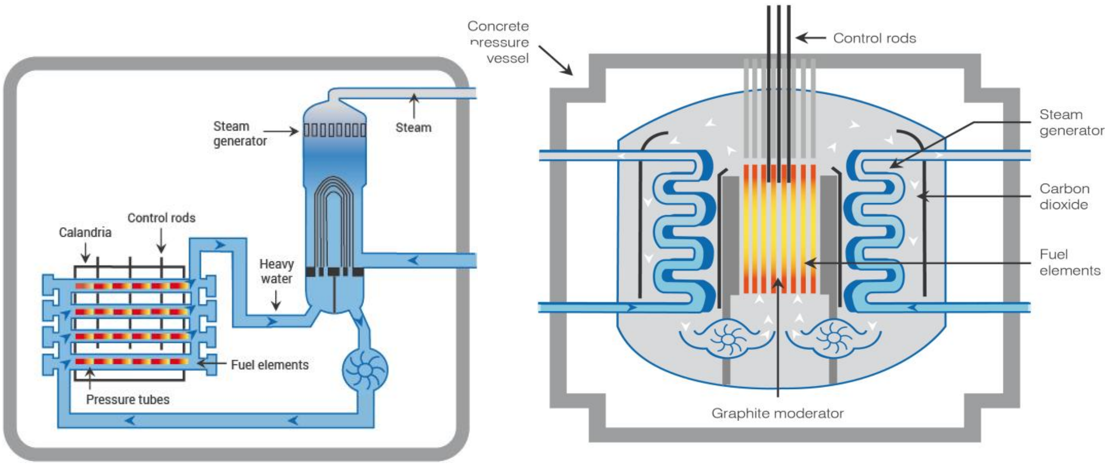


Figure 3.2 (a) CANDU Reactor (b) Gas-Cooled Reactor adopted from [61][63]

(d) Gas-Cooled Reactors

Gas-Cooled Reactors (GCRs): GCRs use gas (eg, carbon dioxide or helium) as the coolant instead of water, and are usually graphite-moderated and with enriched U235 fuels. Using a gas-based coolant makes it possible to achieve a significantly higher coolant temperature, which is essential for obtaining a high-efficiency steam cycle.

(e) Fast Neutron Reactors:

Fast neutron reactors use only fast neutrons to maintain the chain reaction with little or no moderator..The advantage of these designs is that they produce significantly less actinides, they can burn existing actinide wastes from other thermal fission reactors, and they can potentially breed new fuels from U238 or thorium.

(f) Liquid Cooled Graphite Moderated Reactor:

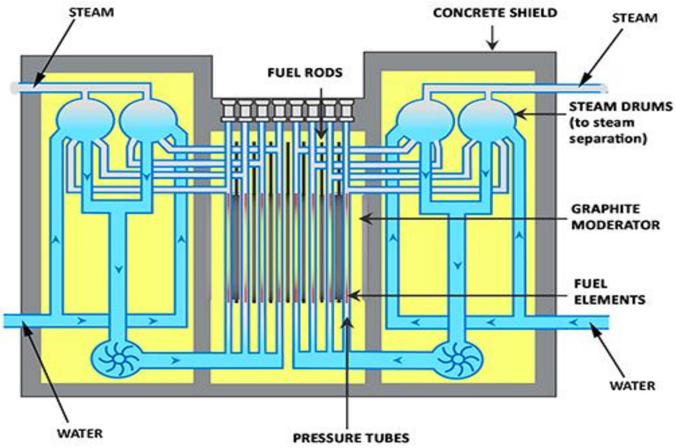


Figure 3.3 Liquid-Cooled Graphite Reactor adopted from[61][63]

The Liquid Cooled Graphite Reactor (like the RBMK) is also one of the nuclear reactors operated by using a graphite block for moderation. Fuel elements loaded in pressure tubes similar to that of the PHWR are passed through vertical holes in the graphite core. Graphite has excellent moderating property, has remarkably high efficiency and a compact design. Light water used for cooling purpose flows upwards over the fuel and within the tubes. Boiling of water takes place over the fuel and the steam which is generated is separated from the water in an external drum. The water is recirculated until saturation, and is sent to the turbine for generation of electricity[61].

(g) Liquid Metal Cooled Reactors

Liquid metal cooled reactors are unique in thtat it uses a liquid metal, usually sodium, as the primary coolant. This means that the heat of the fission products is removed by the primary coolant and transferred to a secondary sodium circuit via a heat exchanger. The secondary circuit transfers the heat to the steam circuit in another heat exchanger called a steam generator. Using sodium as the coolant has several advantages. First, it does not need to be pressurised like water, so one could build the sodium circuit with thinner-walled pressure vessels. Secondly, since it does not boil like water, the coolant does not need to pressurized greatly reducing the potential for a LOCA. Ultimately, the combination of these features makes the whole cycle more efficient, allows for more uses of high temperature heat in industrial applications, and through a fast-neutron-spectra can be used as part of a fuel recycling strategy.

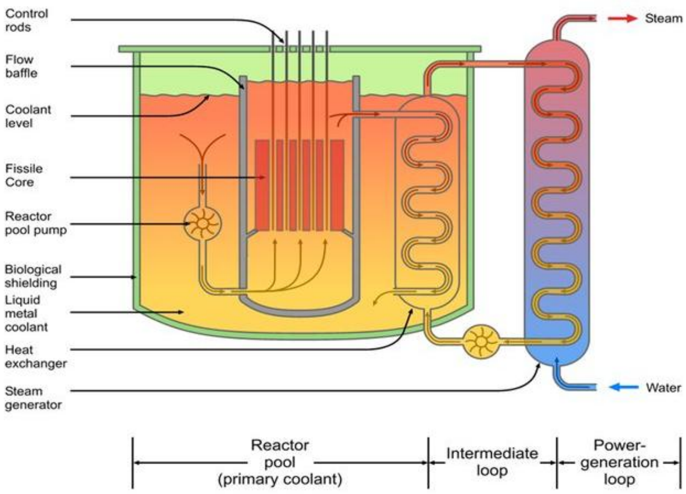


Figure 3.4 LMFBR adopted from [63]

(h) MOLTEN Salt Reactor

A Molten Salt Reactor (MSR) is a type of nuclear reactor that utilizes a liquid fuel mixture of carrier salt and fissile material, where the molten salt serves both as a fuel and a coolant simultaneously. The concept of MSR involves dissolving the nuclear fuel in a molten fluoride salt coolant, offering several advantages over traditional solid-fuel reactors [77]. MSRs are characterized by their design flexibility, safety advantages, and high-temperature operation, typically running at temperatures around 700-750°C, leading to enhanced thermal efficiency. MSRs have gained attention due to their potential for efficient utilization of thorium resources. They are considered a promising future nuclear reactor concept included in the Generation IV roadmap, offering significant versatility supported by their design flexibility, safety advantages, and unique salt chemistry characteristics [78]

3.2.2 Nuclear Reactor Components

Each nuclear reactor type and design has its own individual characteristics and unique operational processes. The following section provides a list of the most common components one might encounter in a nuclear power plant system.

(a) **Reactor Core**: the core of the reactor where the fuel assemblies are located and the nuclear reactions take place. The core is fed with coolant which transports heat from the fuel to other components in the system. The core typically also contains the control rods that are inserted into the core (along with other reactivity control devices such as shut off rods). Some cores include a separate moderator (like CANDUs), while others the moderation function is performed by the coolant (like in

PWR and BWRs). Finally, a core could be surrounded by a reflector or moderator to reduce the numbers of neutrons leaking from the core.

(b) Primary Coolant System: A coolant (e.g., water, liquid metal or molten salt, or gas) is circulated through a thermally conducting medium in the reactor core to remove the heat of nuclear reactions, transport it to the heat sink, then return the cooled fluid back to the core. The system usually contains pumps, piping, valves, heat exchangers, and coolant treatment systems.

(c) Moderator: The moderator slows down fast neutrons produced during nuclear reactions, increasing the probability of fission in the fuel and thereby continuing the chain reaction. Moderators are most commonly water or graphite and could either be separate systems (like in CANDU), or the moderation process could be done by the coolant itself.

(d) Control Systems: This system is comprised of several subsystems including reactivity control system, pressure and inventory systems, boiler pressure control etc. The reactivity control systems controls reactor power by moving control rods and other mechanisms (like liquid zone controllers in CANDU) to vary the neutron absorption (loss) in the core. By lowering (or increasing) the amount of absorption the reactor power can be increased or decreased as needed. Other control systems feature active or passive components to ensure primary side pressure and inventory, secondary side pressure and inventory, and other systems remain within specifications. The control systems keep the reactor in a stable state and ensure that it can be properly run under a wide range of operating conditions.

(e) Steam Generator (no relevant for BWRs): Heat carried by the primary coolant loop of the reactor is transferred to the secondary coolant loop of the reactor which is at a lower pressure, and causes the secondary side to boil inside the steam generators. This steam goes to the turbine to drive the turbines, is condensed and then returns to the boiler through boiler feedpumps.

(f) **Turbine and Generator**: The turbine converts the internal energy in the steam to mechanical energy and translates this to the rotor of the electrical generator; the generator in turn translates its mechanical energy into electrical energy, which is sent to the electrical grid.

3.2.3 Unique feature of the CANDU type Reactor

The key components of a CANDU reactor are the reactor core, the reactor vessel (pressure vessel), heavy water as the moderator with a separate high pressure heavy water coolant, the Calandria, horizontal fuel channel arrangement, and the heat transport system. The reactor core contains 100s of separate fuel channels, each with 12 natural uranium fuel bundles and the high-pressure coolant. Each of these fuel channels can be opened up by the CANDU fueling machine, and separately refueled while at power.

The Calandria, cylinder encircles the reactor core and holds the low pressure heavy water moderator. Each of the fuel channels passes through the calandria and is sperated by the concentric calandria tube and pressure tubes. These fuel channels run through the Calandria and house the fuel assembly in which the coolant flows. The concentric calandria/pressure tube arrangement (with a gas annulus between the two tubes) minimizes heat loses from the primary circuit to the moderator system. .

CANDU are fueled by natural uranium, which does not need to be enriched. Natural uranium is composed of 0.7 per cent uranium-235 (235U) and 99.3 per cent uranium-238 (238U). CANDU operate at a lower pressure and temperature regime compared with PWRs while slightly higher than the BWRs. Typically the pressure of CANDU reactor is 8-10 MPa and the coolant temperatures are around 285-300°C [64].

3.2.4 Operational Principles

The key feature that enables a nuclear chain reaction is the fission of uranium or plutonium nuclei that produce a number of neutrons with enough energy to induce further fission of nearby nuclei[65][66]. Hence, controlling the rate at which neutrons are produced during certain reactions and absorbed during others is one of the key aspects of reactor control. This is accomplished by the insertion or removal of specially designed neutron-absorbing materials that bind with neutrons and control the resulting nuclear reactions.

More generally, reactor control systems are multi-layered and contain an assortment of automatic processes that monitor a variety of reactor parameters (neutron flux, coolant flow, temperatures and other variables) and trigger mechanisms to intervene and maintain reactor stability and (eventually) achieve desired power output. In addition to these control system, independent and redundant safety systems are employed to mitigate the risks of accidents or a failure of any of the control elements.

3.3 Nuclear Reactor Safety

Nuclear power plants benefit from the energy arising from the fission reactions. The process, control and safety systems are design to ensure that the risks involved in power reduction are acceptably low. Regardless of how robust and reliable these systems are, various irregularities in the rector’s behavior may arise in expected and unexpected circumstances. The goal of Safety Analysis is to demonstrate, using computational methods, that the design meets all safety requirements for a wide range of postulated failures or events.

At a nuclear power plant, events can be identified and classified into categories primarily based on the frequency of the occurrence such as Normal Operation, Anticipated Operational Occurrences, Design Basis Accidents and Beyond Design Basis Accidents [61][67]. According to IAEA “the protection of employees, the public, and the environment from the harmful radiological effects that the nuclear facilities can cause could be ensured by establishing the highest safety standards.” [5]. Measures to achieve the highest standards of safety include;

- Limiting radiation exposure of workers and those in the public; limiting the release of radioactive materials to the environment.
- Decrease the probability of events leading to loss of control of the reactor core, fission chain reaction, radioactive material, depleted radioactive fuel, or nuclear waste.
- Minimizing the impacts of such events in case they were to happen.

The fundamental safety objective forms the basis of the safety requirements, and therefore applies to all phases of design, construction, commissioning, licensing, operation, decommissioning as well as management and disposal of nuclear waste

3.3.1 Defence in Depth Concept

NPPs must have multiple layers of safety both in plant design and management procedures. One of the key concepts of safety in NPP design is the Defence-in-Depth

Method. This concept's application extends to all safety-related activities. It encompasses organizational, behavioral, and design aspects across various operational states. This comprehensive approach ensures that all safety-related activities have multiple independent layers of protection. If a failure occurs at one level of protection or barrier, subsequent levels or barriers are in place to provide continued safety assurance [61]. The IAEA classifies these levels of defence [68] as:

Level 1: Prevention of abnormal operation and failures. The foundation of nuclear safety lies in preventing issues before they occur. This level emphasizes the importance of robust design and meticulous operation. It includes utilizing high-quality components that meet stringent industry standards, Implementing regular, comprehensive maintenance schedules, and adhering to strict operational protocols. By focusing on these elements, we aim to maintain normal operations and minimize the risk of equipment failures.

Level 2: Control of abnormal operation and detection of failures. Despite best efforts, abnormalities can occur. The second level focuses on early detection and rapid response to potential issues. Key features include: Advanced automatic control systems Sophisticated alarm networks, trained operators ready to take swift, and appropriate action. These measures help ensure that minor deviations are quickly identified and corrected before they escalate into more serious problems.

Level 3: Control of accidents within the design basis. Some incidents, while uncommon, are anticipated in the plant's design. This level addresses such scenarios through; Engineered safety features (e.g., emergency core cooling systems), Containment structures and Predetermined emergency procedures. These systems and protocols work in concert to manage design-basis accidents effectively, maintaining plant safety.

Level 4: Control of severe plant conditions, including prevention of accident progression and mitigation of the consequences of severe accidents. In the unlikely event of beyond-design-basis accidents, this level comes into play. It involves: Robust containment structures capable of withstanding extreme conditions, Additional safety systems for managing severe accidents and Strategies to prevent accident progression and mitigate consequences. The goal here is to regain control of the plant and minimize potential impacts, even in severe circumstances.

Level 5: Mitigation of radiological consequences of significant releases of radioactive materials. The final level addresses the potential release of radioactive materials. It encompasses: Comprehensive emergency response plans, Evacuation procedures for surrounding areas and Clear communication channels with the public and authorities. These measures aim to protect people and the environment in the event of a significant radiological release.

These layers of defence work together to prevent, control, and mitigate incidents, ensuring the safety of nuclear plants at all times.[68]

3.3.2 Safety Assessment Methods

Adequacy of the safety barriers to minimize the amount of radioactive material released while maintaining their integrity is demonstrated by the means of a range of varied quantitative studies for assessing and addressing potential safety risks using both deterministic and probabilistic methodologies [69]. The safety analysis is required to ensure that the defense-in-depth approach has been integrated into the plant's design. It must demonstrate that the nuclear power plant can comply with acceptable limits on radioactive releases and adhere to the dose limits across all operational states, as well as maintain acceptable limits during accident scenarios.

Providing satisfactory answers and supporting materials for this fundamental safety discipline may require calculations, experimental measurements, and simulations which are demanding in terms of time, funding, and manpower. Computer codes are valuable pieces of nuclear analysts' tool sets for the assessment of various operating and accident conditions. The results obtained from these studies help scientists and operators to predict the system behavior under real and anticipated reactor conditions [69].

3.4 The need for a Consolidated Reactor System Code Analysis Workbook

The field of nuclear engineering relies heavily on reactor system code analysis to understand reactor behavior under various conditions. While multiple specialized software platforms exist for thermal hydraulics, neutronics, and structural mechanics as discussed in previous chapter, their use presents significant challenges.

Interoperability issues, inconsistent modeling assumptions, and limited multi-physics integration often hinder comprehensive analysis. These problems can lead to divided understanding and potential errors in reactor system evaluation [70]. In addition, existing tools face usability and accessibility hurdles. Many require extensive training, have unintuitive interfaces, and lack customization options. High costs of commercial packages and substantial computational requirements further limit access, especially for academic institutions and smaller organizations.

To address these issues, there's a growing need for a unified Reactor System Code Analysis Workbook. This platform would allow for improved self-directed learning as part of an individual's training plan and can also help improve analysts intuition in regards to dynamic events. The proposed workbook aims to serve as a foundation for future development, potentially standardizing approaches for self-directed learning in the field.

The development of a more accessible, integrated platform could significantly advance reactor analysis capabilities by improving the training time and level of understanding of student being training in systems analysis. However, balancing comprehensive functionality with user-friendliness remains a key challenge. As the field evolves, addressing these issues will be crucial for enhancing nuclear reactor safety and efficiency.

3.4.1 Challenges of existing code and user guide

Limitations in usability, accessibility, integration and interoperability coupled with a complex and non-intuitive user interfaces are common challenges associated with existing tools used in nuclear reactor analysis. These can hinder the efficiency, effectiveness, and widespread adoption of these tools. Many nuclear reactor analysis tools and their user guides require extensive training and expertise to use effectively, resulting in a steep learning curve for new users.[71][73]

Commercial software packages may have better training platforms and tutorials, but often come with high licensing costs and authorization limiting access for academic institutions, small organizations, and researchers with limited budgets and background. [72]. Many existing tools lack flexibility in customization, making it difficult to adapt them to specific research needs or reactor designs. High computational resource requirements posses another challenge as some tools may have high computational resource requirements, including memory [71], processing power, and storage, limiting their accessibility for users with limited computing resources.

Among suggested approach to address this challenges includes to focus on open-source initiatives, enhanced flexibility and customization, improved computational efficiency and focus on user-friendly interfaces which this workbook supports to emphasize improved user interfaces and experiences to reduce the learning curve[74].

3.4.2 Impact of the Reactor System Code Analysis Workbook

The development of a Reactor System Code Analysis Workbook is essential to address the challenges associated with disparate codes and software platforms in nuclear reactor analysis. The workbook would support enhanced user training and awareness, reliability, and efficiency in reactor system analysis, thereby advancing safety, performance, and innovation in the field of nuclear engineering.

a. **Integrated Example and Solutions**: By consolidating various computational codes and software platforms into a single workbook, users would have access to a comprehensive suite of tools for reactor system analysis. This integrated platform would facilitate seamless data exchange, interoperability, and consistency across different simulation domains.

b. **Standardized Workflows and Templates**: The workbook would provide standardized workflows, templates, and guidelines for initial learning in the area of reactor system analyses. This would ensure consistency in modeling assumptions, numerical methods, and validation practices, thereby enhancing the accuracy and reliability of analyses.

c. **Training and Education Resource**: The workbook would be a training/education resource for undergraduate and graduate students of nuclear engineering, and for practicing engineers and researchers in industry and academia. It would provide tutorials for current and future reactor system analysis techniques, computational codes and development platforms.

d. **Validation and Verification Framework:** The workbook shall include a concise verification and validation framework to enable end users to perform their own due diligence and maintain confidence in the results generated from the computer simulations. It would provide standardized validation datasets, benchmarking exercises, and verification processes for assessing the performance of computational codes and software platforms.

Chapter 4

Methodology

4.1 Workbook General Approach

The essential components of the training package comprises the procedural steps such as problem definition, modelling, clear representation based on the dimensions and geometry of the design object. The hydrodynamic components of the models were divided into nodes and represented in block diagrams as Nodalization to simulate the operating characteristics of the system, input file development, practical hands-on exercises and several simulated flow cases of reactor components in steady and transient conditions for both single, multi-channels, bends, and pipe loops.

The number of nodes for the pipes components; heat structures (HS), and other components were defined accordingly. The pipes (PIPE) are modeled as series of control volumes with moderate node sizes, Branches (BRANCH) are modeled as 1 Junction IN and 2 Junctions OUT or vice versa of a single volume. The source and sink volumes are modeled as Time Dependent Volumes (TMDPVOL) to provide pressure boundary conditions to the required problems. Components were connected using single junctions (SNGLJUN), time-dependant junctions (TMD-J) or valves (VALVE) depending on the need. Heat sources were modeled using thermal components (HEAT STRUCTURE) which are the primary heat source attached to the required pipe components and uniform and/or non-uniform heats is created along the axial direction with a constant heat flux. Pipes representing the hot side of HX is the right side of the HS serving as the heat sink (Pipe 200P) of the heat exchanger (HX) to the second Pipe (Pipe 150) by natural circulation and heat conduction. The heat transfer in the components are modeled by a heat structure, which is attached on the respective components. The close loop model operates under forced circulation by use of pump components, and also model as a TDJ another loop as an optional solution.

All cases modeled and simulated were analytically validated based on fundamental theory of thermodynamics, fluid flow and heat transfer and sensitivity analysis of each cases performed.

4.2 Theory

The study of fluid flow and heat transfer (Thermodynamics) remains a vital component in the development of new technologies, ranging from nanotechnology resistors to large-scale nuclear power plants. Engineers and scientists continually push the boundaries of possibility to achieve enhanced performance and efficiency.

Prior to the scientific study of heat transfer began to take shape in the 18th century.o this, the prevalent theory of heat transfer was based on the concept of "caloric," proposed by Antoine Lavoisier in 1783 and further bolstered by the work of Sadi Carnot in 1824 which the Scientists adhered to in the early 19th century. However, Benjamin Thompson challenged Lavoisier's theory, demonstrating how heat could be generated through work, ultimately discrediting the caloric theory [18].

About the same time, Joseph Fourier laid down the foundations of continuum mechanics in his work on heat conduction in solids, known as Fourier's Law started in 1807 to 1822. Another significant contribution was made by James Clerk Maxwell when he identified heat as a form of molecular motion between 1831 and 1879. In his research, Maxwell elucidated the mechanism by which energy is transferred among gas molecules in contact. Standing on the shoulders of giants that these pioneers were, a number of scientists including William Henry Bragg, Osborne Reynolds, Carl Ludwig and Ludwig Prandtl published their seminal works for what is now known as heat-transfer..

4.2.1 Fluid Flow in Pipes

This is a fundamental aspect of thermal-hydraulic systems analyzed by RELAP5 and ASYST governed by the Navier-Stokes equations, it describes the conservation of momentum for a fluid. For simplicity, these codes often use simplified forms of these equations appropriate for the conditions encountered in thermal-hydraulic systems. These equations govern fluid flow and heat transfer in pipes, which serve as the basis for computer-based thermal hydraulic analysis in RELAP5 and ASYST. Understanding these fundamental principles of fluid flow, heat transfer, and steady-state analysis, user can gain proficiency in using this codes for thermal-hydraulic simulations in engineering analysis.

The continuity equation describes the conservation of mass in the fluid flow. It states that the mass flow rate (mass per unit time) in a pipe remains constant:

$$\frac{\mathrm{d\rho}}{\mathrm{dt}} + \nabla.(\rho \mathrm{V}) = 0 \tag{4.1}$$

Where ρ is the fluid density, t is the time ans V is the velocity vector.

In steady-state conditions, where the fluid properties don't change with time, this simplifies to:

$$\nabla.(\rho V) = 0 \tag{4.2}$$

The momentum equation, or Navier-Stokes equation, describes the motion of the fluid and the forces acting upon it:

$$\rho\left(\frac{dV}{dt} + (V.\nabla)V\right) = -\nabla P + \nabla.\tau + \rho g \tag{4.3}$$

Where P is pressure, τ is the stress tensor representing viscous forces, and g is the acceleration due to gravity. For steady-state, incompressible flow with no body forces, this simplifies to:

$$(V.\nabla)V = -\frac{1}{\rho}\nabla P + \nu\nabla^2 V \tag{4.4}$$

Where ν is the kinematic viscosity

4.2.2 Pipe Flow Equations

The behavior of fluid flow in pipes can be described by several important equations and principles. First, the **continuity equation** expresses the principle of conservation of mass for fluid flow. The equation for an incompressible fluid is the statement that the product of the density of the fluid (ρ) and the velocity of the flow (V) is constant. For incompressible pipe flow:

$$A_1 V_1 = A_2 V_2 \tag{4.5}$$

Where A is the cross-sectional Area of the pipe.

Secondly, in thermal-hydraulic system, determination of the pressure drop and heat transfer rate along the piping system is vitally important for performance and safety. Julieus Weisbach developed a relation based on the Bernoulli principle to determine the pressure drop as a function of the pipe diameter, material type, and fluid velocity. The formula that is used to calculate the pressure drop (ΔP) from frictional losses in a pipe is called **Darcy-Weisbach:**

$$\Delta P_L = f \frac{L}{D} \frac{\rho V_{avg}^2}{2} \tag{4.6}$$

Where L = length of the pipe, D = diameter of the pipe and is the Darcy friction factor, which is function of the Renold's Number..

$$f = \frac{64}{Re} \tag{4.9}$$

For turbulent flow, the friction factor is usually determined through an empirical correlation, or formula, because the friction is complex and depends on the flow conditions. An example of such a formula is the Colebrook-White equation, which predicts the friction factor from the Reynolds number (Re) and the relative roughness of the pipe (k/D)

$$\frac{1}{\sqrt{f}} =- 2log_{10} \left(\frac{k/D}{3.7} + \frac{2.51}{Re\sqrt{f}} \right) \tag{4.10}$$

Where f is the Darcy-Weisbach friction factor, D is the diameter of the pipe, Re is the Reynolds number and.k is the roughness height of the pipe. For fully turbulent flow in pipes with high roughness, the Haaland equation can be used instead:

$$\frac{1}{\sqrt{f}} =- 1.8log_{10} \left[\left(\frac{k/D}{3.7} \right)^{1.11} ++ \frac{6.9}{Re} \right] \tag{4.11}$$

This equation provides a more straightforward, though approximate, calculation of the friction factor. Accurate description of pressure loss and heat transfer rate strongly related to the determination of frictional losse f. In fully developed laminar flow, the friction factor is solely dependent on the Re number which is a dimensionless number (Re) (ratio of inertial forces to viscous forces) proposed by Osborne Reynolds use in determining the flow regime.

$$\mathrm{Re} = \frac{V_{avg} D}{\nu} \tag{4.12}$$

Where ν represents the kinematic viscosity of the fluid, V_{avg} is the average velocity of the fluid in the pipe, and D is the diameter of the pipe. For the Re values lower than 2300 flow regime is considered to be laminar. In contrast to laminar flow, turbulent flow is characterized by its chaotic and irregular behavior. This turbulence is attributed to the increased heat transfer coefficients resulting from the mixing action within the flow. According to the literature, the flow in a pipe is turbulent for a Reynolds number of Re above 4000, and fully turbulent when Re is above 10000. [17].

4.2.3 Pressure Drop Analysis

Pressure drop analysis allows optimization of the piping system design by selecting appropriate pipe diameters, minimizing pipe lengths, and reducing unnecessary fittings to achieve desired flow rates with minimal pressure drop. The following factors affects the pressure drop in pipe flow analysis

a. **Frictional Losses**: Longer pipes generally result in higher frictional losses due to the increased surface area in contact with the fluid. This leads to a greater pressure drop along the length of the pipe. Frictional factor also plays a vital role.
b. **Flow Velocity:** Pressure drop is directly proportional to the square of the flow velocity. Therefore, longer pipes may require higher flow velocities to maintain the desired flow rate, resulting in higher pressure drops.
c. **Pipe Diameter**: The diameter of the pipe also affects pressure drop. Larger diameter pipes generally experience lower frictional losses and, therefore, lower pressure drops compared to smaller diameter pipes for the same flow rate.

d. **Fluid Properties**: The properties of the fluid being transported, such as viscosity and density, also influence pressure drop. Higher viscosity fluids experience higher frictional losses, leading to higher pressure drops.

e. **Pipe Roughness**: The roughness of the pipe's interior surface affects frictional losses and, consequently, pressure drop. Rougher surfaces result in higher frictional losses and higher pressure drops.

f. **Piping System Layout**: The arrangement of pipes in the system, including bends, fittings, and valves, can contribute to additional pressure drop. These components introduce additional resistance to flow, which increases pressure drop.

g. **Energy Considerations:** Pressure drop results in a loss of energy in the system. Understanding the magnitude of pressure drop helps in evaluating the energy requirements for pumping or circulating the fluid through the pipes.

4.2.4 Heat Transfer Equation

Heat transfer in thermal-hydraulic systems, such as those simulated by RELAP5, plays a crucial role in determining system behavior. For example, in thermal hydraulic systems, fluids are typically transported through pipes and Fourier’s law governs heat transfer through the pipe walls, taking into account the temperature difference between the fluid inside the pipe and its environment. The rate of heat transfer (q) through the wall of the pipe due to conduction is described by Fourier’s law:

The rate of heat transfer (q) through a material (where q per second is equivalent to q / seconds, with seconds having units of time) is proportional to the negative gradient of temperature ($-\nabla T$) in the direction in which heat flows:

$$q = -kA\frac{dT}{dx} = kA\frac{T_1 - T_2}{L} \tag{4.13}$$

Where q is the heat transfer rate (in watts, W), k is the thermal conductivity of the pipe material (in watts per meter per Kelvin, W/mK), A is the surface area of the pipe wall, and $\frac{dT}{dx}$ is the temperature gradient (in Kelvin per meter, K/m).

While conduction releases heat into a cold fluid, a conduit can also lose heat through convection, when the surrounding fluid moves with the heat. We characterize this convective heat loss through the heat transfer coefficient (h):

$$q_{conv} = hA\Delta T \tag{4.14}$$

Where q_{conv} is the heat transfer rate due to convection (in W), h is the convective heat transfer coefficient (in W/m^2K), and ΔT is the temperature difference between the fluid and the pipe wall (in K).

4.3 Sensitivity Analysis and Uncertainties

In order to promote new-user knowledge of the flow phenomena, for each case in the Workbook, a series of sensitivity analysis is performed to demonstrate the dependency of the solutions on key input parameters. One-at-a-Time (OAT) Sensitivity Analysis of RELAP5 Simulation was adopted to evaluate the sensitivity of our models' output to changes in individual input and/or output parameters while keeping the other parameters constant. We first identify the input parameters in the codes that may have a significant impact on the output variables of interest. These include parameters such as geometries, dimensions, fluid properties, boundary conditions, and component characteristics.

We further established a base case scenario with set nominal values for all input parameters. For basic flow in a pipe model, inlet pressure was first set with known flowrate to determine the variation in temperature and possible pressure loss depending on the pipe orientation and geometry. In other cases, we set pressure at both ends at a predetermined values to compute the flowrate. These values are then varied (from -10 to +10 % of set nominal value) to evaluate the changes in parameter and analyzed the corresponding effects on fluid flow and heat transfer such as time of convergence, curve trends. Above process was repeated 3 to 5 times for each input parameter of interest.

4.4 Code Components Description

Components are organized collections of volumes and junctions and, to a lesser extent, the program is organized on components. Components are designed for either input convenience or to specify additional specialized processing. A pipe component is an example of a component designed for input convenience, since by taking advantage of typical features of a pipe, several volumes and junctions can be described with little more data than for one volume.

Components are numbered with a three-digit number, 001 - 999 as in Table 4.1. Components need not be in strictly in consecutive order so that changes to a model of a hydrodynamic system requiring addition or deletion of components are easily made.

Volumes and junctions within a component are numbered by appending a six digit number to the component number, CCCXXYYZZ. The CCC is the component number.At this scale, YYZZ are zeroes and XX is numbered sequentially starting at

01 for the volumes and junctions in the one-dimensional components presently defined [19].

Table 4.1 Code Component Numbering

Card Item	Format
Control options	100-199
Time step options	200-299
Minor edits	301-399
Plot requests	20300XXY
Hydrodynamic components	CCCXXNN

4.4.1 Common Features of the Code Components

Each volume's flow area, length, and volume must be supplied as input. Each one dimensional volume has a x-coordinate direction along which fluid flows in a positive or negative direction, and may have y- and z-coordinate directions if cross-flow connections are made to the volume. The x-volume flow area is the volume cross-sectional area perpendicular to the x-coordinate direction. The x-volume length is the length along the x-coordinate direction and similarly for the y- and z-coordinate directions. The hydrodynamic numerical techniques require that the volume be equal to the volume flow area times the length for each coordinate direction.

This requirement is easily satisfied for constant area volumes, but poses difficulties for irregular shaped volumes. Since it is very important that such a system code conserves mass and energy, with momentum being an important but lesser consideration, it is recommended that an accurate volume be used. The component input routines permit the volume, flow area, and length of each volume to be entered as three nonzero positive numbers or two nonzero positive numbers and a zero. The volume horizontal angle specifies the orientation of the volume in the horizontal plane.

The code numerics have no requirement for this quantity; they were entered so a graphics package could be developed to show isometric views of the system as an aid in model checking. The horizontal angle is checked to verify that its absolute value is less than or equal to 360 degrees, but no further use is made of the quantity. The volume vertical angle specifies the vertical orientation of the volume. This quantity would also be used in the graphics package and, in addition, specifies the vertical orientation of the volume coordinate direction important for calculating hydrostatic head. The vertical angle must be within the range 90 to -90 degrees.

The angle 0 degrees means the x-coordinate direction is in the horizontal plane; a positive angle means that the coordinate direction is directed upward; a negative angle means it is directed downward. Slanted vertical orientation, such as an angle equal to 45 degrees, is permitted. Note that as the vertical angle changes from zero, the y-coordinate is always in the horizontal plane, and that the x- and z-coordinates, and their associated faces move out of their original horizontal and vertical planes, respectively. The direction of fluid flow is indicated by the sign of the velocity relative to the coordinate direction. A junction connects a specified end of one volume to the specified end of another volume. This, in turn, establishes relative positioning of the volumes. Because of gravity heads, the relative position is important to any volume with a nonzero vertical component of a volume coordinate direction. If the coordinate direction in a volume with a vertical component is reversed but no other changes are made, the inlet and outlet ends of the volumes are also reversed.

RELAP and ASYST are both fundamentally transient-based solvers wherein the system of equations, initial conditions, boundary conditions and component information are used to solve the governing equations. For problems involving steady-state, code inputs are such that nothing (boundary conditions, valve positions, etc.) change throughout the simulation. Therefore, the code should predict the response of system starting from the initial conditions and then approaching steady (invariant) conditions as time progresses. If the initial conditions specified in the component input files are close or equal those of the end-steady-state, then RELAP outputs would show little change in time. If the initial conditions specified in the input files are far from the steady-state, then a large ramp can occur.

The user should be aware that poor definition of the initial conditions in the input file can cause large oscillations in code results, which may in turn cause a code crash with little or no warning. Some examples include non-physical flows direction or magnitude, non-physical temperatures, and thus while the code is attempting to resolve these discrepancies and produce physically consistent results, for some time steps oscillations can cause variable to go out of range (i.e., fluid properties outside of the available database). One path forward when starting a new model where initial conditions are not known, a user can start with simple initial conditions and make other components as “favorable” as possible. This would include initially setting the power to zero, or valve positions to fully open, ect…, and then running the code with input cards that gradually adjust and control the simulation towards the desired end-state.

For example one might start at zero, or very low power, and then gradually ramp the power up to the desired power level over 10s of seconds. Then a user can write a restart file, and for all subsequent simulations use the restart file which now should include physically accurate initial conditions. Finally, before implementing a transient (such as a valve opening to atmosphere to simulate a LOCA, the user should simulate period of constant conditions prior to the valve position change. The purpose is to establish a steady state prior to the transient of interest such that the effects of inconsistent initial conditions do not influence the actual transient under consideration.

4.4.2 Time-Dependent Volume

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

The state conditions as a function of time or some time-advanced quantity are entered as a table, with time or the time-advanced quantity as the independent or search variable. The table must be ordered in increasing values of the search variable, and each succeeding value of the search variable must be equal to or greater than the preceding value. Linear interpolation is used if the search argument lies between search variable entries.

End point values are used if the search argument lies outside the search variable entries. If constant state values are desired, only one set of data consisting of any search value and the associated constant data needs to be entered. The program recognizes when only one set of data is entered, and computer time is saved since the equation of state is evaluated only once rather than every time advancement. Step changes can be accommodated by entering two adjacent sets of data with the same time or an extremely small time difference. The default search argument is time. If no trip number is entered, or if the trip number is zero, the current advancement time is used as the search argument.

4.4.3 Single Volume

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

4.4.4 Time-Dependent Junction

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

In a simple pipe modeling application, a time-dependent volume and junction can be used to specify the inlet flow. Likewise, a time-dependent volume and junction can model the feedwater flow into a reactor steam generator. Controlling the fluid flow out of the pipe or controlling the water/steam flow out of the steam generator through a time-dependent junction is not recommended.

4.4.5 Single-Junction

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

4.4.6 Pipe

A pipe component is a series of volumes and interior junctions, the number of junctions being one less than the number of volumes, and the junctions connect the outlet of one volume to the inlet of the next volume. Pipe components can be used for those portions of the system without branches. Pipe components offer input conveniences, since most characteristics of the volumes and junctions in a pipe are similar or change infrequently along the pipe, and input data requirements can be reduced accordingly. Because of the sequential connection of the volumes, junctions are generated automatically rather than being individually described.

Although the input is designed to assume considerable similarity in volume and junction characteristics, any of the volume and junction features (such as flow area, orientation, pipe roughness, or control flags) can be changed at each volume or junction.

4.4.7 Branch

Branch components are provided to model interconnected piping networks based on one-dimensional fluid flow, which is adequate for most cases of branching and merging flow. Such situations include parallel flow paths from upper and lower plenums, and any branch from a vessel of large cross-section. A branch component consists of one system volume and zero to nine junctions. The limit of nine junctions is due to a card numbering constraint. Junctions from other components, such as single-junction, pump, other branch, or even time-dependent junction components, may be connected to the branch component. Use of junctions connected to the branch, but defined in other components, is required in the case of pump and valve components and may also be used to attach more than the maximum of nine junctions that can be described in the branch component input.

4.4.8 Pump

The pump component model can be separated into models for hydrodynamics, pump-fluid interaction, and pump driving torque. The pump component input provides information for the hydrodynamic and pump-fluid interaction models and may optionally include input for an electric motor to drive the pump. The hydrodynamic model of a pump component consists of one volume and two associated junctions.

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

a. Pump Performance Modeling

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

b. **Pump Characteristic Curve**

A pump characteristic curve is a graphical representation of the performance characteristics of a pump, showing the relationship between its operating parameters and its capacity to deliver fluid at various conditions[19]. The curve typically plots the pump's head (pressure) against its flow rate under different operating conditions.

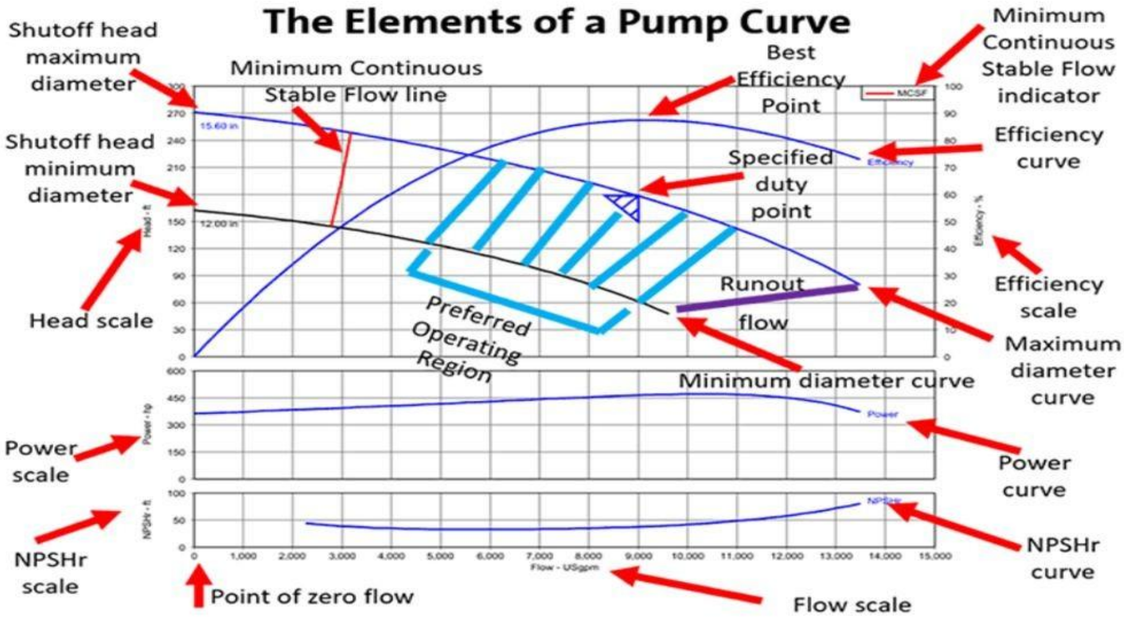


Figure 4.1: Typical Single Stage Curve adopted from [79]

The key components of a pump characteristic curve is described as in Figure 4.5 as follows;

i. **Pump Head (Pressure):** The vertical axis of the curve represents the head or pressure generated by the pump. Head is typically measured in units such as meters (m) or feet (ft) of fluid column, or pressure units like Pascals (Pa) or pounds per square inch (psi). It indicates the pump's ability to overcome resistance and lift fluid to a certain height.

ii. **Flow Rate:** The horizontal axis of the curve represents the flow rate of fluid through the pump, usually measured in units such as cubic meters per hour (m^3/h), gallons per minute (GPM), or liters per second (l/s). It indicates the volume of fluid the pump can deliver over a given time period.

iii. **Operating Points:** The intersection of the pump characteristic curve with system curves as depicted in Figure 4.6a represents the operating points of the pump under different system conditions (e.g., varying flow resistance). The actual operating point depends on the system's resistance (e.g., friction losses, elevation changes) and the pump's performance.

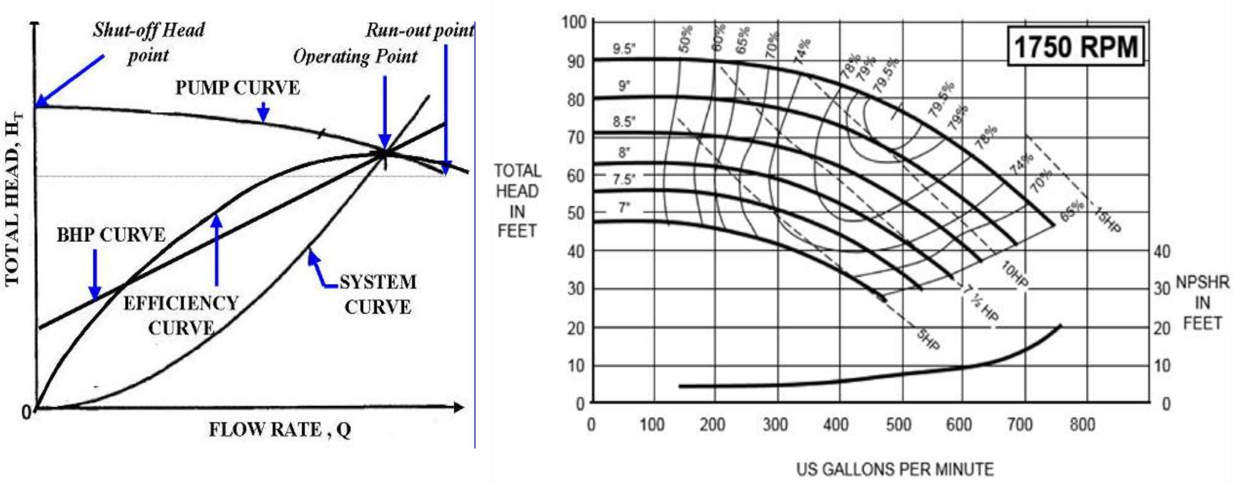


Figure 4.2: Pump Curve

(a) Best Operating point (b) The efficiency adopted from [79]

iv. **Efficiency:** Some pump characteristic curves also include information about pump efficiency, showing how efficiency varies with flow rate (see Figure 4.6b). This provides insights into the pump's energy consumption and overall effectiveness in converting mechanical power into hydraulic power.

Understanding the pump characteristic curve is essential for selecting, operating, and troubleshooting pumps in various engineering applications.

Chapter 5

Code Components Nodalization and Input File Development

5.1 Nodalization

Nodalization and input file development are crucial aspects of reactor system modelling, involving the creation of computational models and input data required for running simulations using reactor system analysis codes. This section provides an overview of nodalization and input file development in the context of reactor system analysis. Nodalization refers to the process of discretizing the physical domain of the reactor system into a set of interconnected nodes or control volumes. Each node represents a specific volume element within the reactor system, such as reactor core regions, coolant channels, heat exchangers, and other components. This involves defining the geometric boundaries, material properties, and boundary conditions for each node to accurately represent the thermal hydraulic behavior of the system.

The nodalization scheme should capture important flow paths, heat transfer mechanisms, and phenomena of interest within the reactor system. Nodalization should be designed to capture the spatial and temporal variations of flow, temperature, and other relevant parameters within the reactor system accurately. It is essential to strike a balance between model fidelity and computational efficiency by choosing an appropriate level of detail in nodalization. Factors such as resolution of features inside each component, nodal volume sizes, and boundary conditions should be carefully considered to ensure the accuracy and convergence of simulations.

In addition the impact of nodalization on code convergence, phenomena interactions, and linkages to multi-physics code-to-code coupling represent advanced user issues not covered in this work.

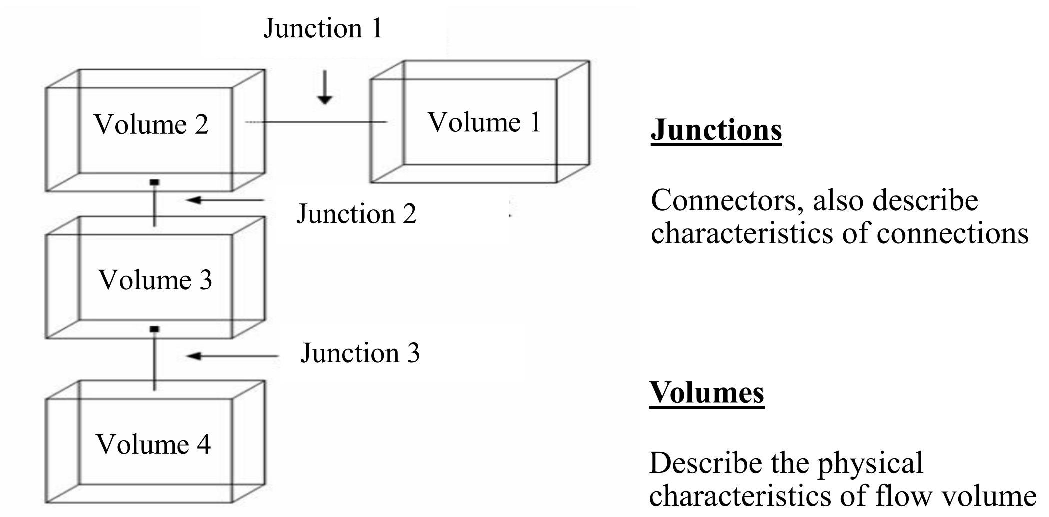


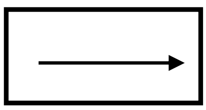
Figure 5.1 Basic Building Blocks for Fluid Network

Basically, the models considered in this project are composed of control volumes as defined by the user in a way that best gives proper and reasonable analysis. Therefore, the process of creating the control volumes is called nodalization. The following factors govern the choice of reasonable node size: numerical stability, run time, and spatial convergence/resolution. To achieve numerical stability, node length to diameter ratio should be unity or greater[16]. Generally, when simulating large and complex systems, nodes should be defined as large as possible without compromising spatial convergence of the results. That is because node size directly influences run time; the smaller the node, the smaller the maximum time step size to remain numerically stable.

However, suitable nodalization is problem dependent, and the modeller must exercise some judgment as to where in the model, nodalization sensitivity studies are required[16]. Researchers found that node sizes positively or negatively affect RELAP5 prediction results. A. Mangal et al found that RELAP5 predictions are very sensitive to nodalization. They found that predictions based on coarse and fine nodalisations differed significantly from the experimental data whereas for the base nodalisation (moderate node size), the predictions are relatively close to the measured results[64]

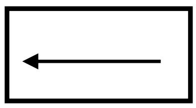
5.2 Pipe Flow Nodalizations

Pipe are divided into numbers of internal volumes. The code assigns internal volume numbers added to the component numbers. As illustrated in Figure 5.2, junction arrows directions define assumed flow direction (however the code will compute negative flows if the assumed direction was inconsistent with the final solution). The angle of inclination to both horizontal and vertical orientation is also a factor to determine flow direction. Negative flows will be to the opposite direction.



Azimuthal Angle 0 or 360^{o}

Inclination angle 0^{o}



Azimuthal Angle -180^{o}

Inclination angle 0^{o}

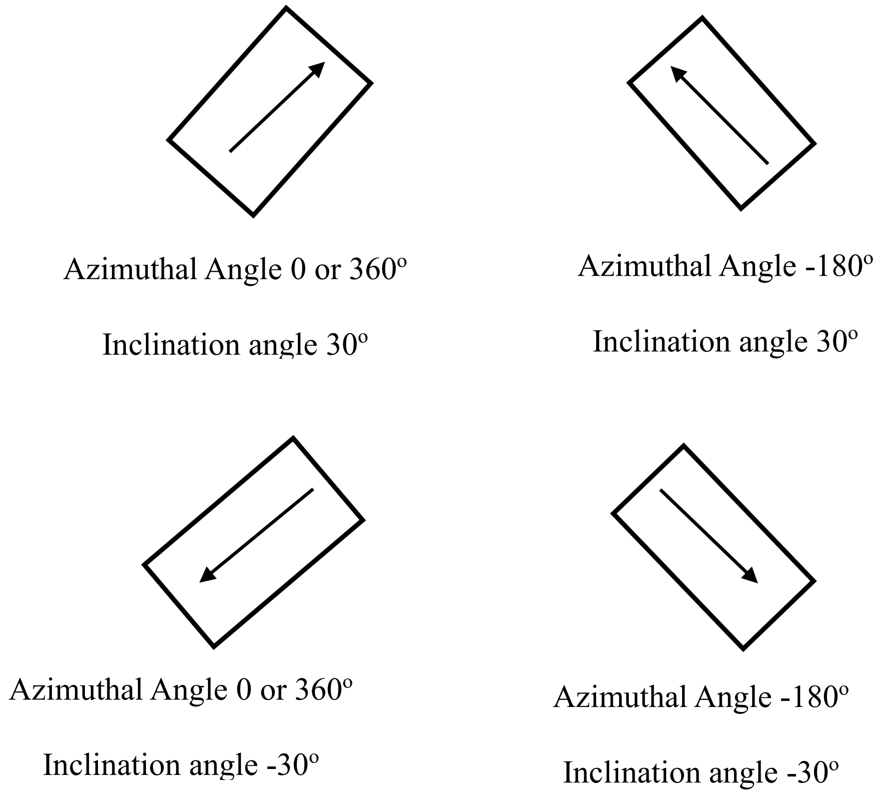
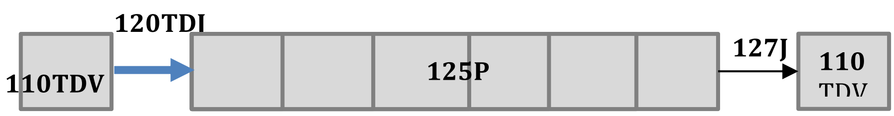


Figure 5.2 Volume Orientation

In this workbook, several model have been analyzed, simulated and result compared with analytical calculation. Below are the Nodalization for the models presented.

5.2.1 CASE #1 : Single Pipe Flow (1-D) Nodalization

The simulation input data and results for the nodalization of Figure 5.9 are presented in section 6.4.2 while the input file is presented in Appendix VI



(a)

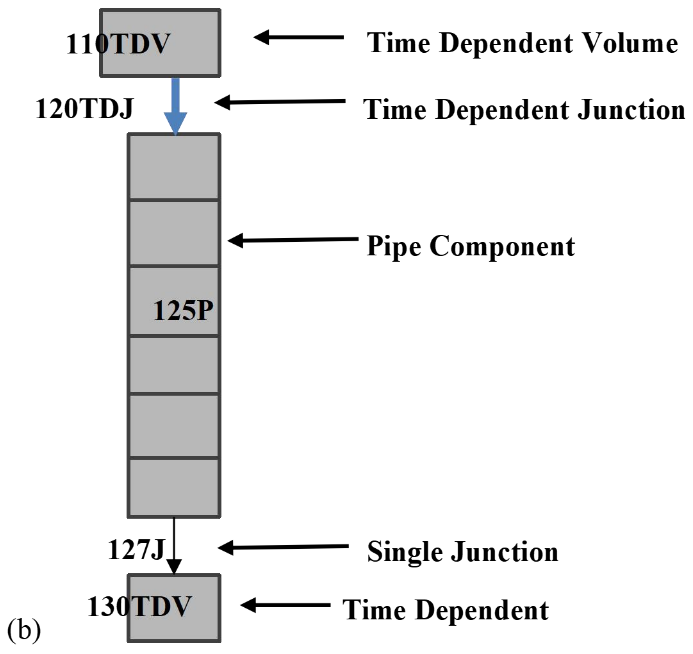


Figure 5.3 1-D Single Pipe Model (a) Horizontal (b) Vertical Orientation

For the above models, the pipe is divided into 6 internal volumes, The volume numbers assigned by the code are 12501, 12502, 12503, 12504, 12505 and 12506The inlet of the pipe (Fig a) is considered to be from the left to right (12501 to 12506) while for the vertical arrangement (Fig b), the inlet of pipe is from the top to bottom based on the junction arrow direction.

5.2.2 CASE #2: Series Pipes Connection

The simulation input data and results for the nodalization of Figure 5.4 are presented in section 6.4.2 while the input file is presented in Appendix VI

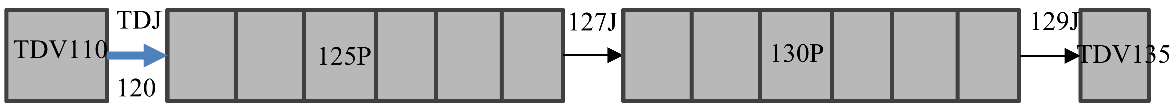


Figure 5.4 Series Pipe connection

5.2.3 CASE #3: Parallel Pipe (Multichannel) Nodalization

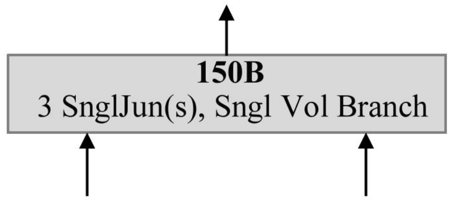


Figure 5.5 Branch Representation

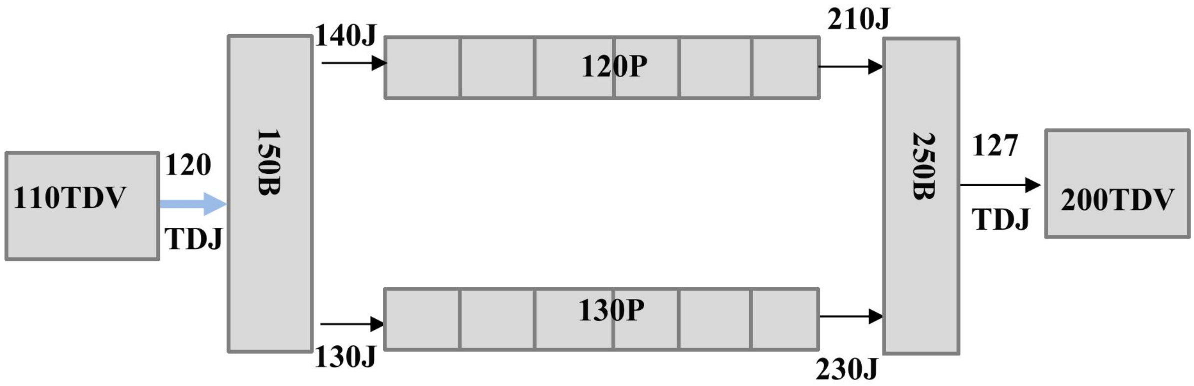


Figure 5.6. Parallel Connection using Branch

5.2.4 CASE #4: Elbows/Bends Nodalization

The simulation input data and results for the nodalization of Figure 5.9 are presented in section 6.4.2 while the input file is presented in Appendix VI

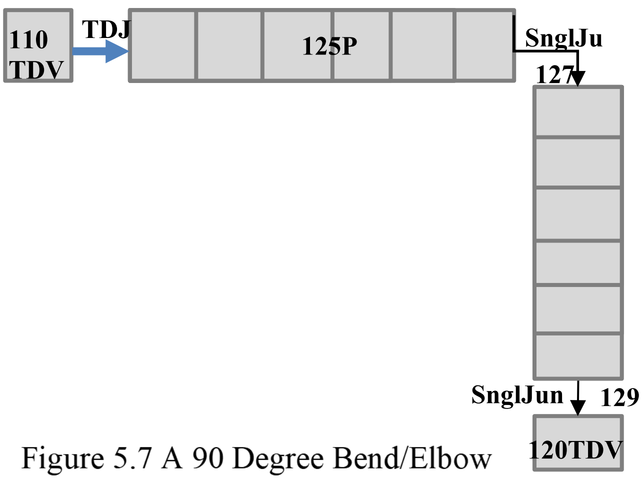


Figure 5.7 A 90 Degree Bend/Elbow

5.3 Heat Structure Nodalization: Single and Multi-pipes Flow

Heat structures represent the selected solid portions of the thermal-hydrodynamic system. Being solid, there is no flow, but the total system response depends on heat transferred between the structures and the fluid, and the temperature distributions in the structures are often important outputs of the simulation (E.g. fuel temperature). System components that are simulated by heat structures include fuel rods, pipe walls, core barrels, pressure vessels, and heat exchanger tubing.

The following are to be noted when modelling a Heat Structure

- Temperatures and heat transfer rates are computed from the one-dimensional form of the transient heat conduction equation.
- A heat structure is identified by a number, CCCG0NN. The subfield, CCC, is the heat structure number and is analogous to the hydrodynamic component number.
- Heat structures are usually closely associated with a hydrodynamic component, it is suggested that the hydrodynamic component number and the CCC portion of the attached heat structures be the same number. Since different heat structures can be attached to the same hydrodynamic component, such as fuel pins and a core barrel attached to a core volume, the G portion can be used to distinguish the different types of heat structures connected to a common fluid volume.
- The combined field, CCCG, is the heat structure-geometry number, and input data are organized by this heat structure geometry number.
- Up to 99 individual heat structures may be defined using the geometry described for the heat structure geometry number.

- The individual heat structures are numbered consecutively starting at 01; this number is the sub-field, NN, of the heat structure number.
- The heat structure input requirements are divided into input common to all heat structures with the heat structure geometry number, Cards 1CCCG000 through 1CCCG499, and input needed to uniquely define each heat structure, 1CCCG501 through 1CCCG999.

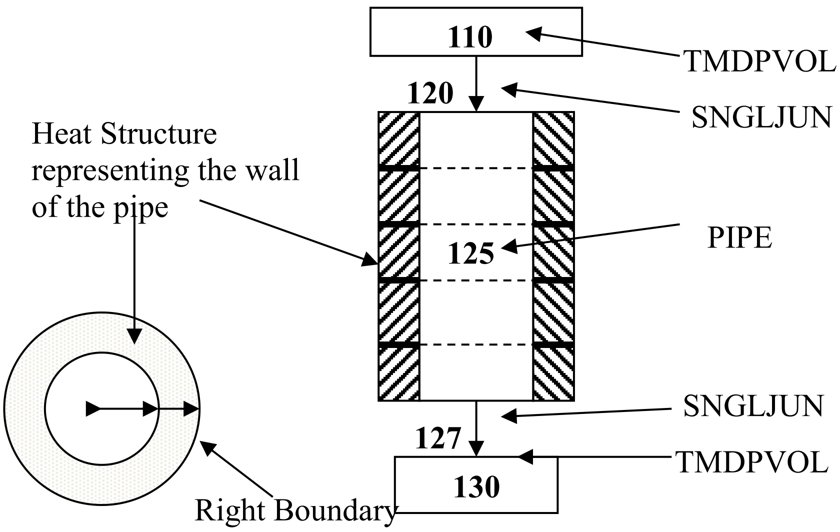


Figure 5.8 Schematic of Heat Structure on Pipe Component

5.3.1 CASE #5: Nodalization of Heat Structure on Single Pipes/Test sections

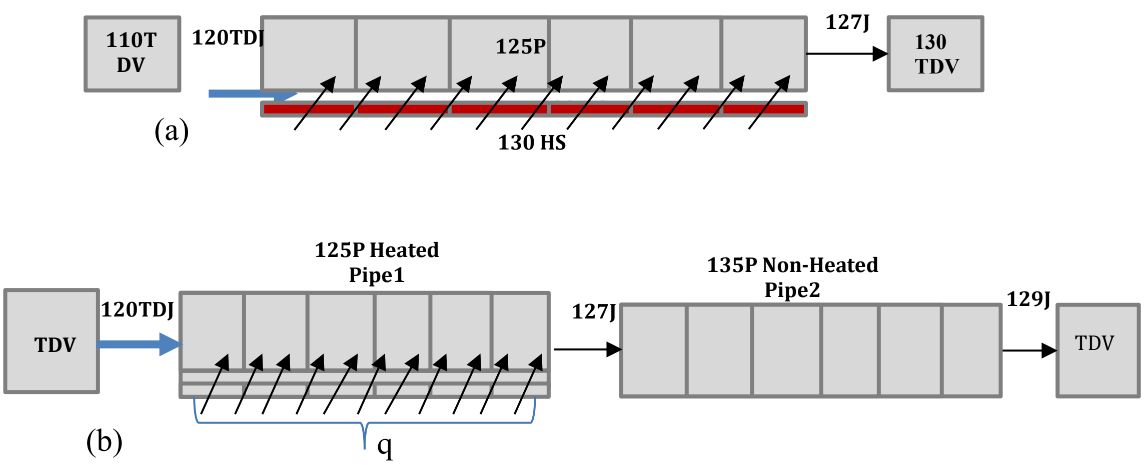


Figure 5.9 Schematic of Heat Structure on Single Pipe

The simulation input data and results for the nodalization of Figure 5.9 are presented in section 6.4.2 while the input file is presented in Appendix VI

5.3.2 CASE #6: Nodalization of Heat Structures on Multi-pipes

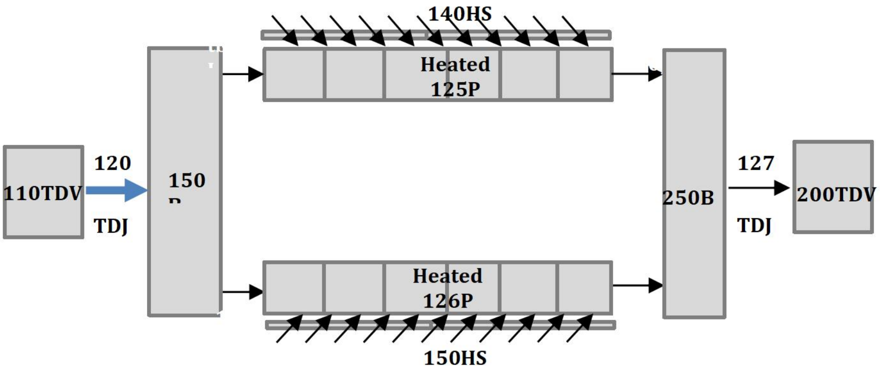


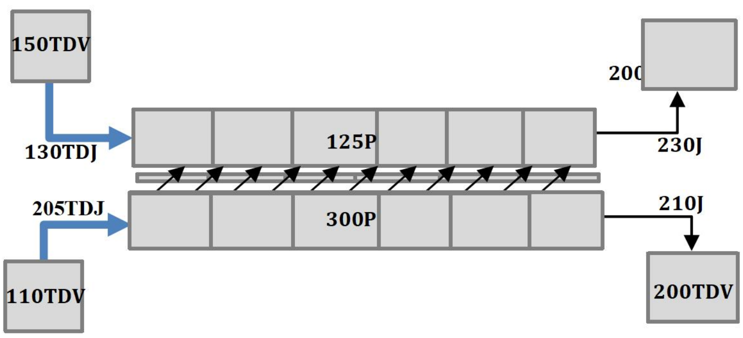
Figure 5.10 Heat Structure on Parallel Pipes

The simulation input data and results for the nodalization of Figure 5.10 are presented in section 6.4.2 while the input file is presented in Appendix VI

5.4 Heat Exchanger (HX) Nodalization

The simulation input data and results for the nodalization of Figure 5.11 are presented in section 6.5 while the input file is presented in Appendix VI and Appendix VII

5.4.1 CASE #7: Parallel Flow HX



(a)

5.4.2 CASE #8: Counter Flow HX

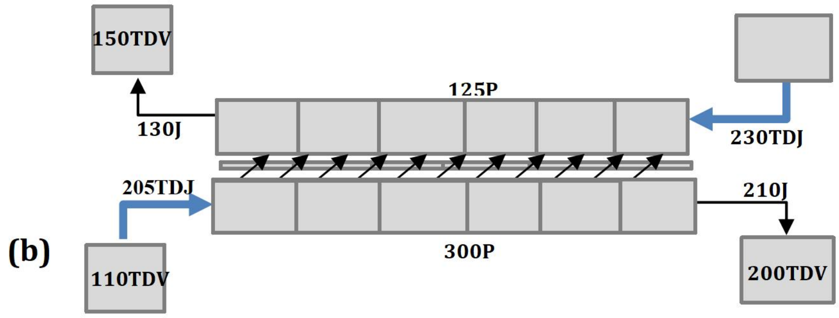


Figure 5.11 Heat Exchanger Nodalization (a) Parallel and (b) Counter Flow

5.5 Pump Nodalization

5.5.1 CASE #9: Pump Component Connecting 2 Pipes

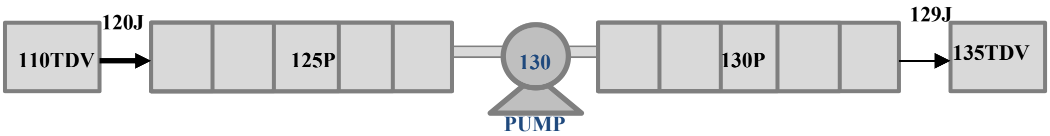


Figure 5.12 Series pipes connection with a Pump

The simulation input data and results for the nodalization of Figure 5.12 are presented in section 6.4.2 while the input file can be seen in Appendix VI

5.6 Loop Nodalization

The loop is simulated with both a pump component and with a TMJ, as such the nodaliation diagrams of Figure 5.13a and 5.13b represent loop with a pump component and with TMJ respectiviely.

5.6.1 CASE #10: With Pump

The simulation input data results for the nodalization of Figure 5.13a are presented in section 6.5.2 while the input file is presented in Appendix VI

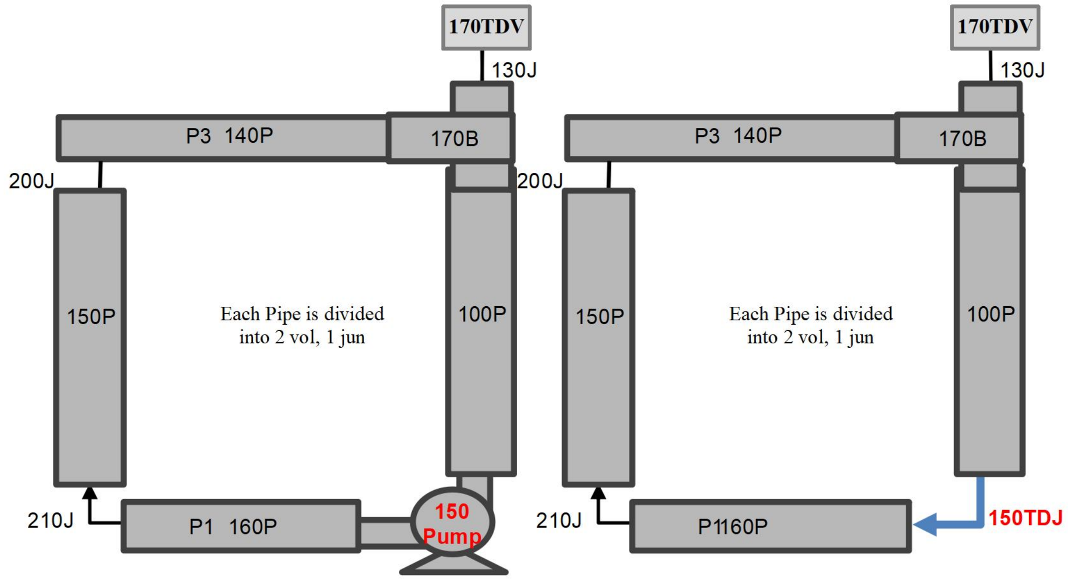


Figure 5.13 Loop Model with (a) Pump (b) Time Dependent Junction

5.6.1 CASE #11: With Time Dependent Junction

The simulation input data results for the nodalization of Figure 5.13b are presented in section 6.5.3 while the input file of Appendix VI is modify by replacing the Pump input lines with the TDJ commands as seen in Appendix VII

5.7 Code Development

5.7.1 Input File Description

Input file development involves creating a structured ASCII file containing all the necessary input data and parameters required to define the simulation setup, geometries, and conditions for running the reactor system analysis code. The input file typically uses and the nodalization and information including boundary conditions, initial conditions, material properties, control settings, and simulation options in a language suitable and acceptable by the system code. They are written in a specific format specified by the reactor system analysis code, often using markup languages or structured text formats.

The input file serves as a blueprint for the simulation, providing instructions to the code on how to perform the analysis and what scenarios to simulate. Input files should be structured and organized to facilitate ease of use, readability, and reproducibility of simulations. Careful attention should be paid to input parameter definitions, units, and formatting to ensure consistency and accuracy, thoroughly documented with clear descriptions of input parameters, assumptions, and modeling choices to facilitate code validation, verification, and peer review.

5.7.2 Guide on Input deck organization

The Control Format: Input is described in terms of input records or cards, where an input record or card is an 80-character record. With many terminals allowing only 80tcharacters per line, it is convenient to limit the data record to 72 characters so that the data and editor supplied line numbers fit on one line (eight columns for line number and separator, 72 columns of data). To avoid this, either request the editor to store line numbers starting at character position 81, put a terminating character before the line number, or don't store the line numbers. The line numbers, if saved, are listed in the output echo of the input data.

5.7.3 Data Deck Organization

RELAP5 input deck consists of at least one title card, optional comment cards, data cards, and a terminator card. The order of the title, data, and comment cards is not critical but it is recommended that for a base deck, the title card be first, followed by data cards in card number order. Comment cards are optional and should be used simply to explain or add additional information to the input or line.

a. **Title Card:** The remainder of the title card is printed as the second line of the first page following the list of input data.

b. **Comment Cards**: An asterisk (*) or a dollar sign ($) appearing as the first non blank character identifies the card as a comment card. Blank cards are treated as comment cards. Comment cards may be placed anywhere in the input deck except before continuation cards.

c. **Data Cards:** Data cards may contain varying numbers of fields that may be integer, real (floating point), or alphanumeric. Blanks preceding and following fields are ignored.

5.7.4 Basic Components Input

The RELAP5 input file development involves specifying the parameters and conditions that define the initial state and behavior of a system in a text software such as notepad, notepad++. The input file is written in LINES and each line is made up of items in that row denoted as WORDS (W). Each line must begin with a unique CARD Number.

Presented below is a general step by step line items/outline of a typical RELAP5 input file using one of our model/problem solved in this project: We will analyse each line of the code for better understand of the file components and structure.

Parameters requirement for input file

– Component dimension (Length, height and radius)

– hydrodynamic parameters (Flow rate)

– Material/s

– Boundary conditions

– Initial (P,T or T,Q) conditions

a. TITLE:

Title Card: Provides a title for the simulation. A title card must be entered for each problem and identified by an equal sign (=) as the first non-blank character.Identify and title the model of interest

```
= Simulating Flow in a Heated Horizontal Pipe
*This model investigates the pressure drop, mass flow rate and temperature
*difference along a uniformly heated pipe.
*--------------------------------------------------------------------
```

b. INITIAL CONDITIONS (Optional):

It is advisable to list the initial and boundary conditions to guide the input of parameters in the deck as shown below. Don't forget to comment them as they are not the executable line for the simulation.

```
*-----------------------------------------------------------------------
*************Initial conditions*********************************
*-----------------------------------------------------------------------
*Mass flow rate= 6kg/s
*Pipe diameter = 5cm = 0.05m
*Pipe Length   = 6cm  = 6m
*Inlet temp    = 323K
*Pipe roughness= 0.00001(denote a smooth pipe with min friction loss effect)
*-----------------------------------------------------------------------
```

c. PROBLEM TYPE AND OPTION

```
*-----------------------------------------------------------
*************Problem Time and Option****************
*-----------------------------------------------------------
*crdno
 100  new  transnt
 102  si si
```

Card 100: Word 1 (W1) is Problem type, user is to choose the problem type either, NEW, RESTART, PLOT, IN-COND, STRIP, or CMPCOMS. NEW specifies a new simulation problem while RESTART specifies file. These word is chosen according to the specific problem type with the NEW and RESTART mostly used.

Card 102: Problem Option (W2). Enter either STDY-ST or TRANSNT to specify the type of simulation.

d. CPU TIME REMAINING

```
*-----------------------------------------------------------------------
***********CPU Time Remaining********************
*-----------------------------------------------------------------------
*crdno
105   5.0     6.0   500.0
110   air
115   1.0
```

Card 105: This card controls termination of transient advancement based on the CPU time remaining for the job. The user can therefore use the card to limit the computational time to some pre-determined amount even if the entire transient has not yet been finished.

W1= CPU remaining limit 1 (s).

W2 =CPU remaining limit 2 (s).

W3= CPU time allocated (s) (optional).

Card 110 This card is required for problems having non-condensible gas. Pick one or up to a maximum of 5 from any of the following non-condensible gas types: argon, helium, hydrogen, nitrogen, xenon, krypton, air or sf6. Cannot be entered on a RESTART problem

Card 115 is a function of Card 110. It defines the mass fraction of the number of gases entered in card 110. if one gas then 1.0 is entered as in the sample file.

e. TIME STEP CONTROL CARDS

```
*----------------------------------------------------------------------
*********************Time Step Control****************
*----------------------------------------------------------------------
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     2000.0   1.0e-6    2.0     3          1         50      2000
```

Card 200: Initial Time Value (optional): If not entered, the simulation time start at zero.

Card 201: Time Step Control

W1= Time end for this set (s).

W2= Minimum time step (s). This quantity should be a positive number < 1.0E-6.

W3= Maximum time step (s). This is the requested time step.

W4=control option: This word is a conditional word in the format of ssdtt. Used to control the printed content of the major edits.

W5= Minor edit and plot frequency. The number of maximum or requested time advances per minor edit and write of plot information.

W6= Major edit frequency. This is the number of requested time advances per major edit.

W7= Restart frequency. This is the number of requested time advances per restart information

f. HYDRODYNAMIC COMPONENT

Describes the components in the system (pipes, pumps, heat structures, etc.) and their initial conditions. The hydrodynamic card numbers is in the form CCCXXNN and are divided into fields, where CCC is the component number (the component numbers go with the nodalization component number), XX is the card type, and NN is the card number within type. Referencing the reference problem's nodalization diagram presented in Section, the number of components identify includes,

- The Source Volume (Time Dependent Volume),
- Connector 1 (Time Dependent Junction)
- The Pipe
- Heat Structure
- Connector 2 (Single Junction)
- Sink Volume (Time Dependent Volume 2)
- Plot Request and
- End

g. THE SOURCE VOLUME.

It is used wherever flow can enter or leave a hydrodynamic system.

```
*----------------------------------------------------------------------
*******Source Volume Properties*******
*----------------------------------------------------------------------
1100000  "inlet"  tmdpvol
*        fa(sq.m)  length(m)  vol(m3)  azi  vert   dz   rough hyd.d flags
1100101  0.00196     1.0        0.0    0.0   0.0   0.0   0.0   0.0  0000000
*
*        ebt(fluid, boron, initial thermodynamic state)
1100200  101
*
*        time   temp(K)  stc.qlty
1100201  0.0    323.0    0.00001
```

Card 1110000: Present the component Name (limit is eight character in W1 and the component type in W2 indicated by TMDPVOL

Card 1100101: Presents the Volumes properties as labeled. Flow area of inlet volume is selected the same as the flow area of pipe. After initiation the program sets the the rest values to zero since they are only required to initiate the flow into the actual pipe component. Refer to figure for the Azimuthal and vert orientation.

Card 1100200 is the data control that sets the thermodynamic state for fluid type, if boron is present or not, initial thermodynamic state of the TMDPVOL using the control term “εbt”. The digit ε specifies the fluid. ε = 0 is the default fluid, ε = 1 specifies H2O, and ε = 2 specifies D2O. The digit b specifies whether boron is present or not. The integer specification, b = 0, specifies that the volume fluid does not contain boron; b = 1 mean there is boron concentration in the liquid water. The input t specifies how the words of the time-dependent volume data in Cards CCC0201 through CCC0209 are to be used to determine the initial thermodynamic state. Entering t equal to 0 through 3 specifies one component (steam/water). Entering t equal to 4 through 6 allows the specification of two components (steam/water and noncondensible gas).

Card 1100201: W1 is the search variable such as time. W2 and W3 are dependent of the control variable in ‘t’ in 200. If t=1 in as previously mention in card 1100200, then W2 and W3 will be the time dependent data of pressure and temperature, if t=3 the TD data for W2 and W3 will be pressure and temperature.

h. TIME-DEPENDENT JUNCTION:

This component is the Inlet connector Component indicated by TMDPJUN numbered as CCC000000. In the expanded format (for the most recent versions of RELAP for example), the connection code is CCCVV000N, where CCC is the component number, VV is the volume number, and N indicates the face number. A nonzero N specifies the expanded format, while a zero can be placed to indicate original RELAP input formats. The number N equal to 1 and 2 specifies the inlet and outlet faces/volumes, respectively, or the volume’s (x) coordinate direction. The number N equal to 3 through 6 specifies crossflow.

```
*---------------------------------------------------------------
*******Inlet Time Dependent Junction Properties*******
*---------------------------------------------------------------
*hydro       name          type
1200000     "inlet"        tmdpjun
*hydro      from vol      to  vol    f.area
1200101     110010002     125010001  0.00196
*hydro        vel/flow         trip
1200200         1               0
*hydro      search(time)    liq.mlow   vap.mflow   int.face.vel
1200201       100.0           6.0         0.0          0.0
```

Card 1200101 refer component name (W1) and type (W2) indicated by TMDPJUN

Card 1200101 refer to Time-Dependent Junction Geometry

W1 refers to the component and volume from which the junction coordinate direction originates.

W2 refers to the component and vol to which the junction co

W3 refers to Junction flow area (m2 , ft2). If zero, the area is set to the minimum flow area of the TMDPVOL

Card 1200200: This card is optional. If this card is missing, W2 and W3 of cards CCC0201 are velocities.

W1 is Control word. If = 0, W2 and W3 of cards CCC0201 are velocities. If =1, they are replaced with mass flows as in the problem we modelling. In both cases, W4 is interface velocity and should be zero.

Card 1200201: W1 consists of a set of search variable (e.g., time) followed by W2 the required data indicated by control Word 1 on Card 1200200

i. **THE PIPE COMPONENT.**

String of volumes with interior connecting junctions: Presented in the format Card CCC0001 with CCC being the component number,

Card 1250000: As we already know, W1 is the component name and W2 is the component type.

Card 1250001: W1 = number of vol (recall in the nodalization, the component is divided into equal vols. This number must be greater than zero and less than 100.

Card 1250101: W1 = the Volumetric flow area while W2 = the volume number

Card 1250301: W1= the Pipe or annulus volume length (m, ft) and W2 = the volume number

Card 1250501: W1 = the Azimuthal angle in degrees. Refer to Figure. W2 = Volume number.

Card 1250601: W1 = Vertical angle in degrees. Again refer to Figure. W2 = Volume number. 90.0 degrees means that the inlet is starting from the top to bottom and inverse for -90.

Card 1250801: W1= The Wall roughness (m, ft). if not applicable enter a very low value as in the current problem. W2= Hydraulic diameter (m, ft) refer to the theory section for the formular. W3=Volume number.

Card 1250901: Presents thee energy loss coefficients.

W1= Forward energy loss,

W2=Reverse energy loss coefficient,

W3 = Junction number (recall the number of junction from the nodalization diagram

Card 1251001: W1 = Volume control flags and W2 = the Vol number.

Card 1251101: W1= Junction control flags and W2 = Junction number.

Card 1251201: W1 = Control word in the packed form of ebt same as in Card 1100200

W2-W6 are quantities based on W1. Five quantities must be entered, and zeros for unused quantities as seen in the working code.

Card 1251300: W1 is a control word as explain in Card 1200200

Card 1251301: W1= Initial liquid velocity or mass flow. Based the choice in card 1200200, Mass flow is appropriate choice

W2= Initial vapor mass flow

W3= Interface velocity. Enter zero.

W4=Junction number.

j. HEAT STRUCTURE

These cards are used in NEW and RESTART type problems and are required only if heat structures are described. The heat structure number is represented in the format 1CCCG000 having different fields as: CCC representing the heat structure number(same as the pipe = 100 in this case, to indicate to other users its association with pipe #100), it is suggested that the heat structures be numbered same as the hydrodymanic volume (pipe(it is attached to as will be noticed in the problem we are considering. G is a geometry number. The combination CCCG is a heat structure geometry combination referenced in the heat structure input data. X is the card type and NN is the card number within a card type.

Card 11000000: W1= Number of axial heat structures with this geometry.

W2= Number of radial mesh points for this geometry.

W3= Geometry type. 1 for a rectangular, 2 for cylindrical, and 3 for spherical. Our current model is a cylindrical reason why 2 is selected.

W4=5 initiate Steady-state initialization flag. Zero is selected as the desired initial condition temperatures are entered on input cards 1CCCG401 through 1CCCG499; use one if a steady-state initial condition temperatures are to be calculated by the code.

W5=Left boundary coordinate (m, ft).

Card 11000100: W1=Mesh location flag: 0 means the geometry data including mesh interval data, composition data, and source distribution data are inputed. Non zero means the the information is taken from the heat structure geometry.

W2=Mesh format flag. This word is needed only if Word 1 is zero.

Card 11000101: These cards are required if Word 1 of Card 1CCCG100 is zero in the following format,

```
*-----------------------------------------------------------------------
**********Heat Structure added to Pipe 100***************
*=======================================================================
*        NVOL NP Geo Stdy In.Rad.
11000000  2   5  2   0  0.025
*       M.Loc Form
11000100  0    1
11000101  4  0.05
11000201  100  4
*         Power curve
11000301  1.0    4*flat power profile assumed
*       in.temp. NP
11000401  350.0 5
*           left    incr.  typ area    NH
11000501  100010000 010000  1  1  2.0  2
*           right    incr.  typ area     NH
11000601      0      0       0   1   2.0  2
*                power peaking factor
*         src.table  src.mltp.   mod.multp. NH
11000701  100        1.0         1.0    0.0  2
*        ht.hyd.d. heated length  grid         boil.fctr NH
11000801   0.0      10.0 10.0  0.0 0.0  0.0   0.0    1.0   2

*  stainless steel
20110000  tbl/fctn  1  1  *  stainless steel
*
*           temperature   thermal conductivity
20110001        273.15             42.5578
20110002        1199.8166667       82.3129
*
*           temperature   heat capacity
20110051        273.1500          3827850.0
20110052        422.0389          3962160.0
20110053        477.5944          4096471.0
20110054        533.15            4230782.0
20110055        588.7056          4331514.0
20110056        644.2611          4432113.0
20110057        699.8167          4499403.0
20110058        810.9278          4633713.0
20110059        1366.483          5372421.0
20210000 power
20210001  0.0  0.0
20210002  0.0  20000.0
```

W1= Number of intervals.

W2= Right coordinate (m, ft).

Card 11000201: These cards are enter only if W1 of Card 1CCCG100 is zero

W1= Composition number

W2=Interval number

Card 11000301: Same condition as above.

W1=Source value.

W2= Mesh interval number.

Card 11000401: W1=Temperature (K, °F).W2= Mesh point number

Card 11000501: Defines the Left Boundary Condition for the heat structure

W1 = Boundary volume number of the form CCCNN000F used to specify the sink temperature. If zero, no volume or general table is associated with the left surface of this heat structure. We already defined the format aside F. If F is 2 or 1, volume coordinate associated values are taken from the y- or z-axes, respectively.These numbers define the flow direction parallel with tube bundles

W2= specify the increment (normally 10000) is added to Word 1, which results in the hydraulic cell number associated wiW3= Boundary condition type. If 0, it means zero temperature gradient is used at the boundary. But 1 indicates that a convective boundary condition where the heat transfer coefficient obtained from Heat Transfer Package 1 is used.

Card 110006011 are the same as Cards 1CCCG501 through 1CCCG599, except for the right boundary. The left and right surface areas must be compatible with the geometry.

Card 11000701: W1 defines the source type. If zero, no source is used. If a positive number less than 1000, power from the general table starting from line 116 in our sample input deck is used with this number as the power source. If above 1000, the number has the form 100t and the source is taken from a reactor kinetics calculation.The field t = 0 specifies total reactor power, t = 1 specifies total decay power, t = 2 specifies fission power, t = 3 specifie sfission product decay power, and t

= 4 specifies actinide decay power. If 10001 through 19999, the source is the control variable whose number is this quantity minus 10000.

W2 refers to the Internal source multiplier. These values are helps to obtain the total power generated in this heat structure by multiplying it by the power in the general table.

W3 = Direct moderator heating multiplier for left boundary volume, 1 in this and most cases unless moderator heating in a reactor simulation is needed.

W4 is same as in W3 but for the right boundary.

W5 denotes the Heat structure number.

Card 11000801 is a nine-word format that is required if or when the left boundary shares energy with the is expected to share energy with the left hand fluid volume.

W1 denotes the Heat transfer hydraulic diameter $\left(\frac{\text{4XFlowArea}}{\text{Heated perimeter}}\right)$. If 0.0, the volume hydraulic diameter is used.

W2 is the Heated length forward (m, ft). Distance is from the heated inlet to the center of this slab. This quantity will be used when the liquid volume velocity is positive or zero. This is used to get the hydraulic entrance length effect. This is used only for the CHF correlation. It must be > 0. To ignore the length effect, put in a large number (i.e., ≤ 10.0).

W3(R) Heated length reverse (m, ft). Distance is from the heated outlet to the center of this slab. This is used only for some of the CHF correlations (if invoked through other user inputs). It must be > 0. To ignore the length effect, put in a large number (i.e., ≤ 10.0).

W4 - W7 is the Grid (spacer length and loss coefficient) parameters in used only for the CHF calculation.

W8(R) Local boiling factor. 1.0 if there is no power source in the heat structure meaning liquid is subcooled and void is zero. This number is greater than 0.0.

W9 is the Heat structure number.

Material Property: This section specifies the Heat structure thermal conductivity and heat capacity per time. In this model, the information is inserted in the code from line 147 - 169 of our input deck being analysed.

Power Distribution Table: This section (Lines 170 - 172) specifies the power distribution. W1 = Time while W2 = Power at that specific time of run. It could go beyond line 172 depending on simulation details and complexity of the power input table..

(k) SINGLE JUNCTION

The single Junction is a basic connection for hydraulic system and is often used with PIPE components. It's a string of volumes with interior connecting junctions. Cards are similar to the Time-deendent Junction as previously discussed, except that user should specify the "from" volume and the "to"

```
*----------------------------------------------------------------------
************Outlet Single Junction*******
*----------------------------------------------------------------------
1270000   "sngljuno"   sngljun
*         from vol    to vol      fa   f.loss   r.loss     flag
1270101   125120002   130010001   0.0  0.0      0.0        0000100
*       flag   liq.mflow   vap.mflow    int.vel.
1270201   1      6.0          0.0          0.0
```

Card 1200000 refer component name (W1) indicated by SNGLJUN and type (W2) same as in TMDPVOL

Card 1270101 as explained in TMDJUN

Card 1270201 refer component name (W1) and type (W2) same as in TMDPVOL

Card 1270101: W1 refers to the Search variable (e.g., time).

W2 refers to the Liquid velocity or mass flow depending on the control W 1

W3 is the Vapor velocity or mass flow.

W4 is the Interface velocity (m/s, ft/s). usually zero

(l) SINK VOLUME

Sink volume is also a time dependent Volume with same Card details as describe earlier. The difference is that this serves as the sink/outlet fixed parameter/boundary condition (Line 96, Card 1300101) and also the component number should be differentiated from other components. In addition, note in this example, that Card 1300200 condition in W1 is changed to "102" to assign the pressure and quality at the outlet of the pipe, as well as the temperature distribution due to the attached heat structure (room temperature in this case).

```
89  *-----------------------------------------------------------------
90  ***********Sink Volume Properties*******
91  *-----------------------------------------------------------------
92  1300000  "sink"   tmdpvol
93  *        fa    l     vol    azi    vert     dz    rough hyd d flags
94  1300101  5.0  1.0    0.0    0.0    0.0      0.0   0.0   0.0  0000000
95  *Time dependent data to be pressure and temperature
96  1300200  102
97  *        time    pressure(Pa)   Quality
98  1300201  0.0       500000.0      0.00001
```

Card 1300201 defines the exit boundary condition,

W1 is the search variable such as time.

W2 and 3 are dependent of the control variable in Card 1100200 of the source volume component. If t=1 in as previously mention in card 1100200, then W2 and W3 will be the time dependent data of pressure and temperature, if t=2, W2 and W3 will be

Pressure and Quality while t=3 amounts to W2 and W3 being flowrate and Temperature.

(m) PLOT REQUEST

The plot request information for each graphical output is made on cards 20300NNM. Where NN is the graph number and may range from 1 through 39, but the total number of graphs must be less than 40. M specify the trend number in the plots. The information for each graph consists of one or more sets of three Was shown un the sample file.. One set for each variable to be shown on the graph. The sets may be entered on one or more cards, using m ranging from 0 through 4.

Cards number are in the form 20300NNM: W1 denotes the plot number and the parameter trend added to the plots.

W2 is the Parameter part of variable name.

W3 is the Axis number and scale indicator

```
*================================
* plot requests
*================================
20300011  mflowj   100010000 1
20300012  mflowj   150010000 1
20300013  mflowj   140010000 1
20300014  mflowj   160010000 1
*
20300031  httemp   100000201 1
20300032  httemp   150000201 1
*
20300021       p   100020000 1
20300022       p   150020000 1
20300023       p   140020000 1
20300024       p   160020000 1
*
20300041   tempf   100020000 1
20300042   tempf   150020000 1
20300043   tempf   140020000 1
20300044   tempf   160020000 1
```

The final line of the code is the ending command which is written as;

```
. End of input .
```

(n) BRANCH COMPONENT

Branch is simply a single volume having more than one junction connections on one side. The BRANCH component is used in other problem solved in this project. The input deck for the BRANCH component as used here is analyse below. This could be 1 inlet with 2 outlet or vice versa. the Card numbers are in the forms of CCC0001 where CCC is the component number. More than one junction may be connected to the inlet or outlet. If an end has no junctions, that end is considered a closed end.

```
*-----------------------------------------------------------------------
*                              component 120
*=======================================================================
*crdno        name          type
1200000      bndbrnch      branch
*crdno   no.juns.    ctl
1200001        3       1
*crdno   area    length volume h-ang  v-ang    delz     rough    dhy    ctl
1200101  0.0196  2.0     0.0 180.0   0.0      0.0      0.0      0.0     0
*crdno   ctl    pressure    temp
1200200   3       1.5+6     350.0
*crdno   from          to            area     floss     rloss     flag
1201101  100020002     120010001       0         0         0       100
1202101  120010002     140010001       0         0         0       100
1203101  120010006     130010001       0         0         0       100
*crdno   flowf    flowg    velj
1201201  0.0      0.0        0
```

Card 1200000: W1 defines the component name and W2 states the component type

Card 1200001: W1 defines the Number of Junctions, “nj”.

Card 1200101 defines the components geometry

W1 is the Volume flow area in the x-coordinate direction (m2, ft2).

W2 is the Length of volume in the x-coordinate direction (m, ft).

W3 is the Volume of volume, W3 = W1·W2. If one of the quantities is zero, it will be computed from the other two.

W4 is the Azimuthal angle. The absolute value of this angle must be ≥ 360degrees.

W5 is the Inclination angle. The absolute value of this angle must be ≥ 90 degrees. The angle 0 degrees is horizontal, and positive angles have an upward inclination.

W6 is the Elevation change (m, ft). A positive value is an increase in elevation.

W7 is the Wall roughness (m, ft).

W8 is the Hydraulic diameter (m, ft).

W9 is a control word as define in Card

Card 1200200: W1is a Control word in the format εbt as defined in Card.

W2-W6 are Quantities as described under Word 1 which is depending on the control word. (Refer to Card)

Card 1201101 is the "from" card and refers to the component where to the connection junction originate and where it connect to the Branch

Card 1202101 and Card 1203101are the "to" cards and refers to the component where the Branch junction is connected to.

(o) PUMP COMPONENT

Pump components provide the direct motive force for fluid movement in a system and the head provided directly affects the flow achieved through the fuel assemblies, and can affect other phenomena such as cavitation and/or flow runout. The pump characteristic curve is a vital input for system simulations, as such the code user should understand how the pump characteristic curve works as described in Section 2 of Chapter 3.

```
*-----------------------------------------------------------------------
*******Pump Properties*******
*      westinghouse homologous pump curves
*      flow area = 0.07 sqm, volume = 0.14 cum
*      ratings:  speed = 369 rpm                flow = 0.81 cum/sec
*                 head = 27.5 m               torque = 500.0 nm
*               mom.i. = 1.43 kg*sqm         density = 614.0 kg/cum
*      actual speed = 369 rpm
*crdno        name         type
2000000      "pump"         pump
*crdno     area       length volume h-ang   v-ang   delz   ctl
2000101    0.00196      2.0      0     0     0.0    0.0     0
*crdno     from           area     floss     rloss     flag
2000108    100010002        0         0         0        0
*crdno     to             area     floss     rloss     flag
2000109    300010001        0         0         0        0
*crdno     ctl        temp    stc.qlty
2000200    101       450.0    0.000001
*crdno     ctl      flowf    flowg    velj
2000201      1      131.0      0.0       0
2000202      1      131.0      0.0       0
*crdno      id     2faz    2fazd    tork    pvel  ptrip   rvrs
2000301     -2      -1       -3      -1      -1     0       0
*crdno     rpvel  initv   rflo    rhead   rtork   momi    rdens
2000302    369.0    1.0   0.81     27.5   500.0   1.43    614.0
*crdno     rmotk   tf2    tf0     tf1     tf3
2000303        0  153.0  0.003   14.5       0
*-----------------------------------------------------------------------
```

Other Used Components

There are other components that may be required to model an entire nuclear power plant system. These includes the Separator, Jetmixer, Turbine, ECC Mixer, or Vessel components. The all are begin in the format CCCVV000N, where CCC is the component number, VV is the volume number (always 01 for these components), and N is the face number. Given its importance in generic modelling of systems the Pump component is briefly described below, while the other components are more advanced and are not covered in this Workbook.

Chapter 6

Simulation Results and Sensitivity Analysis

The chapter presents the simulation result of each scenario discussed in the previous chapter and included in the Workbook Based on the each input model, repeated simulations were carried out to extensively evaluate the result and its validity, offering insights into its accuracy, reliability, and applicability to real-world scenarios. The sensitivity analysis is based on varying the input parameters and a comparative analysis carried out with results directly extracted from RELAP and ASYST simulation outputs to demonstrated what to expect to new user.

New users can also post-process the outputs to extract the code output file into a CSV file, then using any of python, MS Excel and other data processing program, plot the graphical results. Let us start off with this exercise and work around it with varying parameters

6.1 1-D Single and Connected Pipes Flow Cases

Sample question: Water at 15 ℃ (ρ = 999.1 $kg/m^{3,\ a}$ and μ = 1.138x10^{-3} $kg/m.s$) is flowing steadily in a 30 m long and 6 cm diameter horizontal pipe made of stainless steel at a rate of 10 L/s. Determine the pressure drop across the pipe.[75]

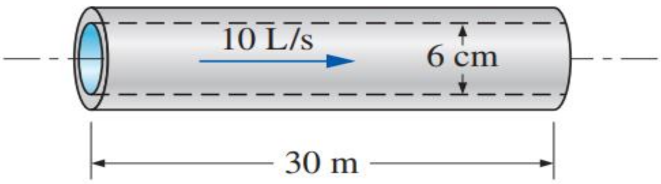


Figure 6.1 Schematic of exercise 8-32 problem adopted from [75]

6.1.1 CASE #1: Single Pipe Flow (1-D)

Case 1.1: Pressure difference in a flow of single horizontal pipe

This case is the solution to the above exercise adopted from [75] used as reference for the 1-D single pipe flow and its sensitivity analysis . In this case we provide an example of boundary condition inputs that represent known flow rate, pipe's dimensions, temperature and unknown pressure differentials.. The Nodalization diagram is given in Figure 4.1 and sample input file for the reference case is provided Appendix I

*---

********** **Input Data**********

*---

*Mass flow rate = 10.0 kg/s

*Pipe diameter=6cm = 0.06 m

*Length=30 m

*Roughness = 0.0018

*Inlet temp = 288 K

Pressure drop is calculated by the code and presented in Figure 6.2 while the plot of set flow rate is present in Figure 6.3. Analytically validation is further carried out in section 6.1.1.2

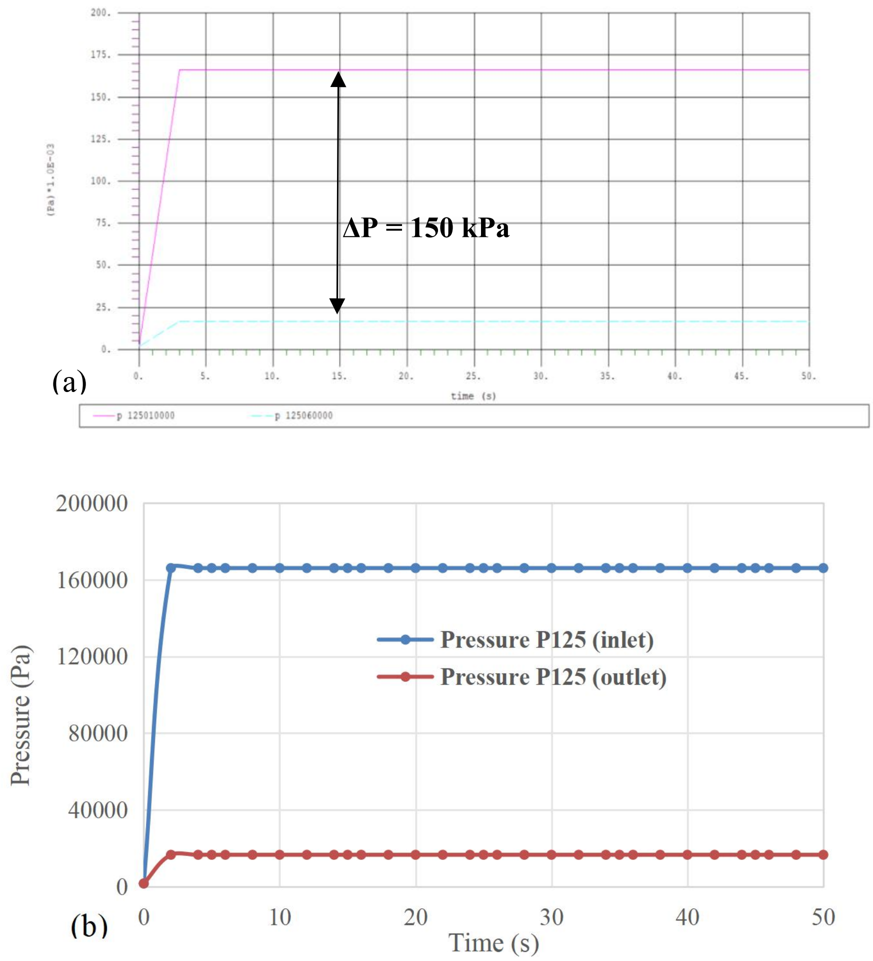


Figure 6.2 Plot of pressure at both end of single horizontal pipe (a) Code output (b) Processed data

The RELAP5 result is then post-process using the strip file in Figure 6.3 to extract the parameter of interest in CSV format and then re-plot using MS Excel (lower graph). The exercise of post-processing and verifying the results to ensure outputs were correctly extracted is a good skill for new users and hence is included in the Workbook.

```
= Title
100 strip csv
103 0
*
1001 tempf 125010000
1002 tempf 125060000
1003 mflowj 125010000
1004 mflowj 125060000
1005 p 125010000
1006 p 125060000
. End of input.
```

Figure 6.3 Strip file input for single pipe

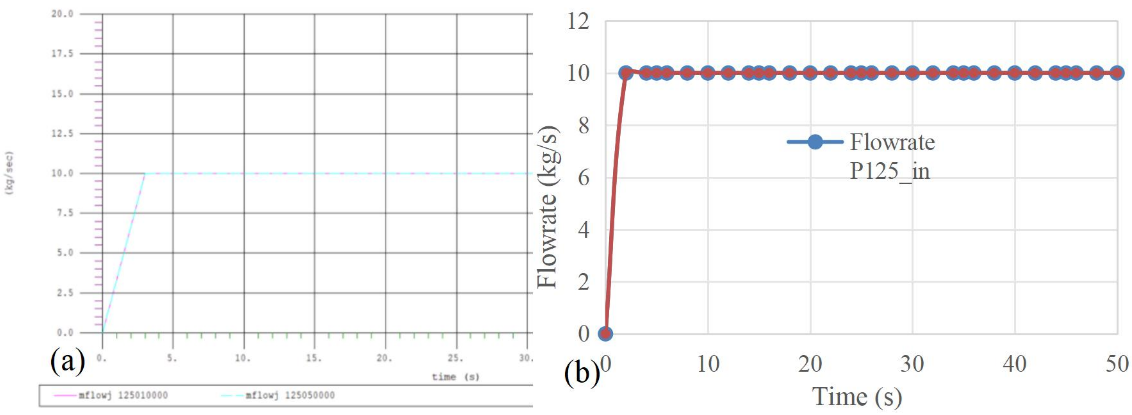


Figure 6.4 Graphs of set Flow rate (a) RELAP Plot (b) Processed Data

From both results (Code output and post-processed plots), pressure drop was experienced as the fluid flow from the pipe inlet to its outlet which is expected due to pipes length and internal friction between the fluid and pipe surface roughness. It is also expected that as the roughness increases the pressure difference will increase. Another factor that will lead to increase pressure drop will be an increased pipe length as presented below for accuracy and dependability of our model simulations.

6.1.1.1 Analytical Verification

Based on the given parameter stated above in case 1a Simulation, we recall and employ the Equation 4.6 to solve for the Pressure Drop, ΔP as defined in section 4.1

$$\Delta P = \left(f\frac{L}{D}\right)\left(\rho\frac{\nu^2}{2}\right)$$

Given, $Q = 10\ \mathrm{l/s} = 0.01\ \mathrm{m^3/s}$ and $D = 6\ \mathrm{cm} = 0.06\ \mathrm{m}$,

$$r = \frac{D}{2} = \frac{0.06}{2} = 0.03\ \mathrm{m}$$

$$\nu = \frac{Q}{A} = \frac{Q}{\pi r^2}$$

$$\nu = \frac{0.01}{0.002827} = 3.54\ \mathrm{m/s}$$

Since f is a function of the Renold Number, Re, as defined in equation 4. we will first solve for Re using $Re = \frac{\rho \nu D}{\mu}$

$$\rho = 999.1\ \mathrm{kg/m^3}\ ,\ \mu = 1.139\ \mathrm{X}\ 10^{-3}$$

$$Re = \frac{999.1 \times 3.54 \times 0.06}{1.138 \times 10^{-3}} = 186475$$

$$Re = 1.86 \times 10^5$$

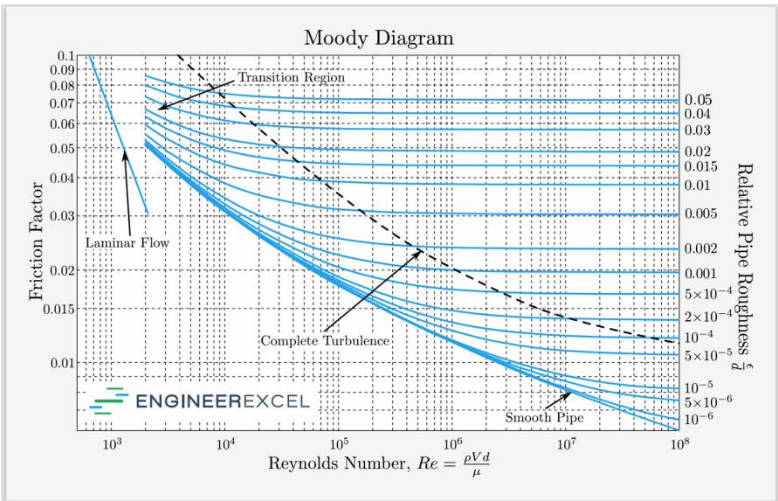


Figure 6.5 The Moody Diagram adopted from [76]

The above Re denotes a fully developed turbulent flow. Referencing the Moody diagram in Figure 6.10 we can deduce the friction factor, *f,* for this range of Re and Pipe'sroughness s;

$$f \approx 0.049$$

Substitutingl parameters, we have;

$$\Delta P = \left(\frac{0.049 \text{ x } 30}{0.06}\right)\left(\frac{999.1 \text{ x } 3.54^2}{2}\right)$$

$$6260.16078$$

Thus at $f = 0.049$,

$\Delta P = 153373.93 \approx$ 153 kPa

The pressure drop between the inlet and outlet of the pipe is calculated to be about 1563 kPa. Comparing with the simulated pressure drop where P_inlet amounts to 8590 Pa and P_outlet is 8760 Pa, we can deduce that the pressure difference will be

$$\Delta P = P_{outlet} - P_{inlet}$$

ΔP= 167 - 15 kPa

$\Delta P = 152$ kPa

In summary, simulated result shows that the ΔP_simulated = 152 kPa while the analytical calculation agrees closely at ΔP_analytical = 153 kPa.

6.1.1.2 Sensitivity Analysis

Case 1.1 Increased Pipe Roughness (0.0018 - 0.0058)

The Nodalization diagram for this sensitivity analysis case is same as Figure 4.1 and only Word 1 of CARD 1250801 in the sample input file for case #1 APPENDIX I change from 0.0018 to 0.0058 to reflect the change

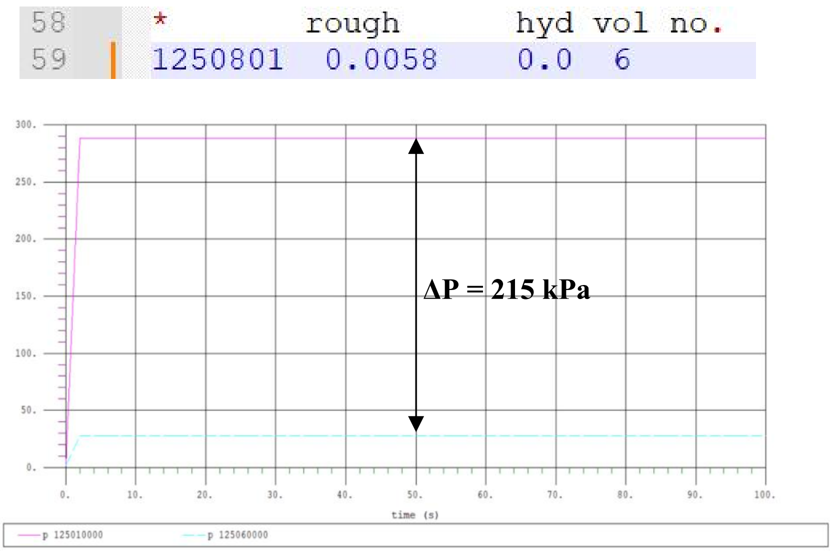


Figure 6.6 Pressure difference for rougher pipe

As the pipe roughness was increased from 0.0018 to 0.058 keeping other parameters constant, the equivalent pressure head sufficient to push the desired flow rate from the pipe inlet to outlet as been determined. This pressure difference is expected to be higher due to the increased roughness. From the simulation results of Figure 6.4 we can confirm that the pressure is 9.24 MPa at inlet compare to the 8.76MPa at the inlet of the smoother pipe of figure 6.1 and for the outlets we observed an increase of 8.64 MPa for rough pipe to 8.59 MPa of the smooth pipe.

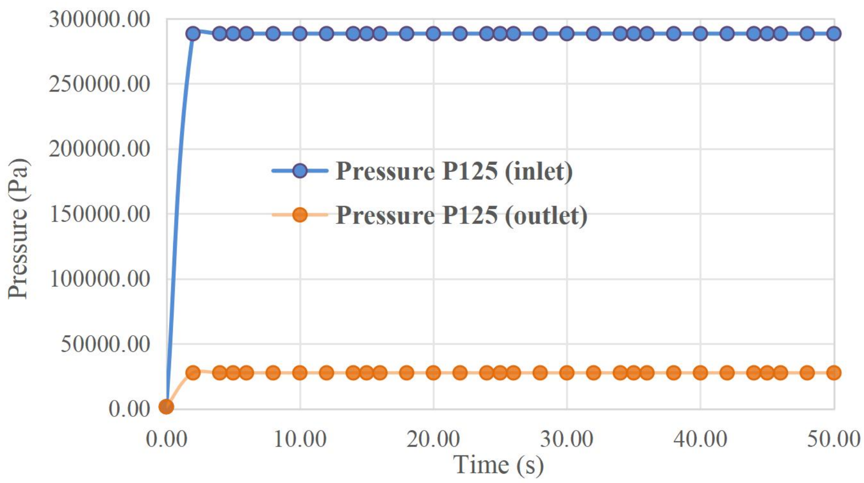


Figure 6.7 Processed pressure difference for rougher pipe

Case 1.2 Increased Length on Rough Pipe by 6m

The Nodalization diagram for this sensitivity analysis case is the same as Figure 4.1 but Word 1 of CARD 1250301 in the sample input file for case #1 provided in APPENDIX I was change from 5.0 TO 6.0 to reflect the change.

```
52   *          lengh   vol no.
53   1250301    6.0   6
```

The expectation of an elongated pipe is to have additional pressure drop as compared to the equivalent case with a shorter length. The figure below shows that for a 36m long case there is a higher pressure drop of 317 kPa compared to the 215 kPa pressure drop for30m that was observed in previous case. Additional analytical validation is further carried out next.

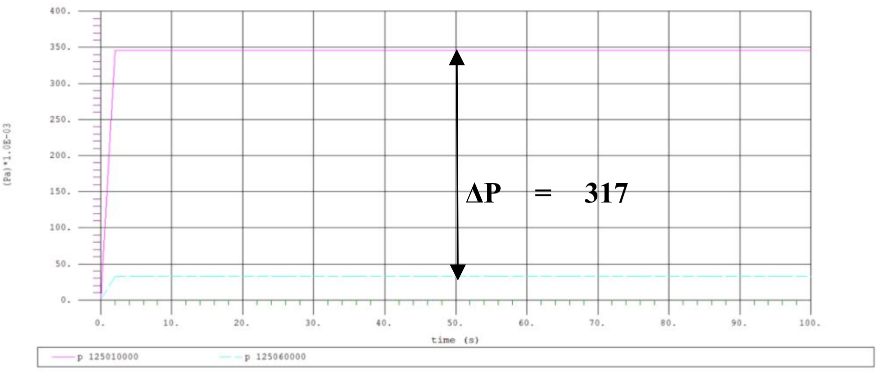


Figure 6.8 Pressure difference for the Increased length on

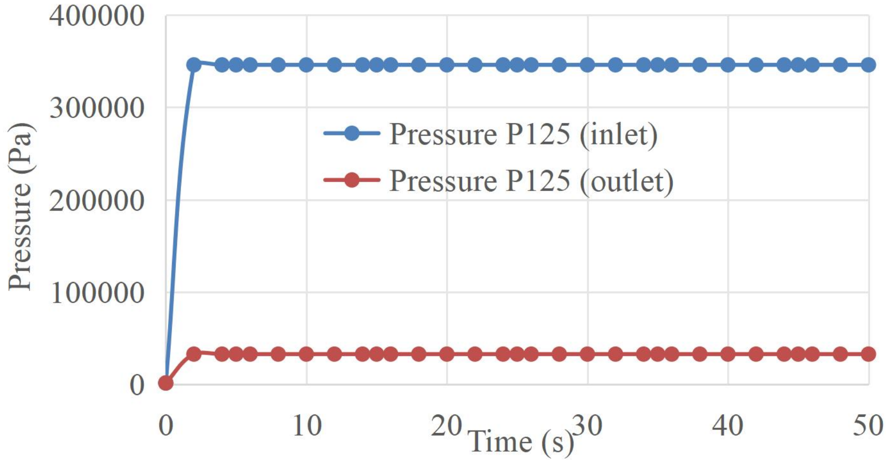


Figure 6.9 Processed pressure difference for the Increased length on

Case 1.3: Flow rate determination in a pipe at a set pressure drop

For this case we provide an example of boundary condition inputs that represent known pressure differentials and an unknown flow rate. To simulate this case, we recall the Source Vol input file description in section 5.11111, when we explain the control term of ebt. When pressure is unknown, t=1 then W2 and W3 will be the time dependent data of temperature and static quality but , flowrate is to be determine, then t=3 where the TD data for W2 and W3 will be pressure and temperature. Thus we will edit the (ebt) of WORD 1 CARD 1100200 from 101 to 103 and values of known pressure and temperature be entered in WORDS 2 and 3 of CARD 1100201of APPENDIX I that resulted in the input file in APPENDIX III.

```
*          ebt(fluid type, no-boron, initial thermodynamic state)
1100200    103
*          time   Pressure(pa)  temp(K)
1100201    0.0    1160000.0     288.0
```

The case inputs are summarized as:

*--
***********Input Data**********
*---
*Pipe diameter=6cm = 0.06 m

*Pipe Length=30 m

*Inlet temp = 288K

*Roughness = 0.00001

*Inlet Pressure, P1= 1.16 MPa

*Outlet Pressure, P2 = 1.06 MPa

= Flow rate determination in Single horizontal pipe

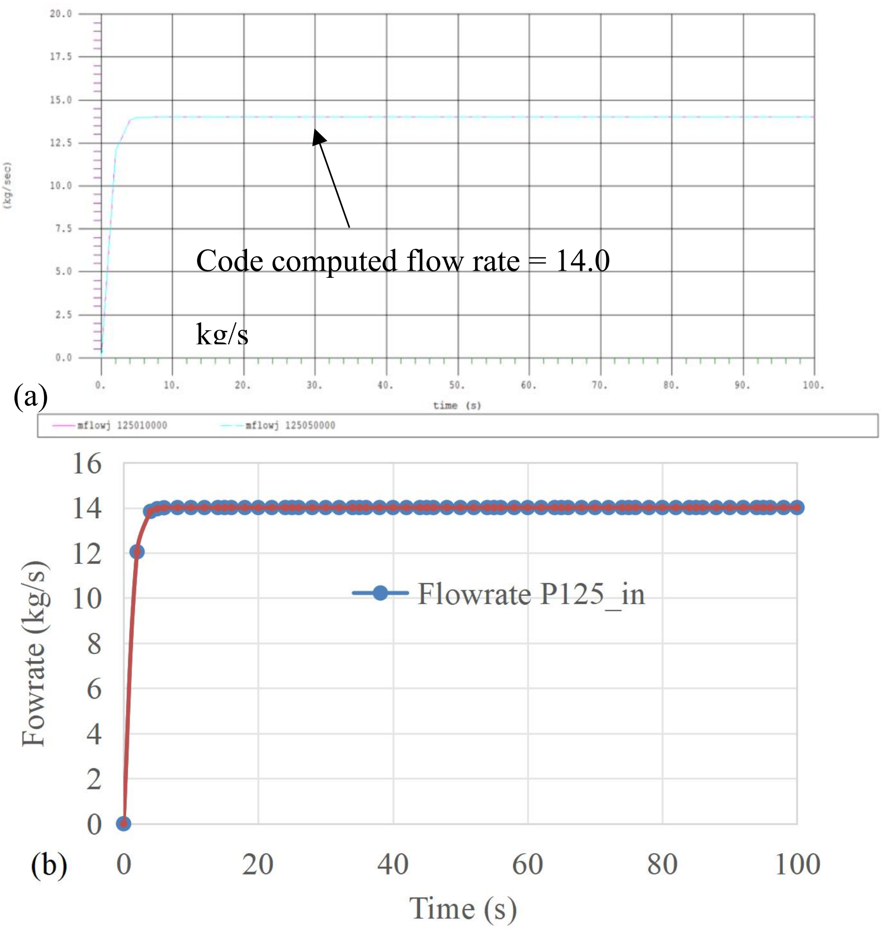


Figure 6.10 Computed flow rate (a) Code output (b) Process data

Figure 6.10 shows the flow rate in the pipe when the inlet and outlet pressure boundary are known. This is the flow rate required to transport the fluid through a 30m pipe from the inlet volume at 1.16MPa to the last volume at 1.06 MPa of the pipe.

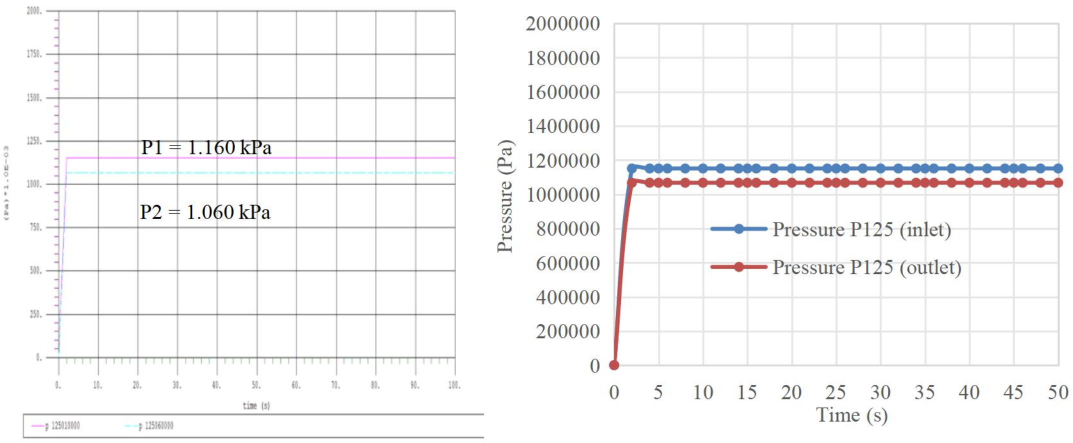


Figure 6.11 Set Pressure difference (a) Code output (b) Process Data

Case 1.4: Flow rate in a Reduced Pipe Length

The nodalization diagram for this sensitivity analysis case is same as Figure 4.1. To reflect the reduced pipe length, WORD …of CARD.. in line of the sample input file for CASE #1 provided in Appendix I is modified to 2.5 from 5.0. This will change the length was from 30 m to 15 m (having a vol number of 6.0, the Cell size of 5.0 is adjusted to 2.5 which when multiply by the vol. Number of 6.0 will give the required length of 15m to reflect the case.

*---
*********** **Input Data** **********
*---
*Pipe diameter=6cm = 0.06 m

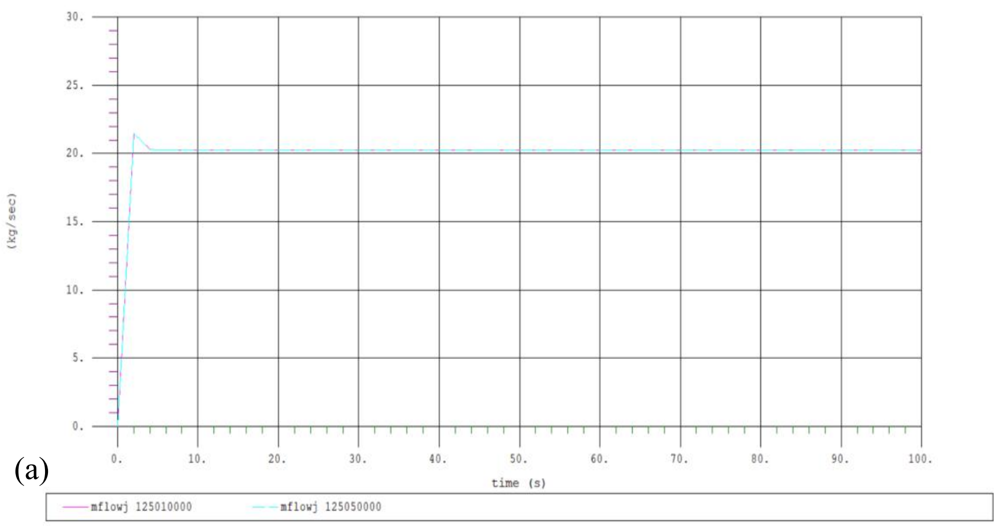
*Pipe length=15 m

*Inlet temp = 288K

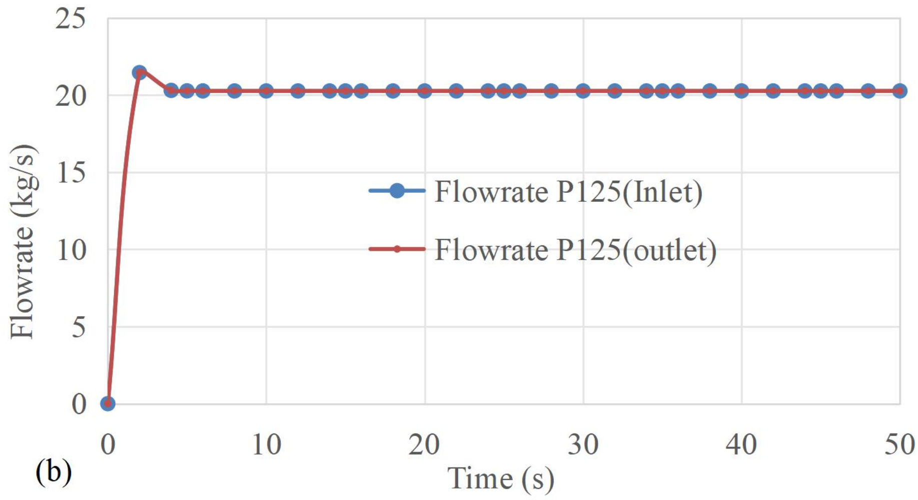
*Roughness = 0.00001

*Inlet pressure, P1= 1.16 MP

*Outlet pressure, P2 = 1.06 MP



(a)



(b)

Figure 6.12 Code computed flow rate at reduce pipe

Again, as length of pipe reduces with same value of pressure drop, flow rate is observed to increase as expected as shown in Figure 6.13 (both the code output result and post-processed code data are shown so that new users can practice plotting in RELAP and in a third party-code).

The flowrate of the 30m length pipe presented in earlier Figure 6.10 shows a flowrate of 14.0 kg/s while the flowrate of shorter pipe in Figure 6.12 indicates a flow rate of 20.0 kg/s. The simulations results shows an increase in the flowrate of the shorter pipe than as observed in the longer pipe and comparing this result with the analytical result, they show agreement..

a. Case #2.2: Increased pipe's roughness on the reduced Pipe

The Nodalization diagram for this sensitivity analysis case is same as Figure 4.1. Word …of Line in CARD of the sample input file for case #1 provided in Appendix I. The roughness was change from 0.0018 to 0.0058 (WORD # of 0.0018 is adjusted to 0.0058 to reflect the case.

The sensitivity analysis of the flow calacultion to roughness is conserved below.

*---

*********** Input Data **********

*---

*Pipe diameter=6cm = 0.06 m,

*Pipe Length=15 m

*Inlet temp = 288K

*Roughness = 0.0050

*Inlet Pressure, P1= 1.16 MP

*Outlet Pressure, P2 = 1.06 MP

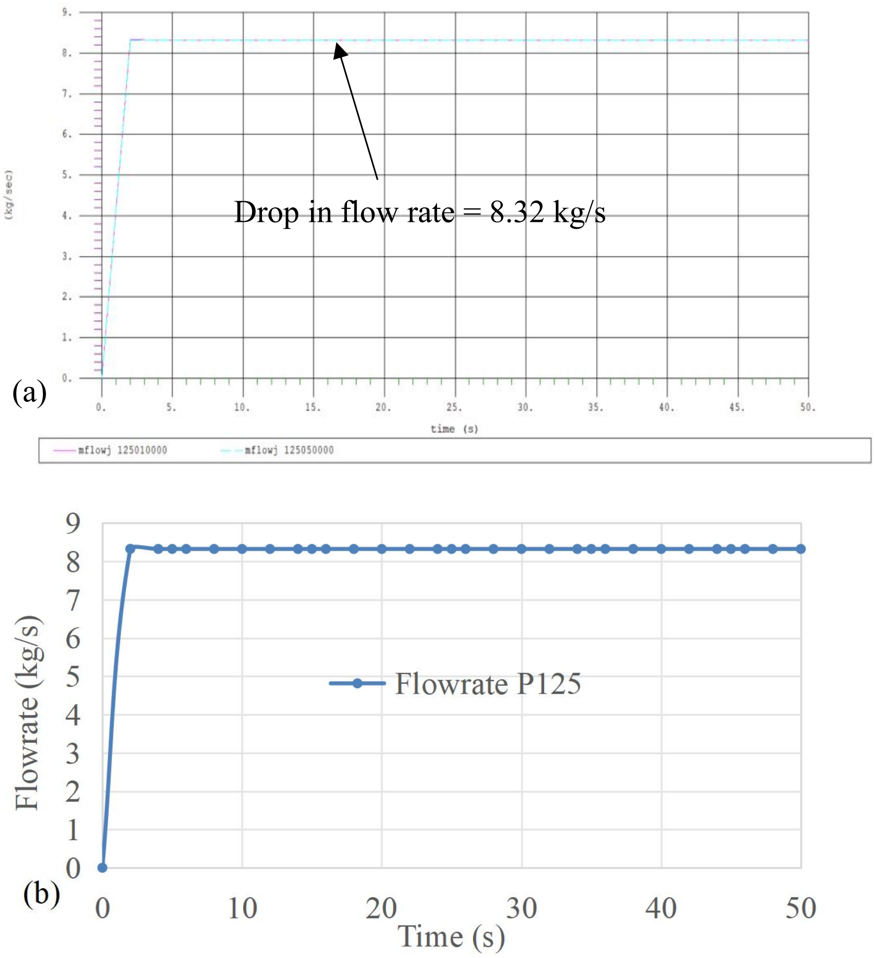


Figure 6.13 Reduced flowrate due to increased pipe roughness
(a) code output (b) Processed data

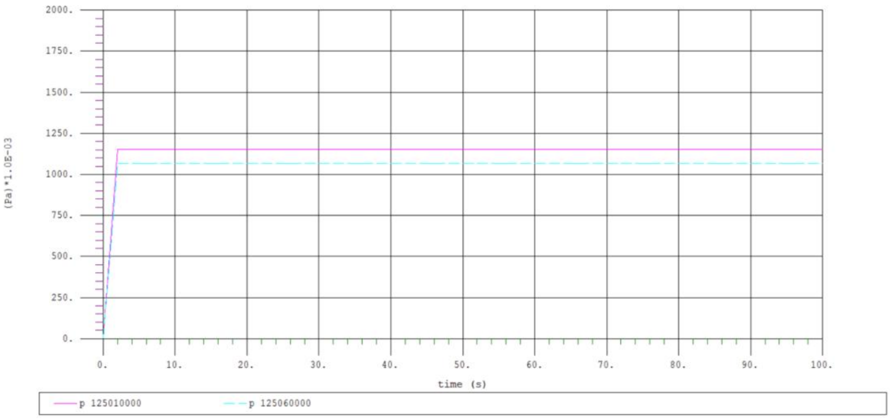


Figure 6.14 Pressure Difference at reduced length of single horizontal

A significant reduction in flowrate for the increased roughness relative to the reduce length pipe with minimal roughness is shown in Figure 6.13.which agrees well with the analytical check. The set pressure in the inlet and outlet of the pipe is shown in Figure 6.14.

6.1.2 CASE #2: Connecting pipes

Nuclear power plant systems usually involve series of interconnected branches with some regions having parallel flow paths, while other regions having serries connected pipes. The following sections provide examples, sensitivity studies, and comparison to theoretical calculations so that new code users will have a complete set of simplified models to learn from.

Case 2.1: Series Connection with Set Flowrate

In this case we provide an example of boundary condition inputs that represent known flow rate, pipe's dimensions, temperature and unknown over all pressure drop in 2 horizontal pipes connected in series. The Nodalization diagram is given in Figure 5.4 and sample input file for the reference case is provided APPENDIX II. The case inputs are summarized as:

*---
*********** **Input Data** *********
*---

*Mass flow rate = 10.0 kg/s

*Pipes diameter= 0.06 m

*Pipes Length=30m each

*Inlet temp = 288K

*Roughness = 0.00001(Very Smooth Pipe)

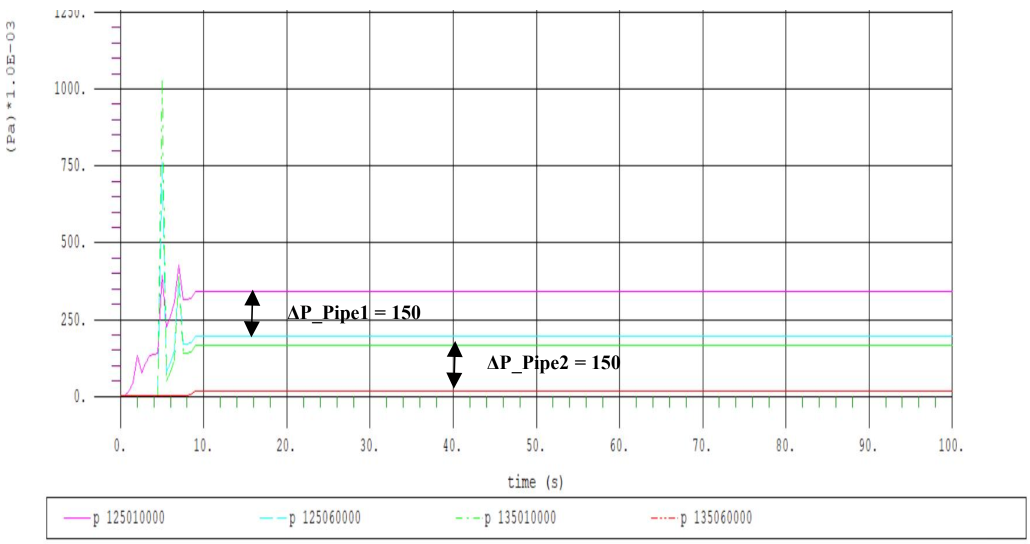


Figure 6.15 Combined pressure trend in both pipes

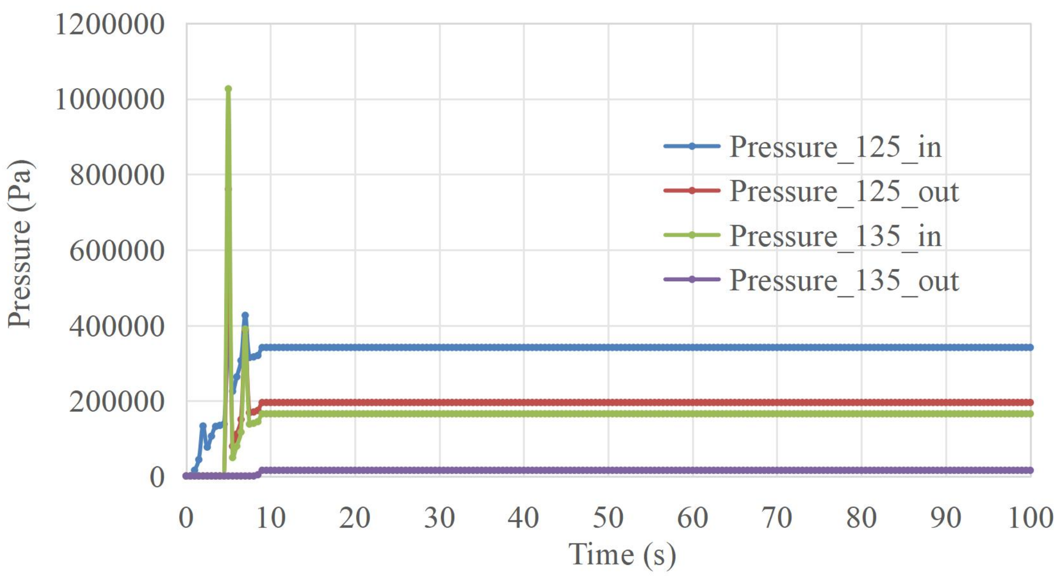


Figure 6.16 Processed pressure drops in both pipes

The results in Figure 6.15 and 6.16 above shows that the converged steady-state solution after about 10s of iterations within the code (prior to this time the code has several fluctuations as it converges to steady state). Using the file in Figure 6.17 to strip the ASCII data from the RELAP output file the user can plot the output in MS Excel following the same step as in section 6.1.1

```
= Title
100 strip csv
103 0
*
* pipe 1 data of interest
1001 tempf 125010000
1002 tempf 125060000
1003 mflowj 125010000
1004 mflowj 125060000
1005 p 125010000
1006 p 125060000
* pipe 2 data of interest
1007 tempf 135010000
1008 tempf 135060000
1009 mflowj 135010000
1010 mflowj 135060000
1011 p 135010000
1012 p 135060000
. End of input.
```

Figure 6.17 Strip file input for connecting pipes

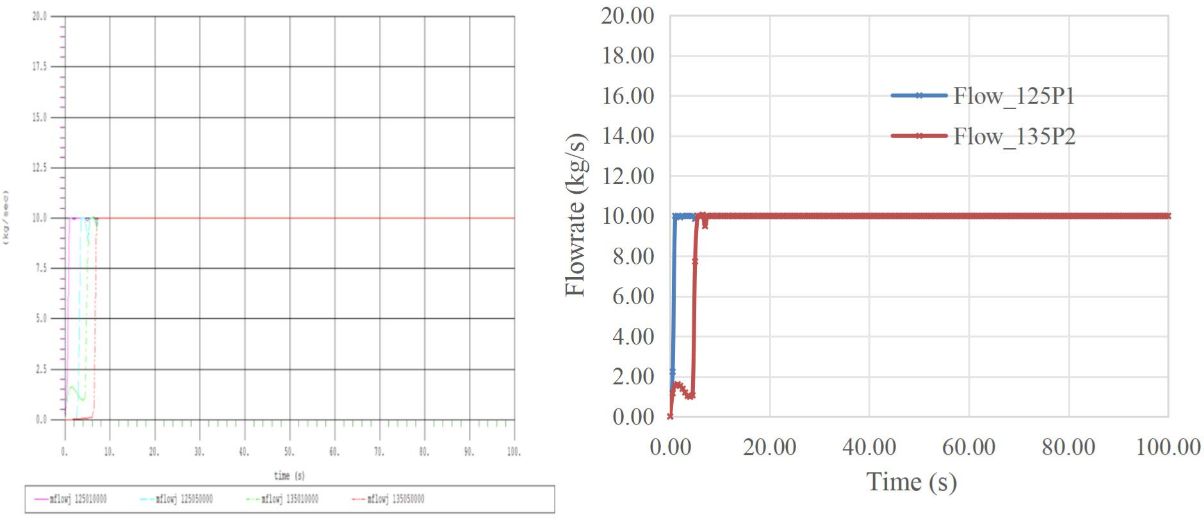


Figure 6.18 Plot of Set Flow rate for both Pipes

Figure 6.15 and 6.16 show the pressure drop between the inlet of pipe 1 to the outlet of pipe 2. The result is extracted from the mentioned Figures of the pressure in each of the pipes. From the extracted result, the inlet of pipe 1 shows a pressure of 346 kPa and a drop in pressure at the outlet to 196 kPa. Similarly. Inlet of pipe2 as 166 kPa to an outlet drop of 16 kPa. The simulated pressure drop of both pipes is;

$$\Delta P_{Total} = \Delta P1 + \Delta P1$$

$$= 150 + 150 = \mathbf{300\ kPa}$$

6.1.2.1 Analytical Validation

Calculating the total pressure drop in both pipes will be the sum of the pressure drop in Pipe 1 and that of pipe 2. Given the length of both pipe to be 30 m each while at same temperature and flow rate as calculated in section 6.1.1.1. using;

$$\Delta P = \left(f\frac{L}{D}\right)\left(\rho\frac{\nu^2}{2}\right)$$

ΔP_Pipe1 = 153 kPa

The combined pressure drop, ΔP_Total can be determined as

ΔP_Total = ΔP_Pipe1 + ΔP_Pipe2

Note that ΔP_Pipe1 = ΔP_Pipe2 = 153 kPa

ΔP_Total = 153 + 153 = **306 kPa**

From the analytical result of the total pressure drop across both pipes is close to the simulated result of **300 kPa**, with the small differences likely arising from differences in the friction factor correlations used in the code.

6.1.2.2 CASE #2 Sensitivity Analysis

- **Case 2.2: Reduced Length Simulation (Flow rate comparism)**

In this case we provide an example of boundary condition inputs that represent known flow rate, pipe's dimensions, temperature and unknown over all pressure drop in half the length (15m each) of the 2 horizontal connected pipes presented in Case 2.1.

The nodalization diagram for this sensitivity analysis case is same as Figure 5.4. To reflect the reduced pipe length, WORD 1 of CARD 1250301 and CARD 1350301 of the sample input file for CASE #1 provided in Appendix I is modified to 3.0 from 6.0. This will change the length from 30 m to 15 m (having a vol number of 5.0, the Cell size of 6.0 is adjusted to 3.0 which when multiply by the vol. Number of 5.0 will give the required length of 15m to reflect the case.

```
*           lengh     vol no.
1250301     3.0       5
*           lengh     vol no.
1350301     3.0       5
```

The case inputs are summarized as:

*--
*********** **Input Data** *********
*--
*Mass flow rate 10.0 kg/s

*Pipes diameter=6cm

*Pipes length=15 m each

*Inlet temp = 288K

*Roughness = 0.00001

The simulation was carried out in both RELAP5 and ASYST. Inlet and outlet Pressures of both pipes are calculated analytical for validation purposes

Temperature and flowrate rate are kept constant in both simulations.

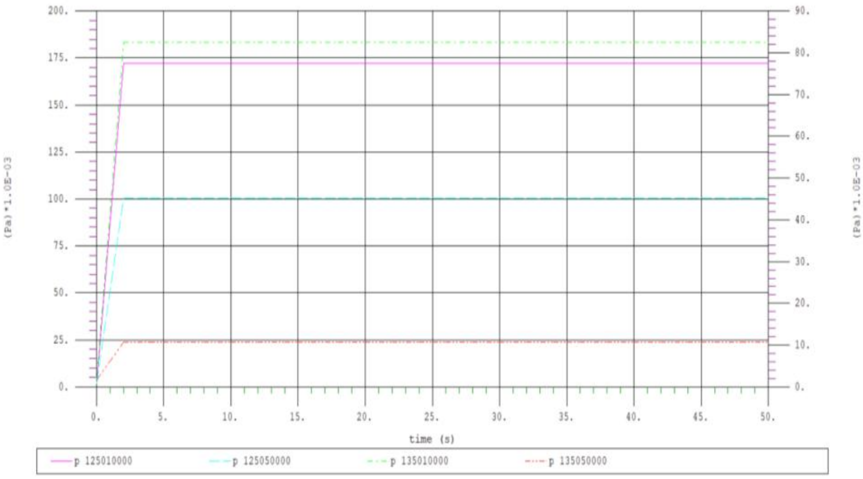


Figure 6.19 RELAP5 Output of the combined pressure drop.

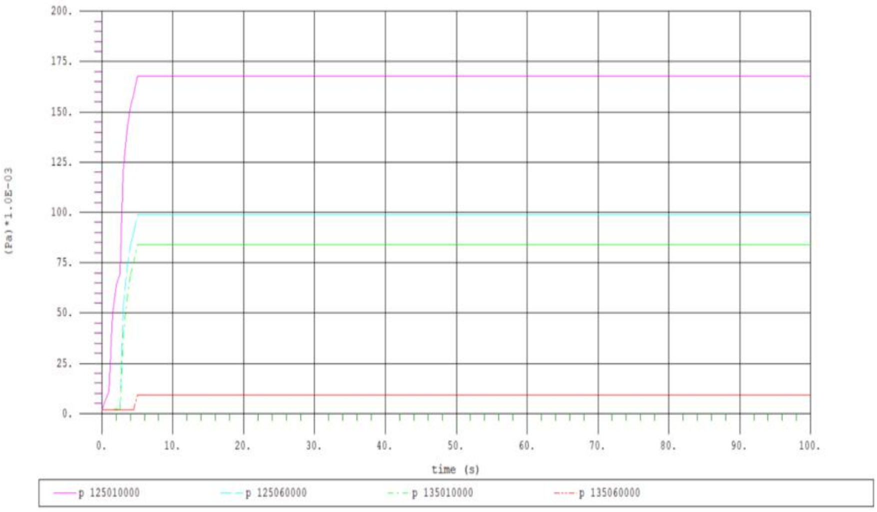


Figure 6.20 ASYST plot of pressure drop in both Pipe

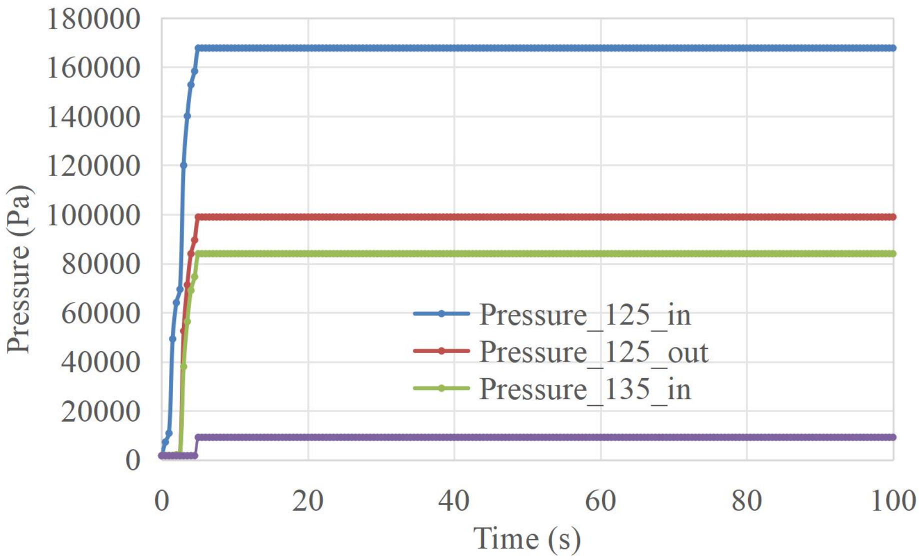


Figure 6.21 Processed pressure drop of both pipes.

Figure 6.19 - 6.21 shows the pressure drop between the inlet of pipe 1 to the outlet of pipe 2. The result is extracted from the mentioned Figures of the pressure in each of the pipes. From the extracted result, the outlet of pipe 2 shows a pressure of 175 kPa and a drop in pressure at the outlet to 10 kPa. This amount to a total of 165 kPa of pressure drop in both pipes. We shall be validating this result analytical but its agrees with the previous 30m long case that also had a pressure drop of **165 kPa**

Analytical Validation of reduced Pipe Length

Given the Pipes in CASE 2.2 with reduced length from 30 m to 15 m for each pipes, the total pressure drop, ΔP while other parameters remain same will be determine analyticaly using equation below;

$$\Delta P_{Total} = \Delta P_{Pipe1} + \Delta P_{Pipe2}$$

Given L = 15 m with all other parameter constant

$$P_{Pipe1} = \left(\frac{0.052 \text{ x } 15}{0.06}\right)\left(\frac{999.1 \text{ x } 3.54^2}{2}\right)$$

$$= 81385.9014$$

$$= 81 \text{ kPa}$$

$$\Delta P_{Pipe1} = \Delta P_{Pipe2} = 81$$

Thus,

$$\Delta P_{Pipe1} = 81 + 81 = 162 \text{ kPa}$$

All three methods show excelent agreement as expected and carried out for all other cases.

6.1.3 CASE #3: Parallel Pipe Connection (Multichannel) Model

In this case we provide an example of boundary condition inputs that represent known inlet flow rate, temperature and pressure in 2 parallel pipes. The objective is to determine the exact flow rate in each pipe which is expected to be equal. The Nodalization diagram is given in Figure 5.6 and sample input file for the reference case is provided Appendix II.b.. The case inputs are summarized as:

*--

*********** **Input Data************

*---

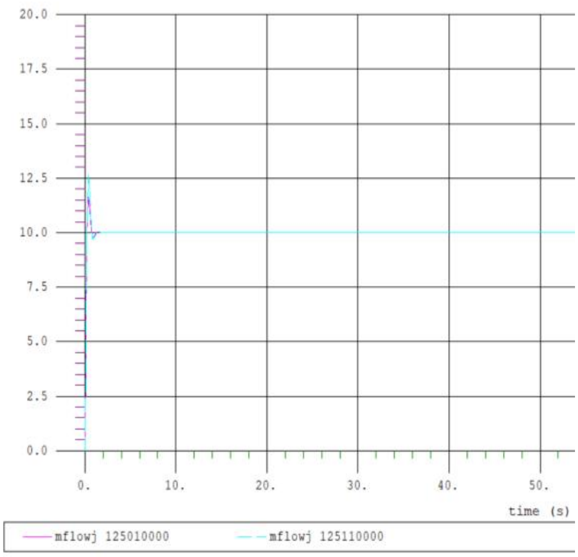
*Mass flow rate 20 kg/s

*Pipes diameter = 0.06 m each

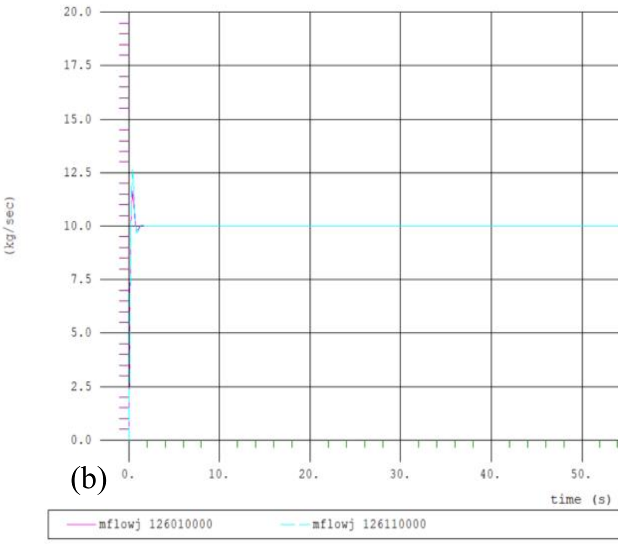
*Pipe length = 6 m each

*Inlet temp = 513K

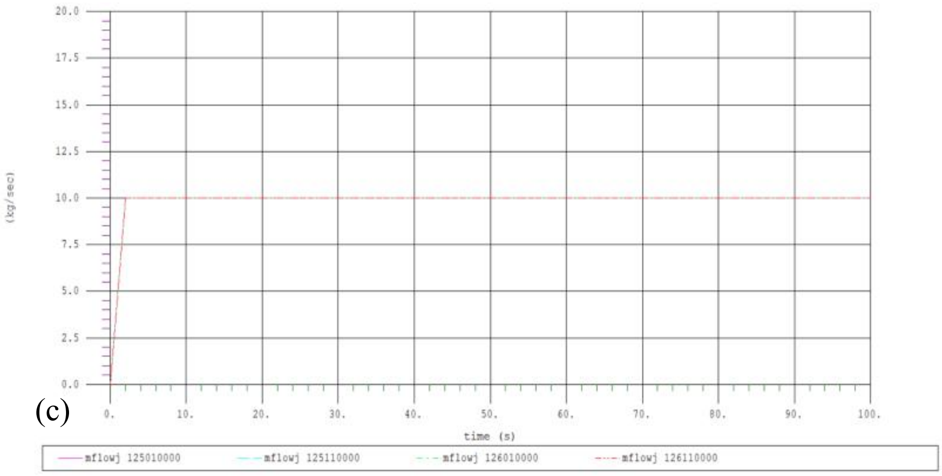
*Pressure = 8MP @ both end of Pipes



(a)



(b)



(c)

Figure 6.22 Flowrate distribution in the parallel connection

(a) Pipe 1 (b) Pipe 2 (c) Both Pipes

The result of simulating two parallel non-heated pipes flow connected by a branch component set at an initial flow of 20kg/s shows an equal distribution of flow on each pipe at 10kg/s each (Figure 6.22). Its is logical to agree with the flow rate in each pipe considering that the inlet and outlet conditions of the TDVs are constant and consistent between the channels.

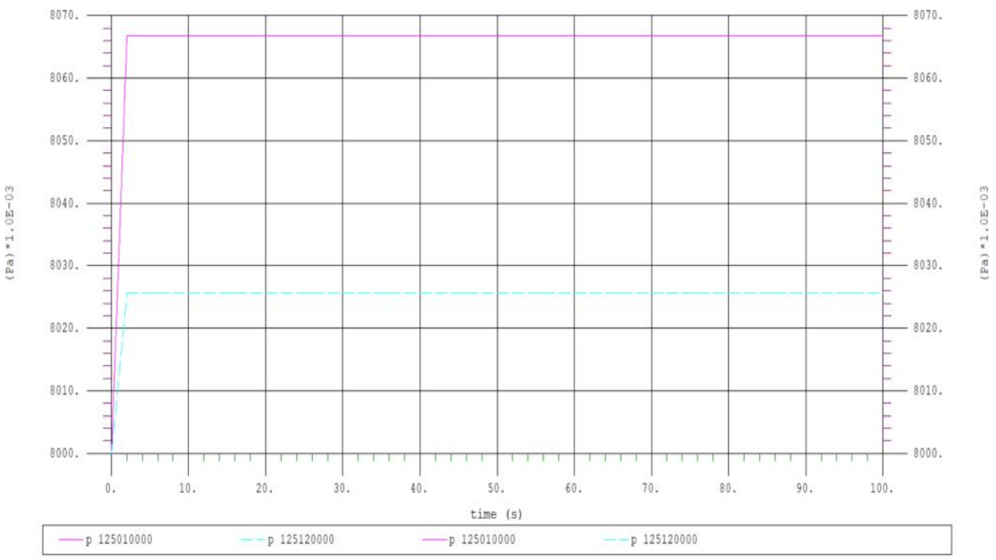


Figure 6.23 Pressure loss in the parallel connection

6.1.3.1 CASE #3 Sensitivity Analysis

Case 3.1 Unequal pipes in parallel

The Nodalization diagram for this sensitivity analysis case is same as Figure 5.6.

For this case we provide an example of boundary condition inputs that represent Pipes with differential diameter (0.06m and 0.03m) connected in parallel. We aim to determine the unequal flowrate in each pipe. To achieve this, WORD 1 (Flowarea) of CARD 1260101 of CASE #3 input file in Appendix II.b was recalculate using the reduce diameter of pipe 2, now 0.03m which gives 0.00071.

```
*          flowArea   vol no.
1250101    0.002827     5
*          flowArea   vol no.
1260101    0.00071     12
```

The case inputs are summarized as:

*---

***********Input Data**********

*--

*Mass flow rate 20 kg/s

*Pipe 1 diameter=0.06 m

*Pipe 2 diameter=0.03 m

*Pipe Length=6 m

*Inlet temp = 513K

*Roughness = 0.00001

*Pressure = 8MP

Considering that there is no heating, the temperature remain unchange. Figures 6.24 and Figure 6.25 shows the parameter changes in both pipes.

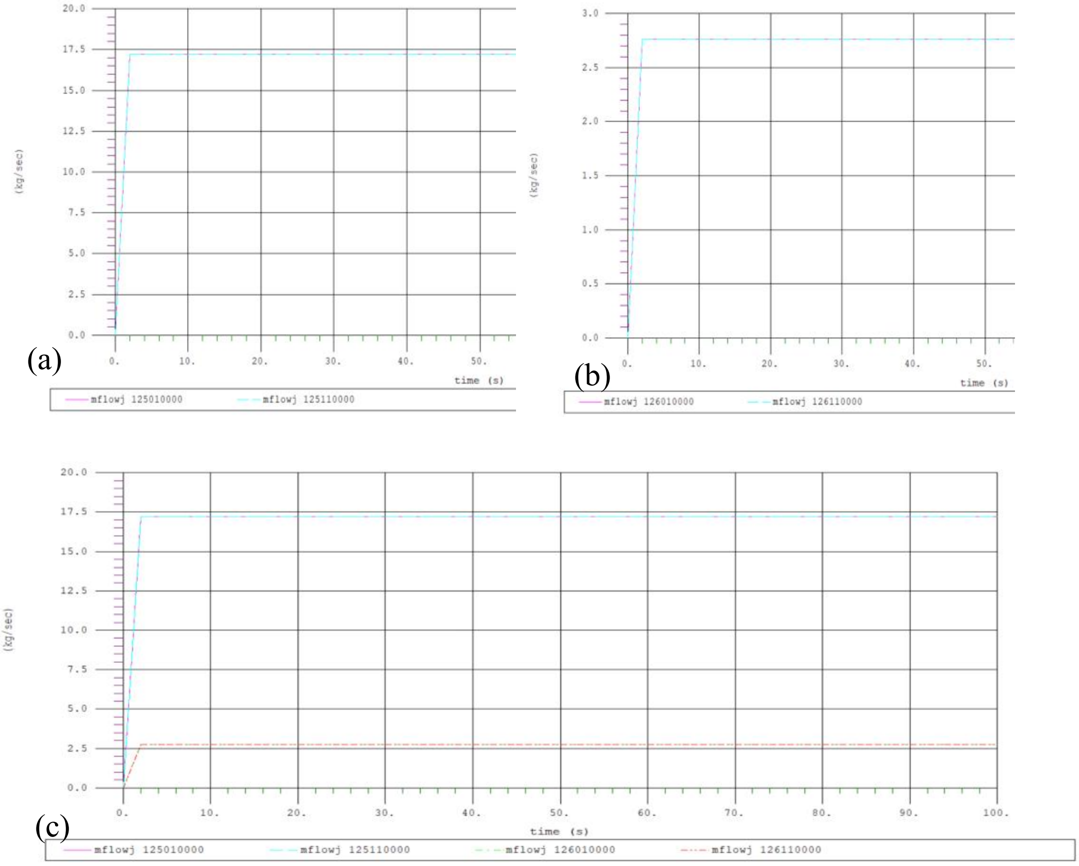


Figure 6.24 Unequal Flowrate distribution in the parallel connection

(a) Pipe 1 (b) Pipe 2 (c) Both Pipes

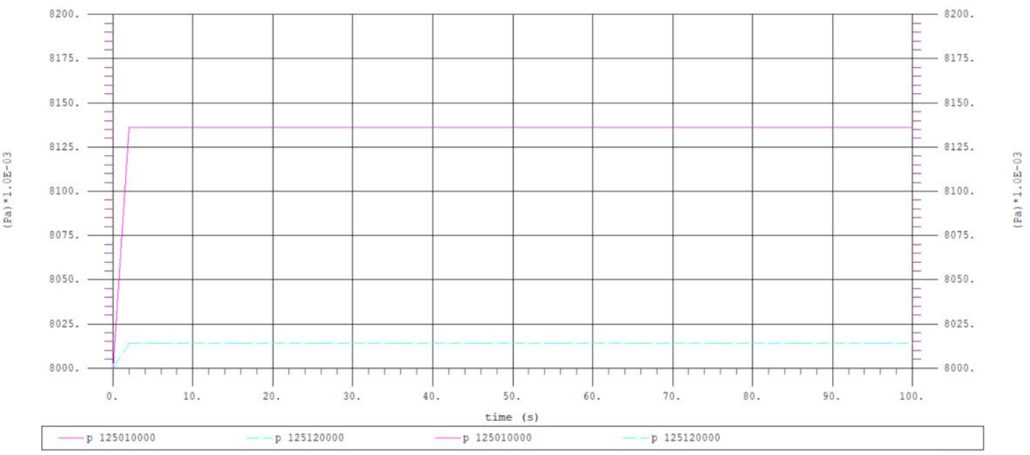


Figure 6.25 Pressure loss in the unequal parallel connection

6.1.4 CASE #4: Bend Pipe / Elbows

Complex systems involve various geometrical features that cause local pressure drop including pipe bends, diameter changes, etc that are referred to in many textbooks as "minor" losses. The pressure drop in these components is defined using a "K-factor" which is an empirically derived coefficient the rates the pressure-drop in that component to the square of the flow rate. The following cases provide examples of loss factors in RELAP and ASYST.

Case 4.1: Flow in a Bend

In this case we provide an example of boundary condition inputs that represent known flow rate, pipe's dimensions, temperature and unknown over all pressure drop in 2 pipes connected 90 degrees. The aim is to determine the pressure drop in the pipe bend. The nodalization diagram is given in Figure 5.7 and sample input file for the reference case is simply changing the orientation of pipe 2 in CASE # by replacing 0.0 in WORD 1 of CARD 1350601 with 90 or -90 depending on either upward or downward flow. The case inputs are summarized as:

```
*-----------------------------------------------------------------
*********** Input Data**********
*-----------------------------------------------------------------
*Mass flow rate 10.0 kg/s

*Pipes diameter= 0.06m

*Pipes Length=30 m each

*Inlet temp = 288K

*Pipe1=Horizontal

*Pipe2=Vertical elevation Upward
```

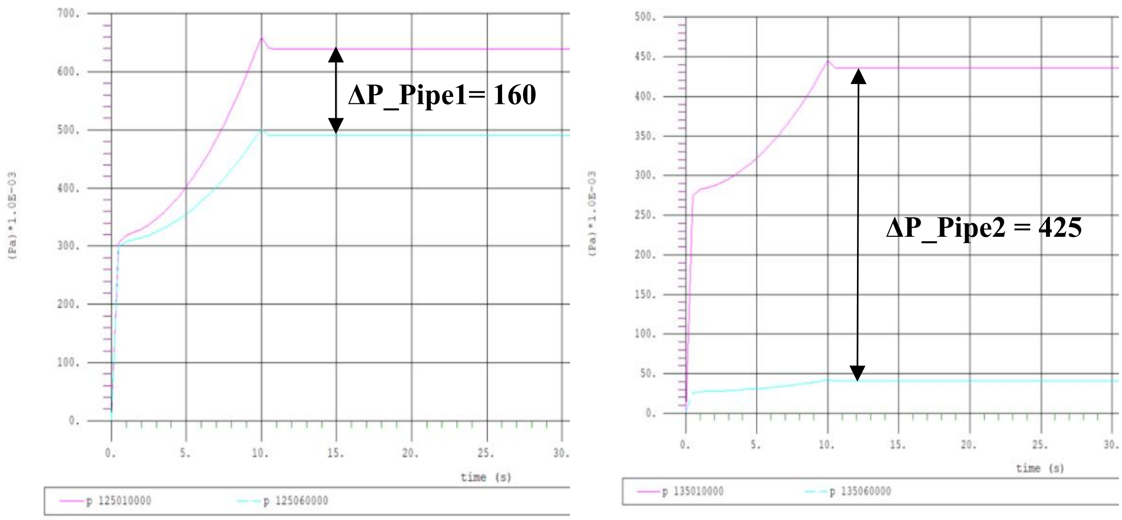


Figure 6.26 Pressure drop determination in a bend (a) Pipe 1 (b) Pipe

Pipe1: P_in = 640kPa and P_out = 490 kPa

ΔP_P1 = 160kPa

Pipe2: P_in = 465kPa and P_out = 40Pa

ΔP_P2 = 425kPa

ΔP_P1 + ΔP_P2 = 425 + 160

= 585 kPa

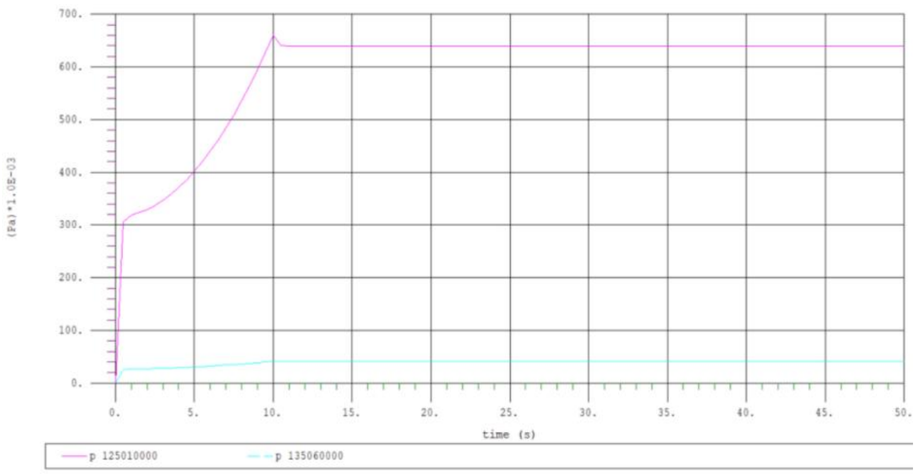


Figure 6.27 Total Pressure drop in the bend pipes (Pipe 1 and Pipe

Pipe1_P_in = 640 kPa and Pipe2_out = 40 kPa

ΔP_Total (Simulation) = 600 kPa

Case 4.2: **Flow rate determination in a bend**

The nodalization diagram for this case is same as in Figure 5.7. We provide an example to determine the flow rate in 2 pipes connected 90 degrees. This is pretty similar case as Case 1.3 except that this is on a bend. As such, the procedure and modification is repeated on the case 4.1 to change the ebt from 101 to 103 and assign the pressure and temperature value summarized as;.

*---

*********** **Input Data************

*---

*Pipes diameter=6cm

*Pipes Length=30 m each

*Inlet temp = 288K

*Pipe1=Horizontal, Pipe2 = vertical downward flow

P1=1160 k Pa and P2=1010 kPa

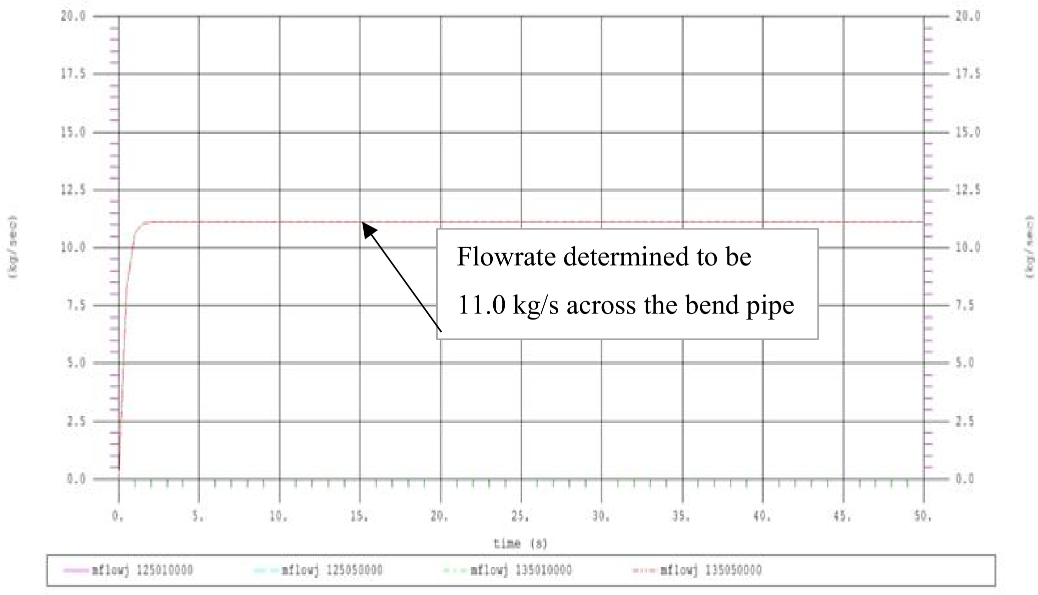


Figure 6.28 Computed Flowrate in a bend pipe

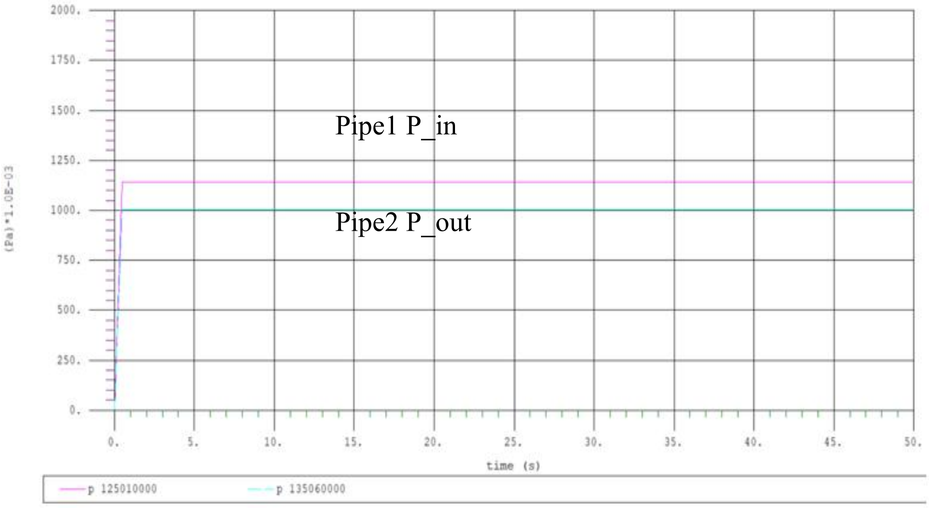


Figure 6.29 Pressure Drop in the Bend

6.2 Heat Structure Cases

6.2.1 CASE #5: Heat Structure on Single Pipe

In this case we provide an example of boundary condition inputs that represent a heated pipe with a known flow rate, pipe's dimensionstemperature and unknown pressure drop in 2 pipes connected 90 degrees. The aim is to determine the fluid/wall temperature and pressure drop in a stainless steel pipe. The nodalization diagram is given in Figure 5.9a and the sample input file for the reference case is provided Appendix V.

The case inputs are summarized as:

*---
***********Input Data**********

*---

*Mass flow rate= 30 kg/s

*Pipe diameter= 0.06 m

*Pipe Length= 6 m

*Inlet temp = 320 K

*Pipe roughness = 0.0018

*Pressure drop is to be determined

*Fluid/Wall Temp. = To be solve by Code

(a)

(b)

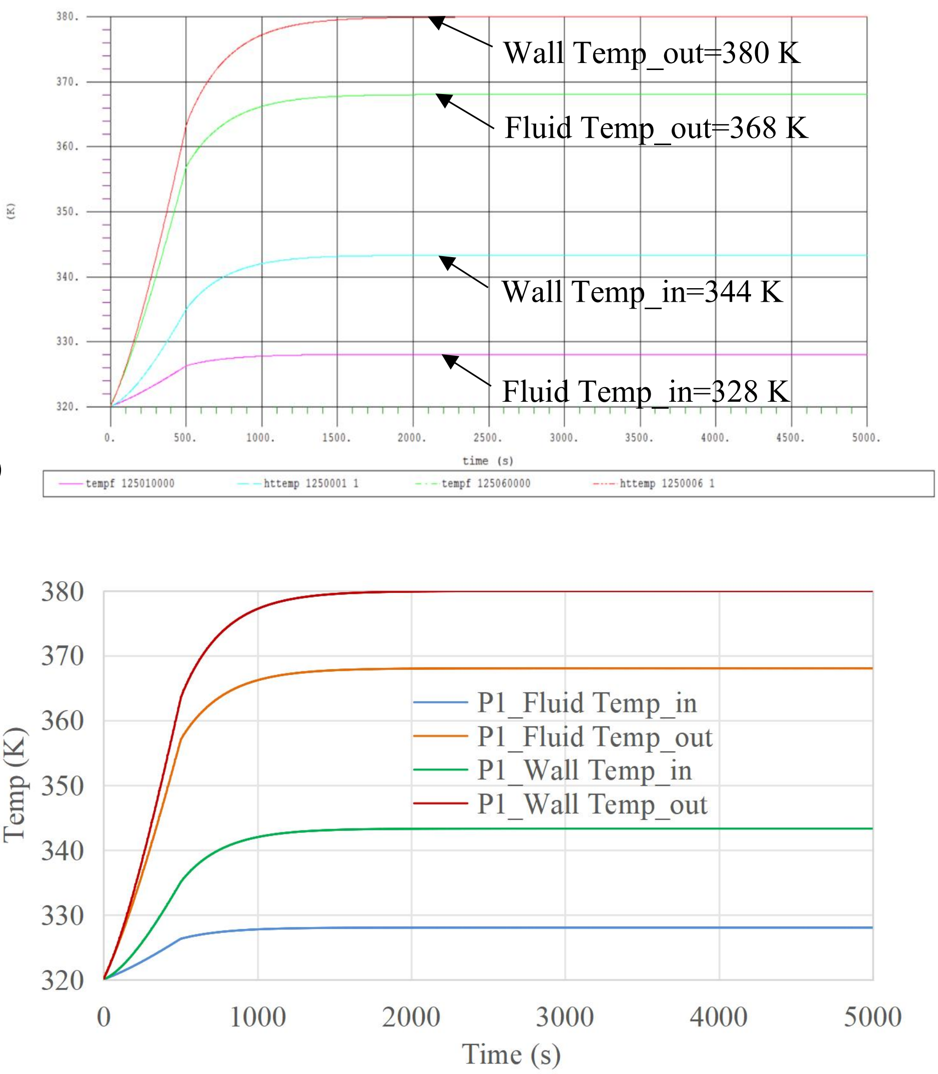


Figure 6.30 Fluid and Wall Temperature trend in the Single Horizontal Pipe

(a) Code plot (b) Processed Data

Flowrate and Pressure from RELAP5/ASYST and Processed Data

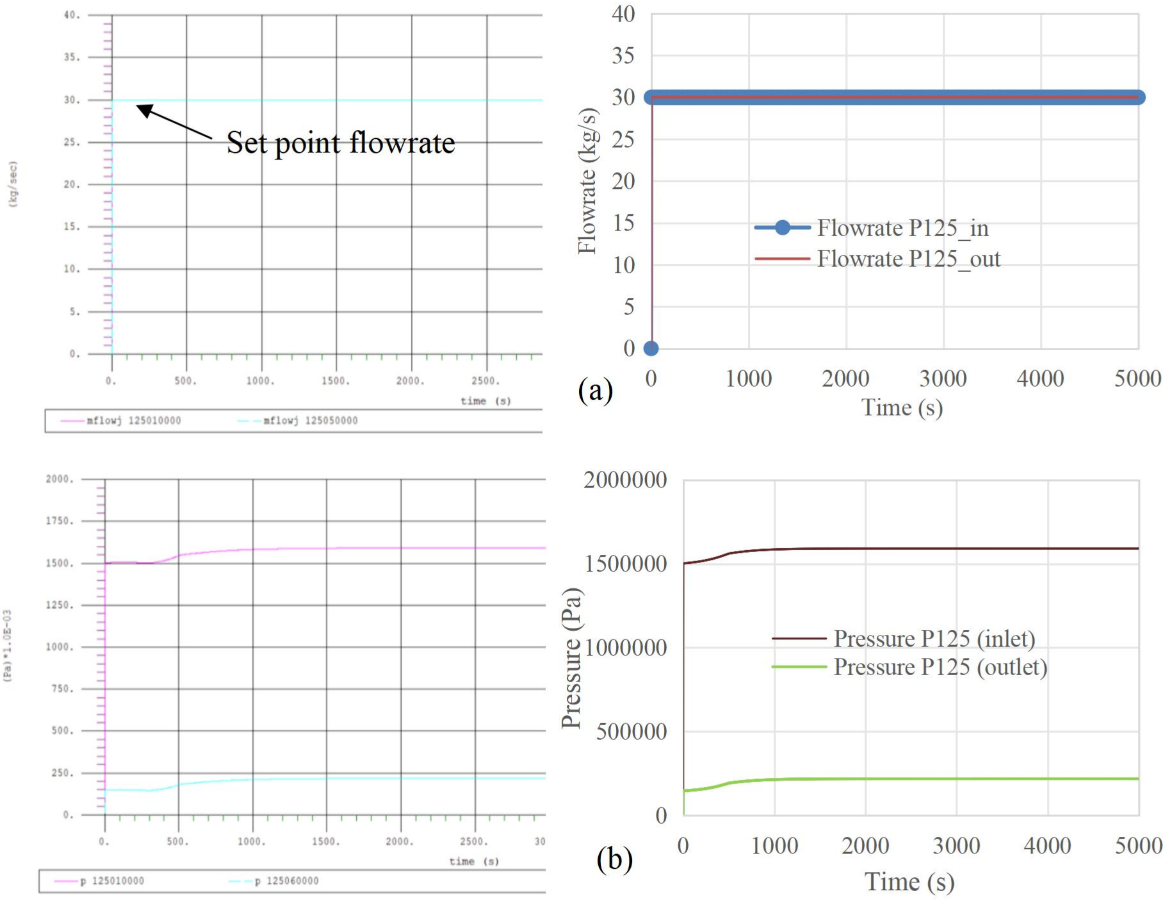


Figure 6.31 Single Heated Pipe (a) Flowrate (b) Pressure trend

6.2.1.1 Sensitivity Analysis

Having determined the parameter changes based on the initial conditions from case 10, we went on to demonstrate to new users the sensitivity to lower (half) and higher (double) powers from the heating power from the initial condition of 50kW.

Case 5.5: Parameter changes at reduced (25 kW) Heating of same pipe length

The nodalization diagram for this case is same as in Figure 5.9. We provide an example to determine the wall and fluid temperature at a reduce heating. To achieve this, WORD .. (Flowarea) in LINEs… of case #5 input file in Appendix V was recalculate using the reduce diameter of pipe 2, now 0.03 m,) as shown in Appedix II to represent the current scenerio The case inputs are summarized as:

*--

***********Input Data**********

*--

*Mass flow rate= 30 kg/s *Pipe diameter= 0.06 m

*Pipe length = 6 m *Inlet temp = 320 K

*Pipe roughness = 0.0018 *Heating power = 100kW

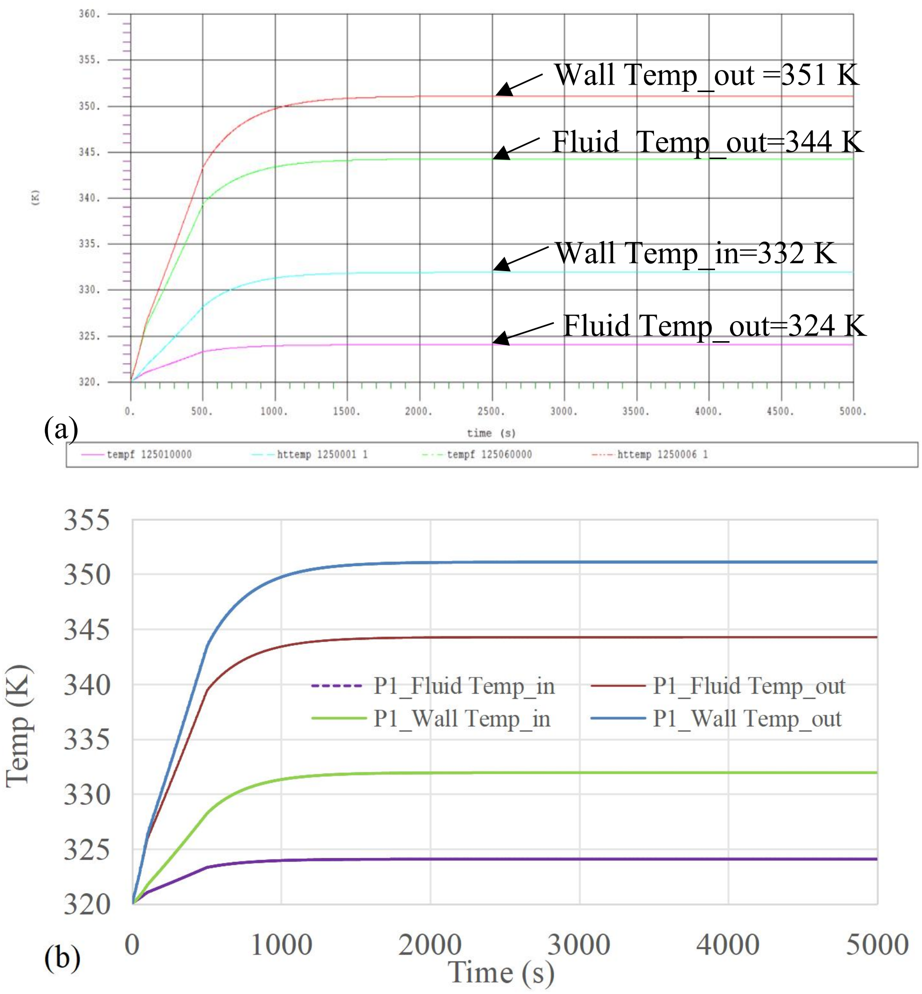


Figure 6.32 Fluid and Wall Temperature trend in a reduced heating (25kW)

(b) Code plot (b) Processed Data

As expected from a lower heating power, temperature drop in both the fluid and wall are also visible in Figure 6.45. This is as expected as less heating will amount to lower temperature in both the pipe wall and subsequent transfer to the fluid. Flowrate remained the same at set point of 30 kg/s

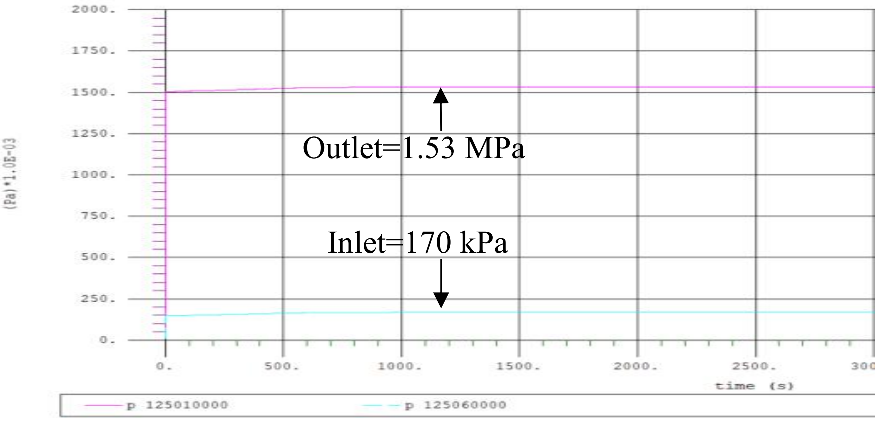
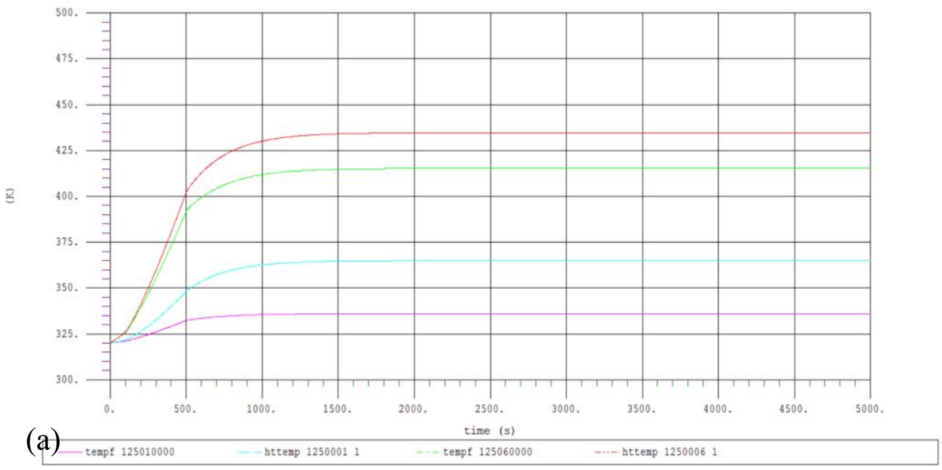


Figure 6.33 Reduced Heating Pressure Drops

Case 5.6: Parameter changes at 100kW Heating of same pipe length

In this case, the procedure of case 5.5 is repeated with change of the heating power from 25000.00 to 10000.0 in WORD of CARD.

Users can observe the effect of the power provided to the heat structure with higer heating power results to increase temperature which is demonstrated in the code output and processed data plots of Figure 6.34



(a)

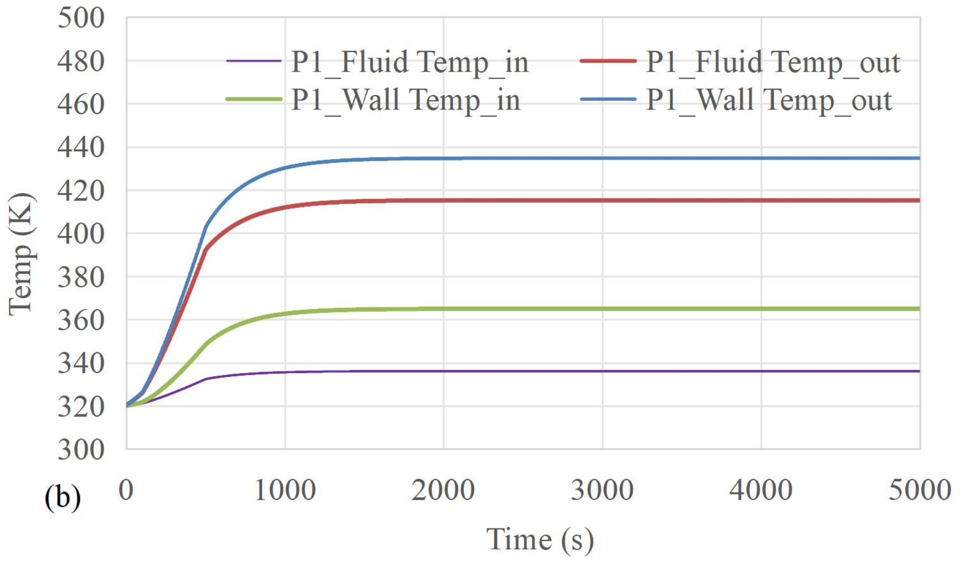


Figure 6.34 Fluid and Wall Temperature Trend in of Increased Heating (100kW)

(c) Code plot (b) Processed Data

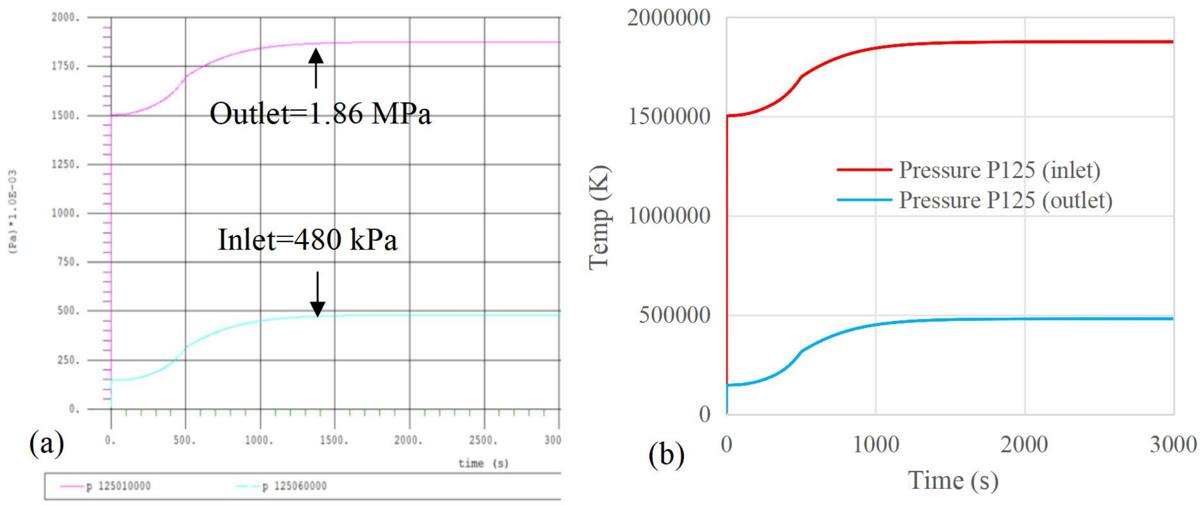


Figure 6.35 Increased Heating Pressure Drops

(a) Code plot (b) Processed Data

6.2.2 CASE #6: CANDU Fuel Bundle

In order to provide new users with a more meaningful example of code inputs for a nuclear power plant, the following example is provided in the Workbook. The 37 fuel element CANDU fuel bundle is modeled and simulated in this section with the operating parameters of the CANDU reactor. Figure 6.26 shows the arrangement of the fuel pins in the bundle with the corresponding dimensions.

These parameters are then subsequently used to derive the heat transfer areas, flow areas, and hydraulic diameters that are needed for the code inputs.

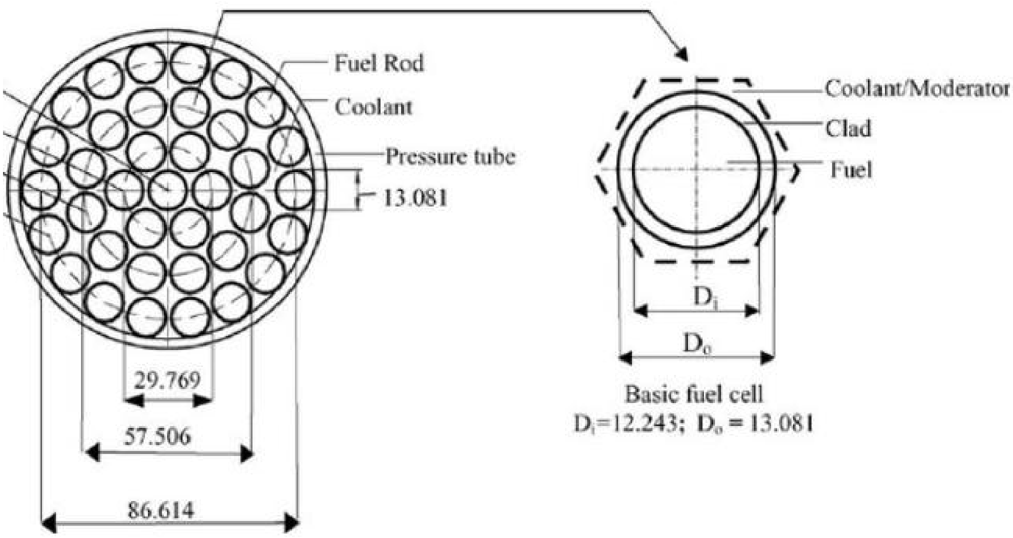


Figure 6.36 CANDU Fuel bundle geometrical representation adoptrf from [72]

6.2.2.1 Fuel Heat Transfer Properties Calculations

Calculations was carried out using MS Excel to determine the several heat transfer properties of the fuel bundle as presented in table 6.1 used in developing the input file.

Table 6.1 Calculated CANDU Fuel Bundle Properties

Items	Properties	Value
Fuel	**Number of Pins**	37
	Pin Diameter	0.012
	Pin Area	0.00011304
	Pins Total Area	0.00418248
	Wetted perimeter	0.03768
	Total wetted perimeter	1.39416
Pressure Tube	Diameter	0.104
	Area	0.00849056
	Flow Area	0.00430808
	Wetted perimeter of tube	0.32656
	Length	6
Fuel Bundle	Number of bundles	12
	Length	6
Heat Transfer	**HT_Dh**	0.01236036
	Flow _DH	0.01001459

By now, we are already familiar with some of the derived data in the table 6 above. To correctly develop the input file of the unique case of CANDU fuel bundle, the highlighted parameter are the unique values to edit in the heat structure component input file in APPENDIX 1V to arrive at the CANDU Fuel Bundle MODEL Input file in APPENDIX V.

We have previously describe in section 5.7.1 the sample input file for Heat Structure Component which is where we will reference and modify to input the most of the parameters in Table 6.1 to reflect the CANDU Fuel Bundle we want to simulating in this case.

- **Number of Pins**: WORD 5 of CARD 11250501/CARD 11250601 for left and right boundary condition (Heat transfer direction) describe the height of the heat structure, in this case, the fuel bundle (0.5 m X 37 pins) which gives us 18.5 m .

```
**left_BC  volume_num  increment  BCtype  SA type  height   hs.num
11250501        0           0       0         1      0.5      12
**left_BC  volume_num  increment  BCtype  SA type  height   hs.num
11250601    125010000   010000      1         1      18.5     12
```

- **Heat Transfer Hydraulic Diameter:** Another important parameter is the heat transfer hydraulic Diameter which goes into WORD 1 of CARD 11250800 and CARD 11250901 of the Heat structure component as already describe in section 5.7.1

```
*          ht.hyd.d.    htd.l.fr.   grd       boil.fctr(1.0 no power)
11250800   0
11250801   0.0123        10.0        10.0   0.0  0.0   0.0    0.0     1.0    12
11250900   0
11250901   0.0123        10.0        10.0   0.0  0.0   0.0    0.0     1.0    12
```

- **Flow Area**: Like in other cases, the flow arear goes in WORD 1 of CARD 1250101 which is the pipe component as shown below

```
*          flowArea   vol no.
1250101    0.0043     12
```

6.2.2.2 Simulation Cases

Case 6.1: Steady State Scenerio in a CANDDU Fuel Channel

In this case we provide an example of boundary condition inputs that represent that of a 37Pins by 12 CANDU fuel bundles with a set flow rate, pressure, and fuel bundle operating parameters. The aim is to determine the fluid/wall temperature and pressure drop in a channelstainless steel pipe during steady state. The nodalization diagram is given in Figure 5.9a and the sample input file for the reference case is provided Appendix V.

The case inputs are summarized as:

*--

*********** Input Data**********

*---

*Mass flow rate 20 kg/s *Pipe diameter=0.06 m

*Pipe length=6 m *Inlet temp = 513K

*Roughness = 0.00001 *Number of fuel elements = 37

*Pressure = 8MP (reactor operating pressure)

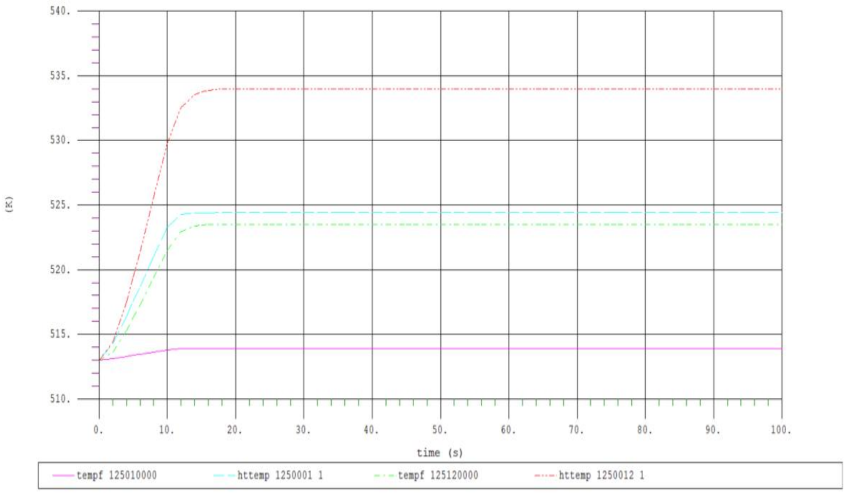


Figure 6.37 Steady state Fluid and Wall Temperature of CANDU Fuel

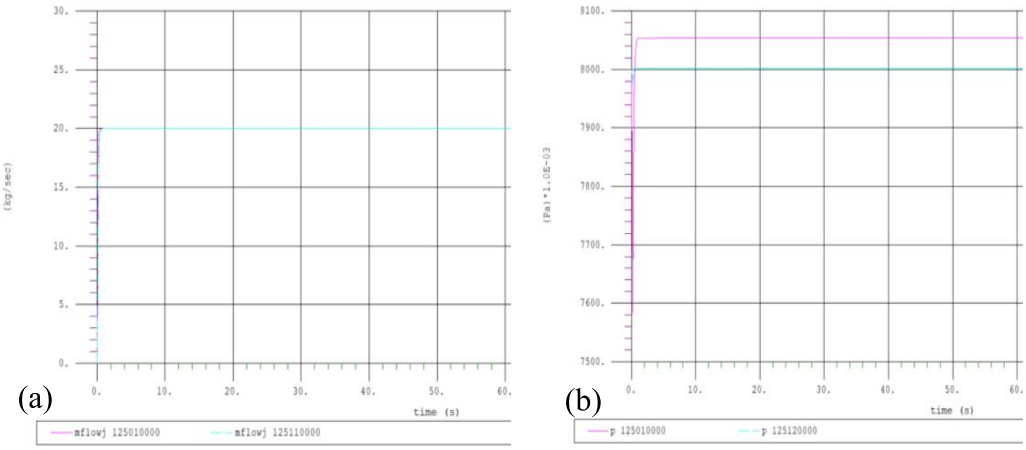


Figure 6.38 Steady State plot of (a) Flow rate (b) Pressure in Fuel Channel

Case 6.2: Transient Simulation (Power Surge 1.0 MW to 5.0 MW, 8 MPa, 20 kg/s)

In this case we provide an example of transition of case 6.1 into transient case where the 1 MW steady run was ramped to 5MW to represent a sudden power surge in reactor operation**.** The nodalization diagram is same as in Figure 5.9a and the Power table (from Line) of the sample input file in Appendix VI was replaced with Table 6.1 below and can be edited to reflect other range of power.

Table 6.2 Power Table for Transient Simulation (1MW-5MW)

```
-------------------------------------------
****** Power Table********************
-----------------------------------------------------
20214000 power
20214001 0.0  0.0
20214002 10.0 1000000.0
20214003 40.0 1000000.0
20214004 50.0 5000000.0
```

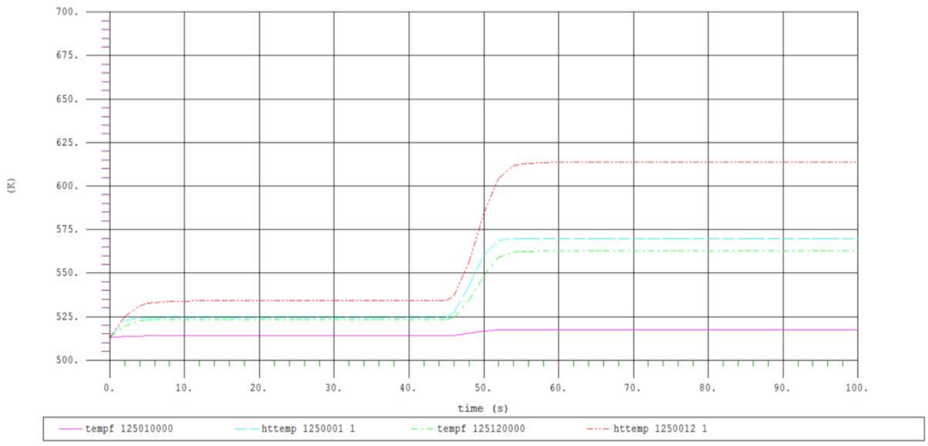


Figure 6.39 Temperature trend during power surge ramp

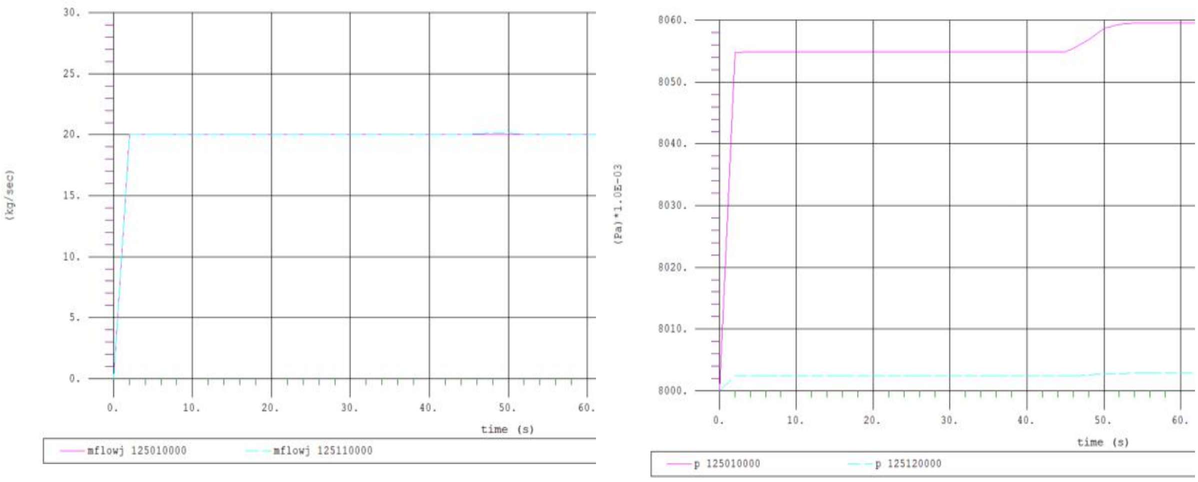


Figure 6.40 Power surge transient (a) Flow rate (b) Pressure Drop in Fuel Channel Model

Both the fluid and wall temperature are observed to follow accordingly when the heating power was ramped up as depicted in the code output shown in Figure 6.39 while the pressure drop experienced is shown in Figure 6.40b. Figure 6.40a present the set flow rate.

Case 6.3: Transient Simulation of Power fluctuation at 8 MPa, 20 kg/s)

In this case, the power variation in Table 6.2 was use to demonstrate power fluctuation scenarios. The user should note that RELAP employs a linear interpolation of the power between each time step in the power table below. All cases involve single-phase flow such that the impact of the power changes on the pressure drop/flow characteristic are minimal as shown in Figure 6.32.

Table 6.3 Power Variation Table

```
*------------------------------------------
******Power Table********
*------------------------------------------
20212500      power
20212501      0.0      0.0
20212502      10.0     1000000.0
20212503      20.0     1000000.0
20212504      30.0     200000.0
20212502      40.0     200000.0
20212503      50.0     2000000.0
20212504      60.0     2000000.0
20212502      70.0     500000.0
20212503      80.0     500000.0
20212504      90.0     5000000.0
20212502      100.0     5000000.0
20212503      110.0    500000.0
20212504      120.0    500000.0
20212503      130.0    2000000.0
20212504      140.0    2000000.0
```

The trend of the wall and fluid temperature during this fluctuation is presented in Figure 6.31while the corresponding pressure drops is shown in Figure 6.32 It is observed that the rise and fall of the temperature follows the increase and decrease of the heating power periods though we are to take recognizance of the system response time in immediately effecting this changes fluctuation which is captured in the time column of the power table above.

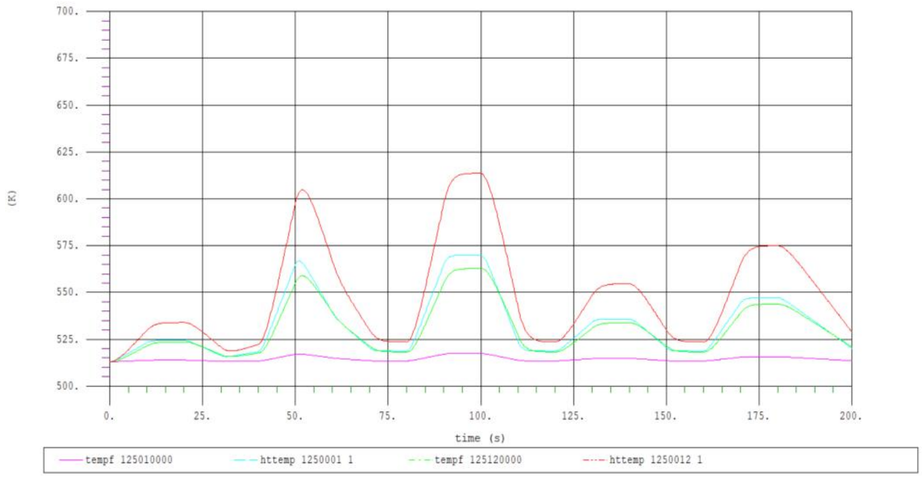


Figure 6.41 Fluid and Wall Temperature during Power Fluctuation Transient

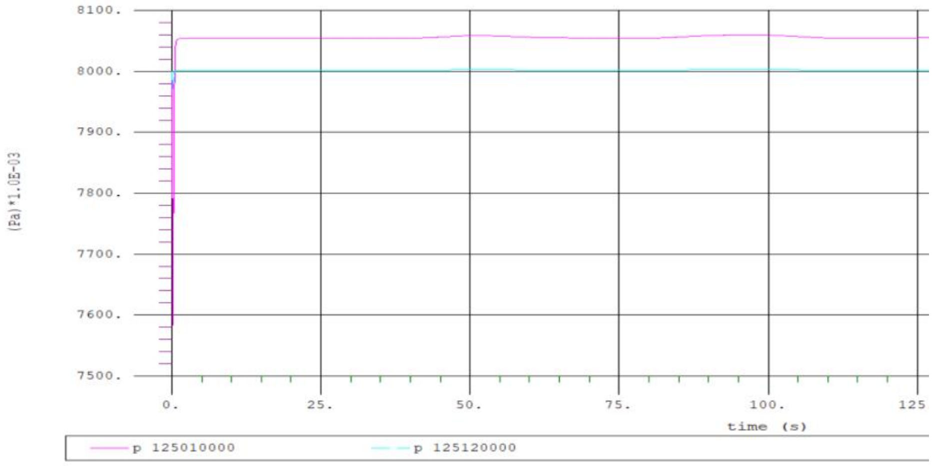


Figure 6.42 Pressure Drop during Power Fluctuation

6.3 Pump Model Cases

As mentioned previously in chapter 5, pump components are a key feature of a system model and are integral in producing accurate system responses. Even though the pump manufacturer usually provides the pump characteristics, knowledge on how to translate this into appropriate RELAP inputs. One important source in understanding pump selection is the pump characteristics which has been discussed in the previous chapter.

The required parameter needed form the pump manufacturer and the peculiar pump characteristic curve are as follow;

The Pump Power, Pump Head, Flow rate, RPM (Revolution per Minute) Efficiency, Impeller size (Relates to centrifugal and positive displacement pumps., NPSH (Net Positive Suction Head.

6.3.1 Pump Properties Defined

```
*----------------------------------------------------------------------
*******Pump Properties*******
*     westinghouse homologous pump curves
*     flow area = 0.07 sqm, volume = 0.14 cum
*     ratings:  speed = 369 rpm                 flow = 0.81 cum/sec
*                head = 27.5 m                torque = 500.0 nm
*              mom.i. = 1.43 kg*sqm          density = 614.0 kg/cum
*     actual speed = 369 rpm
```

Figure 6.43 Pump Characteristics

The above pump is chosen for the next case as it's properties capable of overcoming the pressure head we considering, the following cases were modeled and calculated for both Rough and Smooth pipe.

6.3.2 CASE #7: Pump Model Result (Rough vs Smooth Pipes)

In this case we provide an example of boundary condition inputs that represent a pump powered flow (forced circulation) through both series pipe connection and an inclined pipe connection. The aim is to simulate the flow condition and observe the flow behaviour while extracting the code calculation of the flow rate and pressure head require to push the fluid through both a smooth and rough pipes at each section of the network.

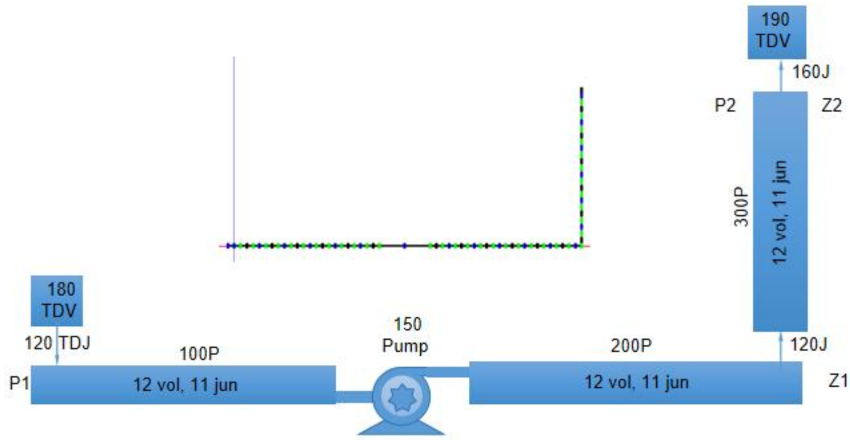


Figure 6.44 Pump Model Representation

Case 7.1 Forced (Pump) flow through rough Pipes

The nodalization diagram is drawn in Figure 5.13 and 6.33 and the sample input file for the reference case is provided Appendix VI. The known pump input parameters are summarized as;

*---

*********** Input Data**********

*--

*Pipe diameter = 0.076 m

*Pipe Length=6 m each

*Inlet temp = 300 K

*Roughness = 0.0018 (Rough pipes)

*Pressure P1= 2.1MP (Ground elevation)

*Pressure P2 = 2.0 MP (Same elevation as P1)

*Pressure P3 = 2.0 MP (Elevated 2 m above P1 & P2)

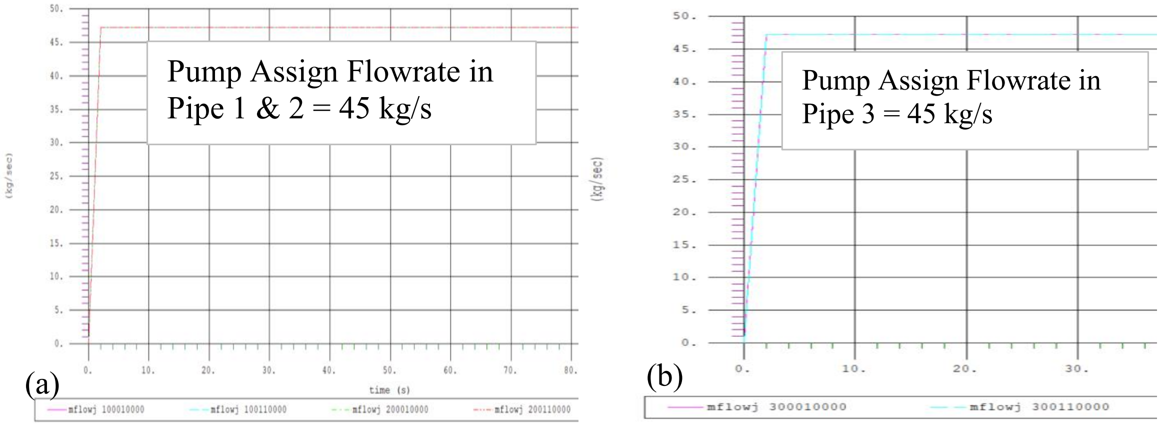


Figure 6.45 Flow rate set by the pump across (a) Pipe 1&2 (b) Pipe 3

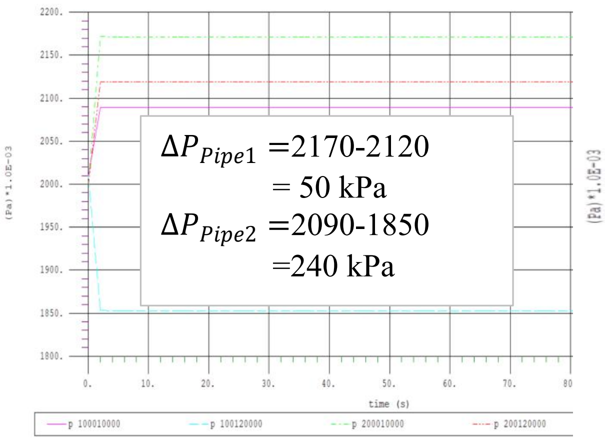


Figure 6.46 Pressure drop between rough pipes 1 and 2 on the same elevation

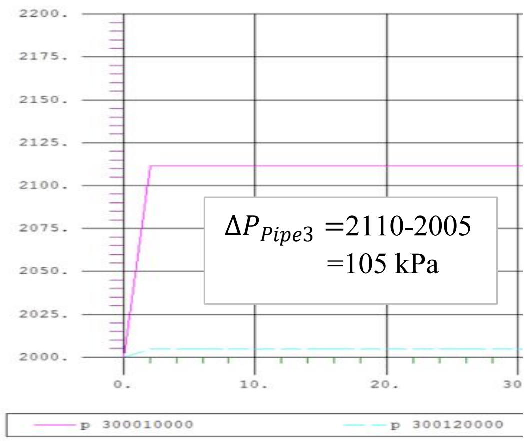


Figure 6.47 Final pressure in pipe 3 at higher elevation

6.3.3 Case 7 Sensitivity Analysis

Case 7.2: Forced flow through smooth Pipes

The nodalization diagram is same as in case 7.1 For the sample input file, All 3 rough pipes are replace with (0.0) to reflect smoothness. This edit is in WORD 1 of CARDS 1000801, 20008010 and 30008010 in the referenced case of Appendix VI. The known pump input parameters are summarized in case 7.1 with only the Roughness change to 0.0 to represent a smooth pipe.

```
*          rough      hyd vol no.
1000801    0.0     0.0   12
*          rough      hyd vol no.
2000801    0.0     0.0   12
*          rough      hyd vol no.
3000801    0.0     0.0   12
```

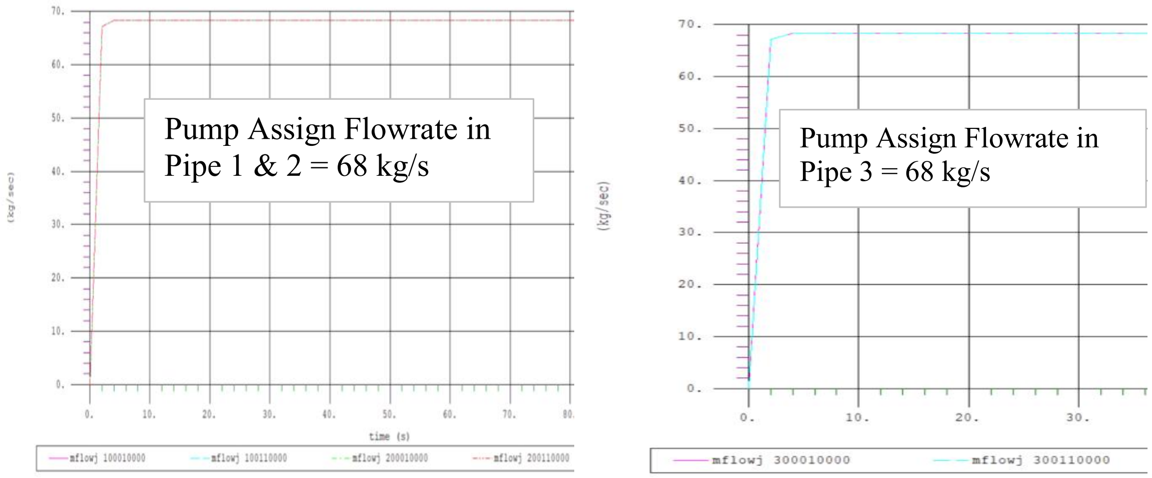


Figure 6.48 Flowrate in Smooth (a) Pipe 1 and 2 on same elevation (b) Pipe 3

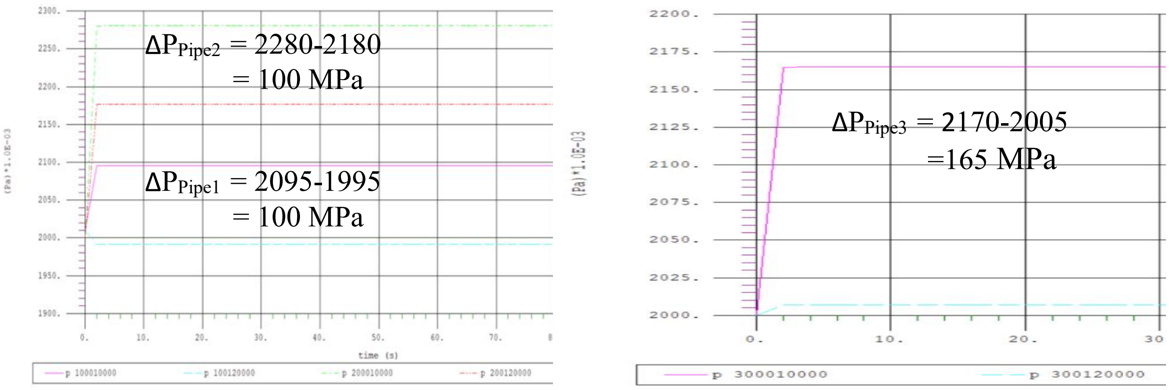


Figure 6.49 Pressure drop in Smooth (a) Pipe 1 and 2 on same elevation (b) Pipe 3

User will notice a significant parameters drop in flow rate between the smooth (Figure 6.39 & 6.40) and rough (Figure 6.41 & 6.42) Pipes which aligns with the theoretical and expected values.

6.4 Loop Model Cases

In order to further demonstrate the Pump model a case including flow in a loop is demonstrate using both the Pump input card, as well as a time dependent junction, to independently verify the pump component model.

6.4.1 CASE #8: Loop with a Pump

In this case we further provide an example of boundary condition inputs that represent a pump powered flow (forced circulation) through a simple loop. The aim is to simulate the flow condition and observe the flow behaviour while extracting the code calculation of the flow rate through the 4 pipe loop. The nodalization diagram is drawn in Figure 4.13a and the sample input file for the reference case is provided in Appendix VII. The loop and pump parameters are summarized as;

```
*-----------------------------------------------------------------
*********** Input Parameter**********
*-----------------------------------------------------------------
*Mass flow rate 20 kg/s
*Pipes diameter=0.06 m
*Pipes Length=30 m
*Inlet temp = 513K
*quality = 0.00001
*Pressure = 1.5 MP
```

```
*------------------------------------------------------------------------
*                        component 170
*------------------------------------------------------------------------
* pump
*     westinghouse homologous pump curves
*     flow area = 0.07 sqm, volume = 0.14 cum
*     ratings:  speed = 369 rpm                  flow = 0.81 cum/sec
*                head = 27.5 m                 torque = 500.0 nm
*              mom.i. = 1.43 kg*sqm           density = 614.0 kg/cum
*     actual speed = 369 rpm
```

The assigned pump flow rate through the 4 pipes loop as calculated by the code is plotted in Figure 6.50, while the pressure trend is in Figure 6.51

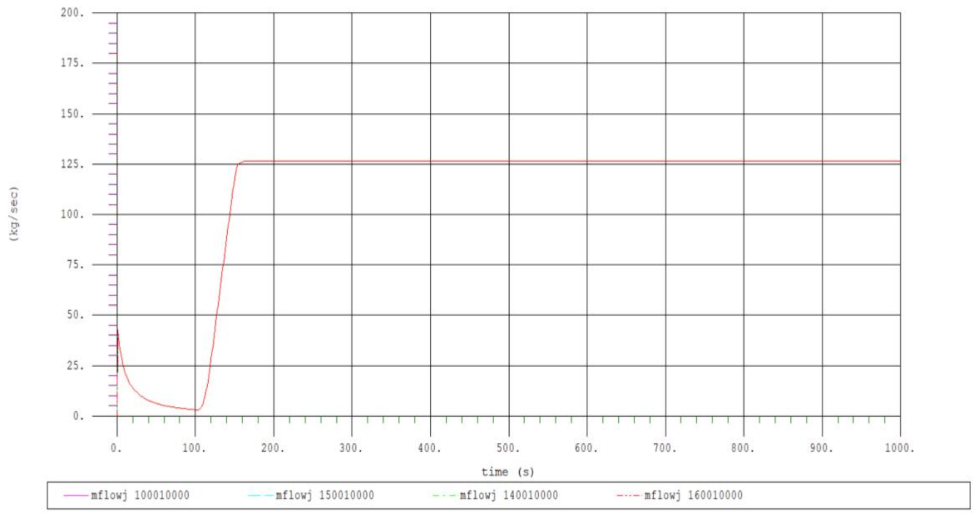


Figure 6.50 Loop Flow rate set by the Pump

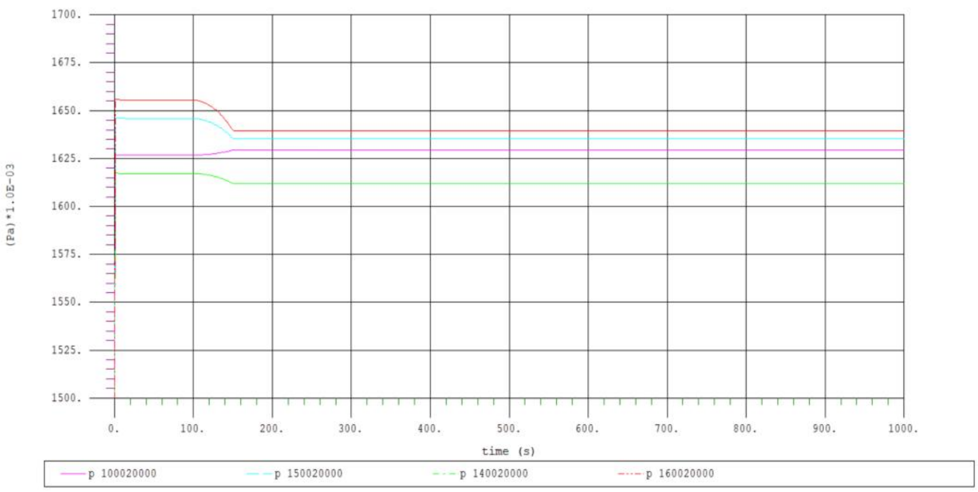


Figure 6.51 Pressure trend across the Loop (Pump powered)

6.4.2 Case #8 Sensitivity Analysis

Case 8.1: With TMDJUN:

Similar to Case #8, The aim in this sensitivity analysis case is replace the Pump component with a Time Dependent Junction and simulate the entire flow condition and in particular observe TDJ sustained flow rate through the 4 pipe loop. In this case and unlike having the pump generated flowrate we have to impose a flow rate for the TDJ so in this case we are going to be using the flow rate generated by the pump in case #8 (120 kg/s) to simulate the loop with TDJ. The nodalization diagram is drawn in Figure 4.13a and the sample input file for the reference case is to entirely replace the pump component input file with that of the Time Dependent Junction provided in Appendix VII.b with the below

```
*-------------------------------------------------------
*******Component 170 = tmdpjun*******
*-------------------------------------------------------
1700000   "IC"  tmdpjun
*         from vol    to vol     f.area
1700101   160020002   100010001   0.0196
*          vel/flow    trip*
1700200       1          0
*         time   liq.mflow  vap.mflow     int.vel.
1700201   0.0       120.0         0.0          0.0
*-------------------------------------------------------
```

The loop and pump parameters are same as in Case #8

The result of simulation is presented in Figure 6.52 which depict the outcome of the Time Dependent Junction sustained flow rate just like the Pump components while Figure 6.53 show the pressure losses.

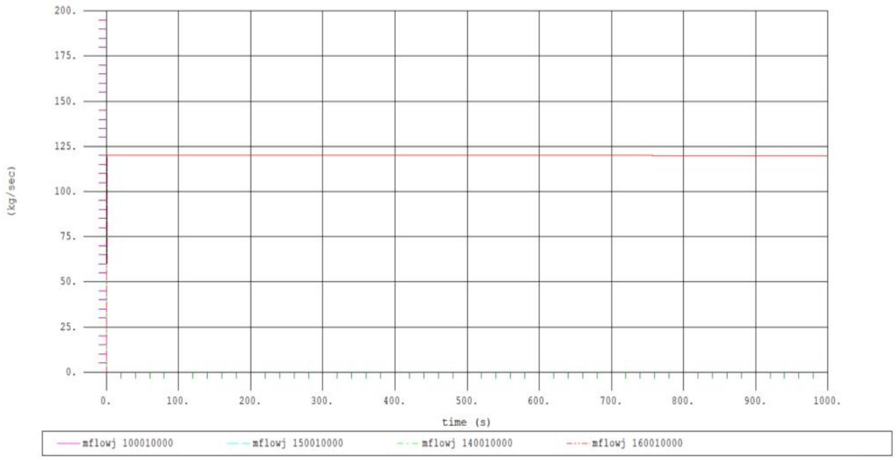


Figure 6.52 Loop Flow rate set by TDJ on heated loop

(a) both pipes (b) One pipe

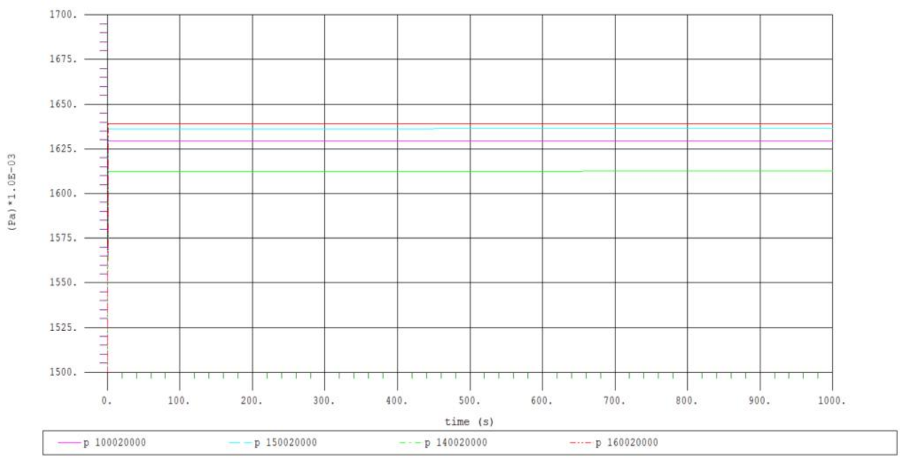


Figure 6.53 Pressure across Loop hydraulic path with TDJ

6.5 Heat Exchangers

The final example for new user training in the Workbook is an example of a heat exchanger application. The left and right boundary condition of the Heat structure is employed to effectively transfer heat from a heated pipe to a separate cooler flow stream. The verification of this simulation is via several cases of both a Parallel and Cross flow HX at reducing and increasing pipe length, increased flow on one side and also varying pipes roughness. A parallel and cross flow heat exchanger is analysed in this workbook with different cases to give detailed understanding of phenomena in the HX. New users can then compare their RELAP simulations to the theoretical predictions for these HX.

RELAP and ASYST have a large amount of flexibility in terms of heat exchange simulations. Because the flow area, hydraulic diameter, and heat transfer areas on the hot and cold side of the structure can be independently modified, a user can simulate a variety of HX designs by ensuring the each value is adjusted accordingly. Heat transfer correlations can also be selected that are most appropriate for the fluid flow and geometry being considered.

6.5.1 CASE #9: The Parallel Flow Heat Exchanger

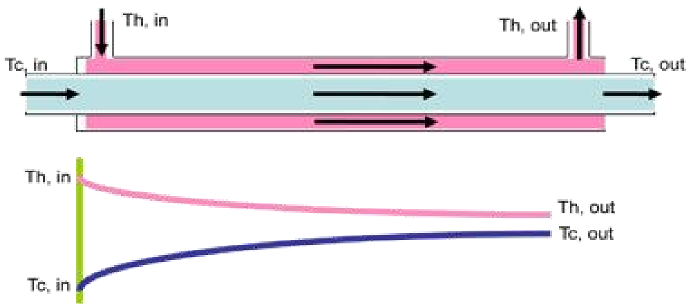


Figure 6.54 Schematic of a Typical Parallel HX adopted from [80]

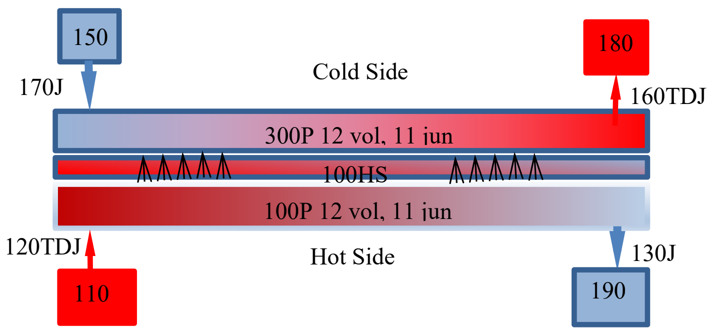


Figure 6.55 Schematic of a typical Parallel Flow Heat Exchanger Profile

Case 9.1. Parallel flow on a 6 m Heat Exchanger pipes

In this case we provide an example of boundary condition inputs that represent a parallel heat exchanger with a known inlet temperature of both the hot and code pipe. The aim is to simulate the model to determine **th**e outlet fluid/wall temperature of both side of the HX to confirm heat balance according to 1st law. The nodalization diagram is given in Figures 5.11a and the schematic is shown in Figure 6.54 while the sample input file for the reference case is provided Appendix VIII. The case inputs are summarized as:

*---

*********** Input Data**********

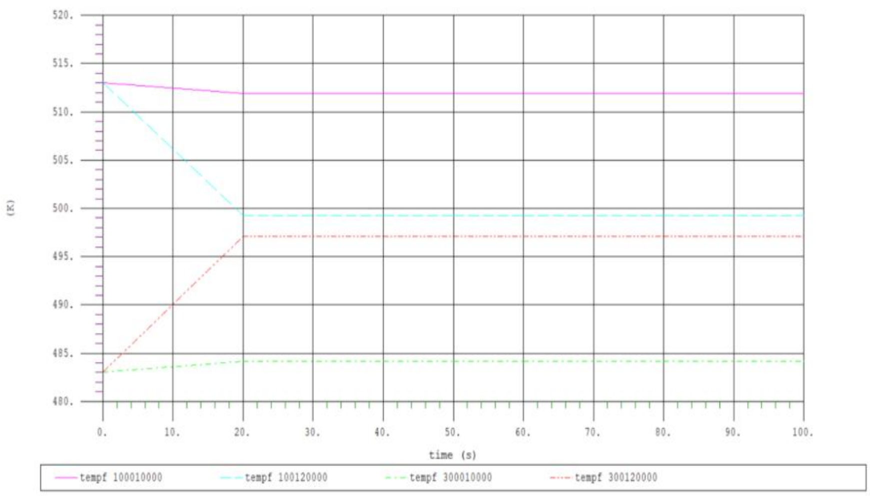
*---

*Pipe diameter = 0.06m *Pipe Length=6m each

*Inlet temp = 513K (hot leg) *Inlet temp = 483K (cold leg)

*Flowrate = 20 kg/s *Pressure = 10 MPa

*Roughness = 0.00001



6.56 Inlet and outlet Temperature Plot for both Hot and Cold Side

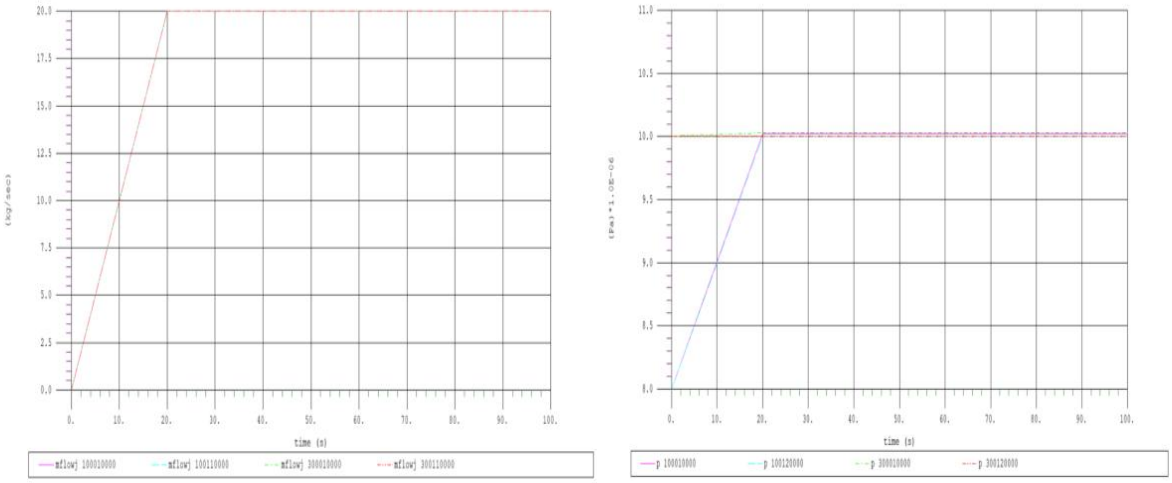


Figure 6.57 Plot of (a) Flow rate (b) Pressure trend on both Sides

Table 6.4 Case 9.1: Summary of Temperature Distribution

$T_{in,h}$	$T_{in,c}$	$T_{out,h}$	$T_{out,c}$	ΔT_h	ΔT_c
512.5	484	499	497	-13.5	13

Flow rate and Pressure on both sides maintained the set value of 20 kg/s and 10 MPa respectively while the temperature distribution and change in temperature on each pipe of the HX is tabulated in Table 6.6 which satisfies the heat balance on both side of the HX according to 1st law of thermodynamic.

6.5.2 Case 9 Sensitivity Analysis

Case 9.2: Increased Flowrate on the Hot Side

This case provide an example of boundary condition similar to case 9 with increased flow rate on the hot side. The aim is to examine the sensitivity of the heat exchange to flow rates the mass flow on the hot leg was doubled and the results provided.

The nodalization diagram and the sample input file for this case is same as mention in case # 9 in APPENDIX VIII but with WORD 2 of CARD 1600201 edited (changed to 40.0) while keeping the cold side flow rate constant at 20.0 to reflect the desired flowrate.

```
*          time   liq.mflow   vap.mflow      int.vel.
1600201    0.0       40.0         0.0           0.0
```

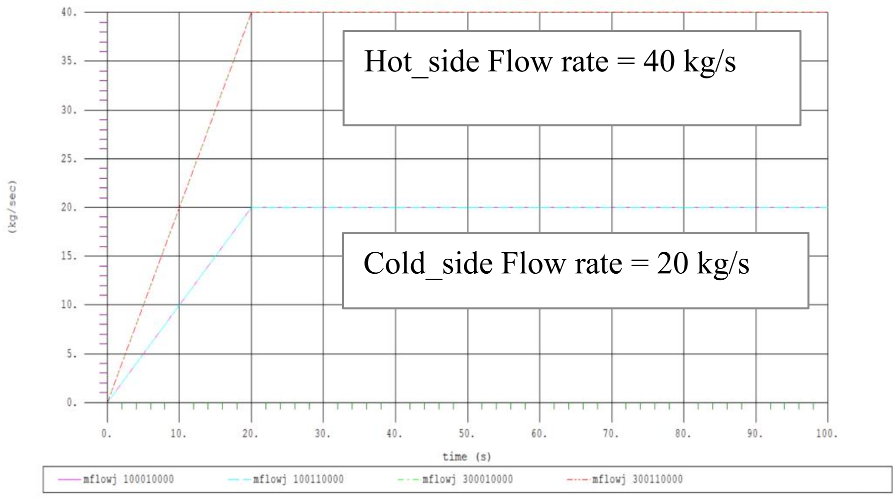


Figure 6.58 Corresponding Flowrates on each side

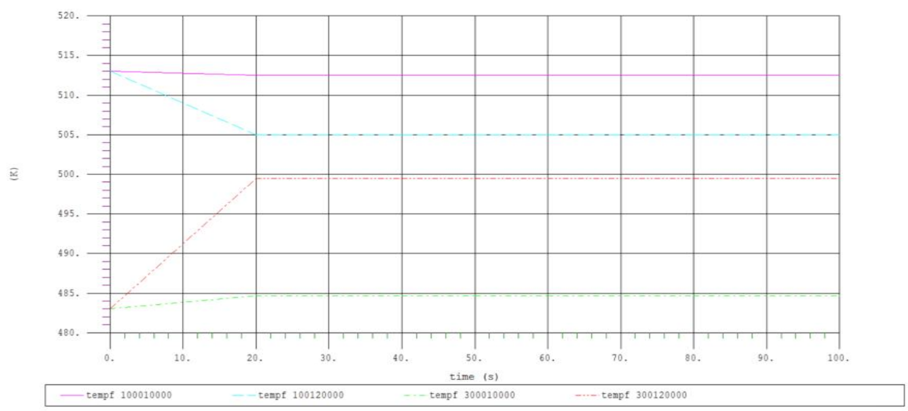


Figure 6.59 Temperature trend at a increased flowrate on Hot leg

The results show that the hot side temperature difference is reduced considerably due to the higher flow rate on that side. Furthermore, the total heat flow from hot to cold is also modestly increased due to the increases in convective coefficient associated with the increase in hot side flows. Both effects are as expected.

Table 6.5 CAse 9.2 Summary of Temperature Distribution

$T_{in,h}$	$T_{in,c}$	$T_{out,h}$	$T_{out,c}$	ΔT_h	ΔT_c
513	485	505	499.5	-8	14.5

Case 9.3: Increased Flowrate on the Cold Side

This case provide an example of boundary condition similar to case 9 with increased flow rate on the hot side. The aim is to examine the sensitivity of the heat exchange to flow rates the mass flow on the hot leg was doubled and the results provided.

The nodalization diagram and the sample input file for this case is same as mention in case 9.1 in APPENDIX VIII but with WORD 2 of CARD 1200201 in the TDJ component edited (changed to 40.0) while keeping that of the cold side flow rate constant at 20.0 to reflect the desired flowrate.

```
*          time   liq.mflow   vap.mflow     int.vel.
1200201    0.0      20.0         0.0          0.0
```

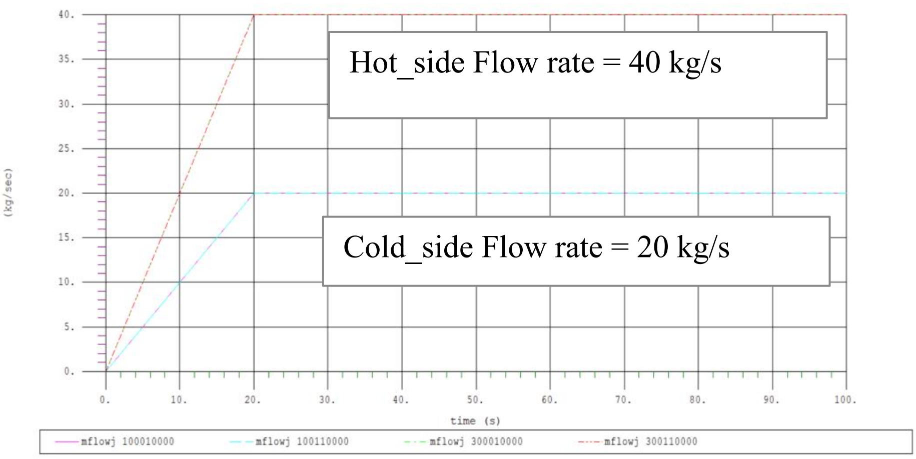


Figure 6.60 Flowrates on each side

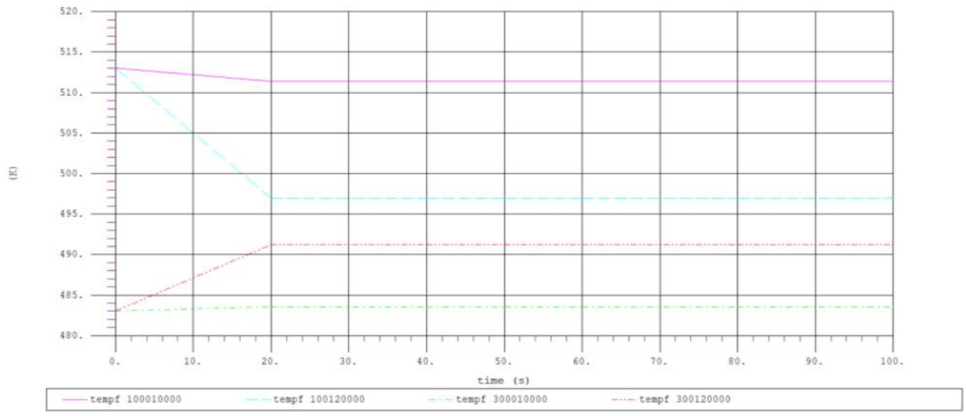


Figure 6.61 Temperature trend at an increased flowrate on Hot leg

For the case with the higher cold side flow we observe that the delta-Temperature gradient across the cold side is decreased as expected. Similar to the previous case there is also a small increase in the total heat transferred arising from the higher cold side flow rate (i.e., increase in HX efficiency).

Table 6.6 Case 9.3 Summary of Temperature Distribution

$T_{in,h}$	$T_{in,c}$	$T_{out,h}$	$T_{out,c}$	ΔT_h	ΔT_c
512	483	497	491	-15	8

Case 9.3 Elongated Length on both Legs

Similar to the previous case, we provide an example of boundary condition that portrays an elongated pipe length of both sides. The aim is to examine the sensitivity of the heat exchanger's effectiveness on two times the original length (12m).

Nodalization is same while the sample input file in APPENDIX VIII was edited by changing Word 1 of CARD 1000301 for the hot side and CARD 3000301 for the cold side to 1.0 which when multiplied by the vol. Number (12) will reflect the increased length.

```
64   *          length   vol no.
65   1000301     1.0      12
131  *          length   vol no.
132  3000301     1.0      12
```

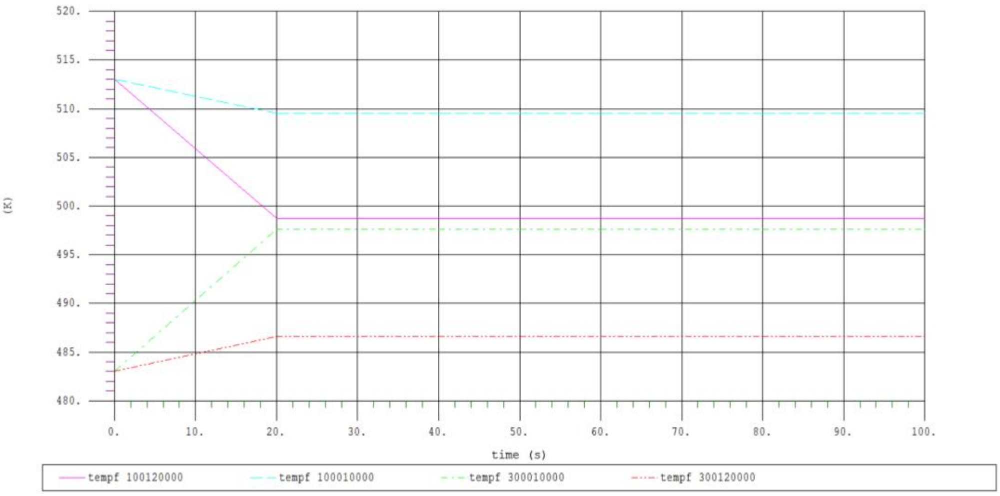


Figure 6:62 Increased effectiveness in heat transfer due to elongated HX

Case

Again, like the previous case, we provide an example of boundary condition that portrays a shorter/reduced pipe length of both sides by half. The aim is to examine the sensitivity of the heat exchanger's effectiveness on half the original length (3m).

Nodalization is same while the sample input file in APPENDIX was edited by changing Word 1 of CARD 1000301 for the hot side and CARD 3000301 for the cold side to 1.0 which when multiplied by the vol. Number (12) will reflect the increased length.

```
64   *          length   vol no.
65   1000301    0.25     12

131  *          length   vol no.
132  3000301    0.25     12
```

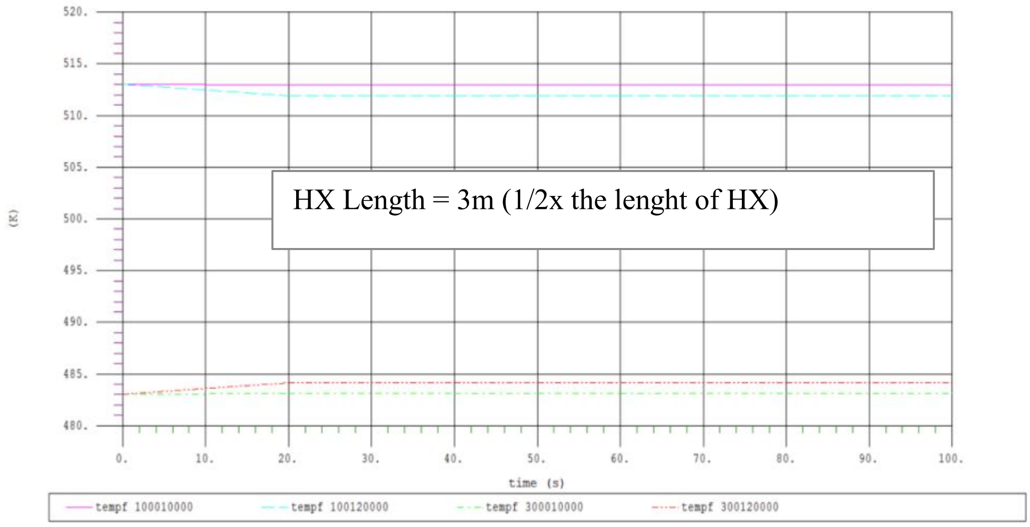


Figure 6.63 Inefficient heat transfer due to reduced HX length

6.5.3 CASE #10: Counter-Flow Heat Exchanger

The same operational condition as applied in the parallel flow was adopted to test the flow in a cross flow HX, and the following cases were simulated. Here what is expected is an increase in the efficiency of the HX arising from the higher efficiency (in general) for counter-flow arrangements.

Case 10.1 : Counter flow on a 6 m Heat Exchanger pipes

In this case we provide an example of boundary condition inputs that represent a Counter flow heat exchanger with same parameter as the parallel case 9. The basic distinction is the direction of flow which in this case one of the pipe's flow direction is in the opposite direction (reversed). The aim is to simulate and compare it efficiency with the parallel flow condition, The nodalization diagram is shown in Figure 5.11b and the schematic in Figure 6.55 while the sample input that was edited is CARD 3000501 (either of the pipes flow direction can be reversed to effect a counter flow by changing the azimothal angle in WORD 1 to -180.

```
133    *         az. ang vol no
134    3000501   -180.0     12
```

The expected flow direction is shown in Figure 6.64

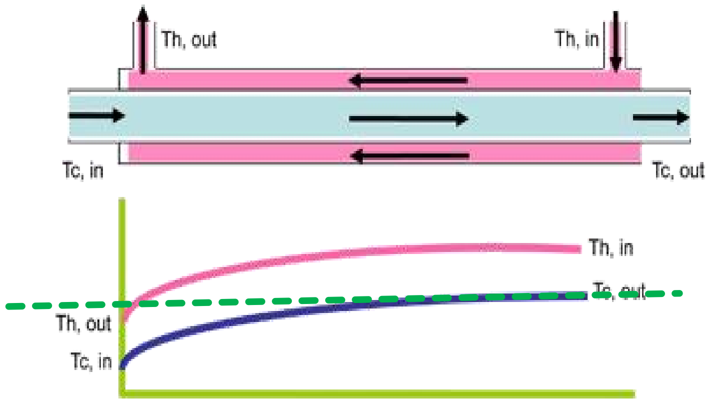


Figure 6.64 Expected Flow Direction of Counter HX adopted from [80]

- **Our model and result of simulation**

The same operational conditions as applied in the parallel flow was adopted to test the flow in a counter flow HX, and the following cases were simulated

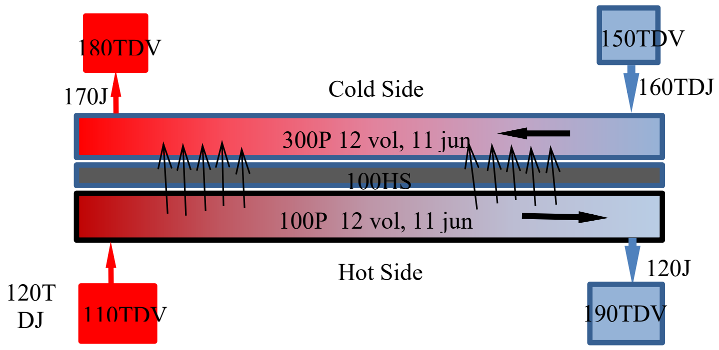


Figure 6.65 Schematic of a typical Counter Flow Heat Exchanger

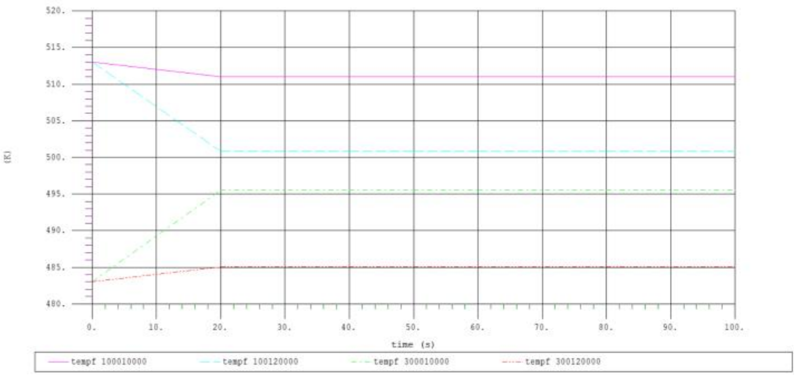


Figure 6.66 Trend of temperature distribution for 6m HX

Table 6.7 Case 10.1 Summary of Temperature Distribution

$T_{in,h}$	$T_{in,c}$	$T_{out,h}$	$T_{out,c}$	ΔT_h	ΔT_c
512.5	484	502	495.5	-10.5	11.5

6.5.4 Case 10 Sensitivity Analysis

Case 10.2 : Counter flow on a 12 m Heat Exchanger pipes

This case follows same approach as the sensitivity analysis of the parallel case. We are going to repeat one of the sensitivity analysis case on this counter current HX as the other cases is simply changing of the flowrate and/or the length. The aim here is to simulate and compare it efficiency for an elongated pipe length. The nodalization diagram is shown in Figure 5.11b and the schematic in Figure 6.65 while the sample input file will be of Case 10.2 with WORD 1 of CARD 1000301 for the hot side and CARD 3000301 for the cold side to 1.0 which when multiplied by the volume number (12) will reflect the increased length.

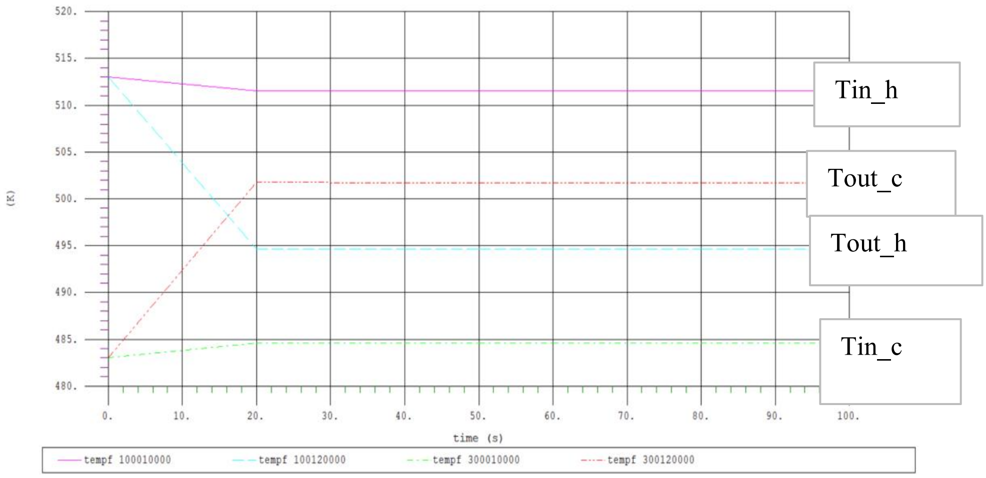


Figure 6.67 Trend of temperature distribution for 12m HX

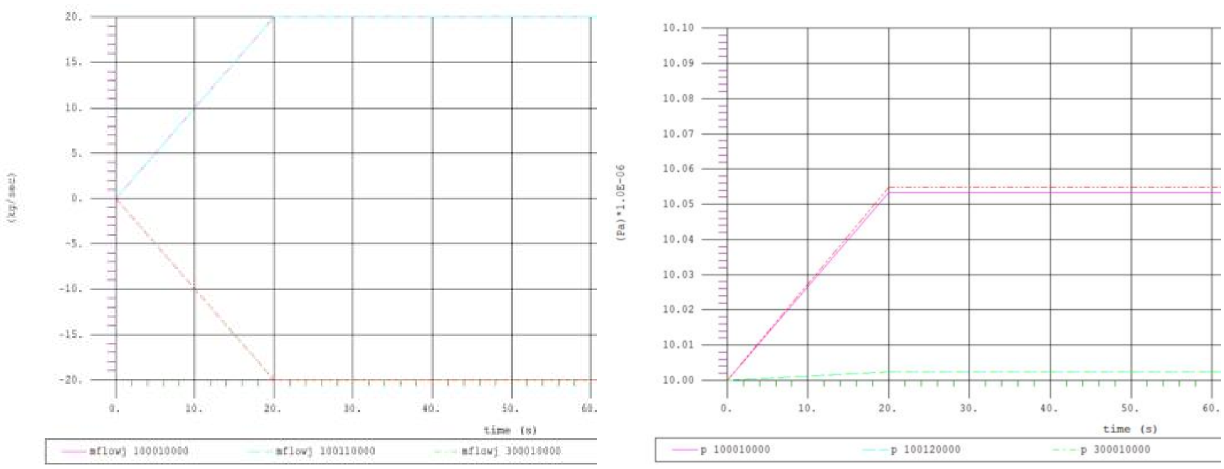


Figure 6.68 Trend of flowrate and pressure

Table 6.8 Case 10.2 Summary of Temperature Distribution

$T_{in,h}$	$T_{in,c}$	$T_{out,h}$	$T_{out,c}$	ΔT_h	ΔT_c
512.5	484	495	502	-13.5	13

From Figure 6.52 and Table 6.7, it will be observed that the efficiency of the counter current flow is higher than that of the parallel model especially at an elongated pipes this is attributed to the the higher effectiveness of the counter flow HX at NTU higher than 2 as compare to the parallel effectiveness which stays contant at 0.5 from NTU Higher than 2. The result are verify theoretically with good agreement.

6.5.5 Analytical Verification

For the analytical Calculation, the length (6m) in sheet 1 is simply replace by 12 and 3 respectively. It was observed that due to the flattening of the C_min/C_max beginning from NTU = 2 mark upward, the effectiveness stays at 0.5 which has an effect in determining the rate of heat transfer.

To determine Heat transfer rate and Outlet Temps of a Parallel flow Heat Exchanger, we recall 1st law of thermodynamic:

$$q = mC_p(T_{in} - T_{out})$$

With could easily calculate the change in Temperatures using Equation 6. above, but the respective q on both sides are unknown ;

$$q_h = mC_{p\_h}(T_{in} - T_{out}) = ?$$

$$q_c = mC_{p\_c}(T_{in} - T_{out}) = ?$$

Table 6.9 Initial HX known Parameters

Flowrate	Cp_h	Cp_c	$T_{in,h}$	$T_{in,c}$	L	D	A	X
20	4.77	4.548	513	483	6	0.06	1.13	0.001

Due to the unknowns qh and qc, we employ the following correlation using the difference between the two known inlet temperatures multiplied by the Min of both specific heat capacities while introducing the efficiency factor, ε.

$$q_h = \varepsilon\, C_{p\_min}\left(T_{in,h} - T_{out}\right)$$

$$C_{p\_min} = \text{ is the lowest of the both, thus}$$

$$C_{p\_h} = C_{p\_min}$$

Where ε, is the efficiency of the HX determine using the following methods

1. By Effectiveness-NTU Curve Method

$$\varepsilon = f\left(\mathrm{NTU}, \frac{C_{p\_min}}{C_{p\_max}}\right)$$

NTU (Number of Transfer Units) =Defined in terms of the overall heat transfer coefficient, μ, the area of heat transfer, A, and the minimum heat capacity, C_{p_min} ,

$$\mathrm{NTU} = \mu * \frac{A}{C_{p,min}}$$

$$\text{Where, } \mu = A * \frac{k}{L}$$

Now we determine the ratio between the heat capacities of fluids,

$$C_r = \frac{C_{min}}{C_{max}}$$

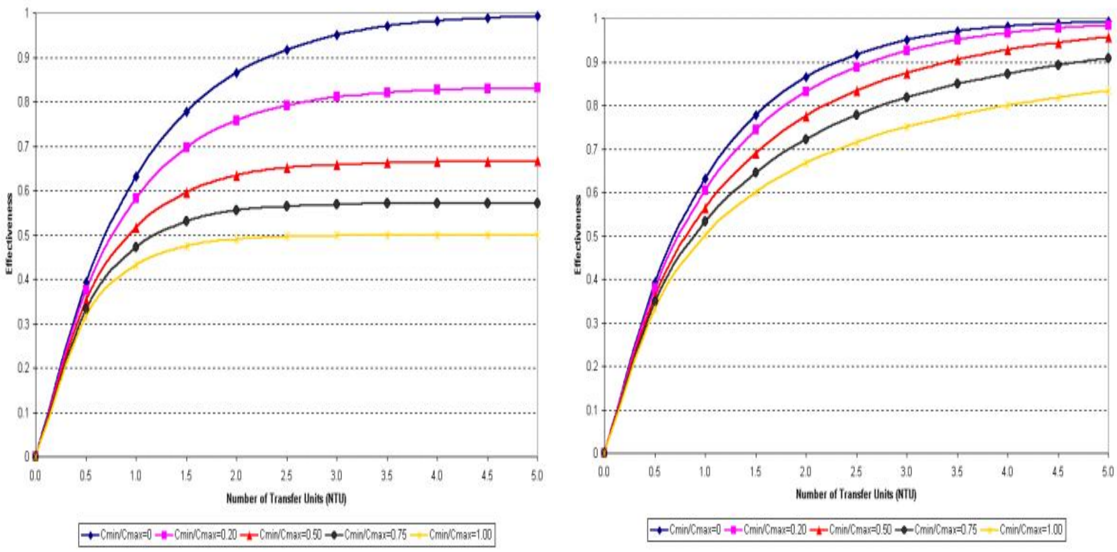


Figure 6.69 Heat Exchanger Effectiveness (a) Parallel Flow (b) Counter flow

Where $C_{p,min}$ corresponds to the minimum value of the heat capacities i.e, which is smaller. With NTU determined and C_r, know we can deduce e from the curve below

By Correlation

Effectiveness is an important parameter in determining the efficiency of the heat exchanger. It is a dimensionless indicator that relates the actual heat transfer rate (q) to the maximum possible heat transfer rate (q_max) that could occur for a particular heat exchanger and a particular set of fluids. With the values of NTU and C_{min}, use the ε formula or curves of the respective heat exchanger to calculate ε. The formula for the effectiveness of a heat exchanger is given by the rate of these heats:

$$\varepsilon = \frac{T_{h,\,i} - T_{h,o}}{T_{h,i} - T_{c,i}}$$

$$\frac{C_{p,min}}{C_{max}} = \frac{m_h C_{p,h}}{m_c C_{p,c}} = \frac{(T_{c,o} - T_{c,i})}{(T_{h,\,i} - T_{c,i})} = C_r$$

$$\varepsilon = \frac{1 - exp[-NTU(1 + C_r)]}{1 + C_r}$$

Solutions recording using both methods in MS excel is presented in Table 6.13

Table 6.10 Summary of correlation method

q_{max}	Efficiency (Method 1)	q	Efficiency (Method 2)	q_{hot}	q_{cold}	ΔT_{hot}	ΔT_{cold}
136.44	0.504	68.71	0.504	-1374.13	1374.13	-14.40	15.11

If q is know then we can use, a third method will be;

$$\varepsilon = \frac{q}{q_{max}}$$

The actual heat rate q is determined in terms of the associated properties can then be computed using;

$$q = C_c * \left(T - T_{c,i}\right)$$

$$q = C_h * \left(T_{h,i} - T_{h,o}\right)$$

In order to calculate the maximum possible heat transfer rate (qmax) that could happen in a given heat exchanger, the following expression is typically used:

$$Q_{max} = C_{min} * \left(T_{h,i} - T_{c,i}\right)$$

For the co-current and counter-current arrangements the analytical results come within 0.1C of the RELAP calcaultions showing strong agreement between the code and theoretical methods.

Chapter 7

Conclusion and Future Work

7.1 Conclusion

In this Workbook, we have developed a comprehensive approach for the use of RELAP5 and ASYST system codes for simulating the behavior of thermal hydraulic systems, particularly in nuclear power plants components. The workbook serves as a guide for new users and their training, providing step-by-step instructions, examples, and exercises to enhance understanding and proficiency in using these codes. In addition to this thesis the accompanying Excel sheets provide the detailed inputs for the code as well as the plotted results for users to compare their output.

Through the development process, several key objectives guided the creation of the workbook.

Firstly, we aimed to provide a clear and concise introduction to RELAP5 and ASYST, covering its background, capabilities, and applications. The introductory section sets the stage for users, helping them understand the significance of the codes in the field of thermal hydraulics.

Next, we delved into the theory governing the computational aspects of using RELAP5 to provide a background understanding of the fundamental physics employed in computing and solving problem to ensures that users are equip and comfortable using these powerful tool.

The core of the workbook lies in RELAP5/ASYST components descriptions, nodalization (block representation of the model components), input code development, Simulations and result analysis including sensitivity analysis and analytical calculations are included for each Case. We designed these sections to progressively introduce users to various aspects of these codes, from general simulation approach of basic geometry nodalization to inputs more representative of reactor operating conditions. Each case is structured in a step-by-step format, guiding users through the components modeling, input file creation, simulation execution, and result analysis. We included different cases as examples and sample input files to illustrate different scenarios and phenomena including steady-state and transient simulations. The basic input files are presented in the APPENDIX section.

Finally, the development of this workbook represents a significant contribution to the educational resources available for RELAP5 and ASYST. The workbook aims to empower users to harness the full potential of this powerful system codes by providing a structured and practical approach to learning these codes.

We believe that the workbook will serve as a valuable resource for students, researchers, and professionals in the field of thermal hydraulics, facilitating the understanding and application of RELAP5/ASYST in diverse engineering contexts.

7.2 Recommendation for Future Work

Looking ahead, there are several avenues for future development and improvement of the workbook.

- Firstly, as these system codes evolves and new versions are released, the workbook will need to be updated to reflect these changes.
- Secondly, the examples and simulations presented in this workbook are limited to single phase modeling to give new user the background require in venturing into the field of nuclear system analysis, additional advanced topics could be covered in future editions, catering to users with more specialized interests in multi-phase simulation.
- More representative components (e.g., CANDU feeder network), or channel grouping methods, should be included as users progressing to full system analysis will need these resources.
- Pump models can be quite complicated, including the homologous curves for all 4 potential quadrants of operations. The examples presented in this Workbook are quite simplistic in this regard and hence more representative pipe flow models should be included.
- Steam generator analysis, including recirculation ratios, downcomer flows, steam separators, etc.. are an integral part of nuclear power plant simulations. Due to their complexity they were not included in this Workbook, however detailed step-by-step modelling approaches would be quite valuable.
- A fundamental part of system codes are so called “trips” and “controllers”. These features allow the simulations to consider control system responses,

safety system actuation, and other user controls. While these represent advanced features of the codes, they are a fundamental requirement for almost all NPP modelling and hence should be included in future versions of this workbook.

Advanced components (e.g. Vessel, ECC, valves) could be developed and included in a more advanced user training Workbook.

APPENDIX: Developed Workbook Input Files

I. 1-D Pipe Flow

= Flow in a Single 30m pipe with no Heat Applied

*to investigates the pressure drop, mass flow rate and temp difference

*---

*********** initial conditions**********

*--

*Mass flow rate 10 litre/sec = =0.01kg/s

*Pipe diameter=6cm = 0.06 m

*Pipe Length=30 m

*Inlet temp = 288K

*quality = 0.00001

*---------------------------------------

***********Defining the problem*******

*--

100 new transnt

102 si si

105 1.0 2.0

110 air

115 1.0

*crdno end time min dt max dt control minor ed major ed restart

201 100.0 1.0E-6 0.1 5 20 50 2000

*--

*******Inlet control volume boundary conditions******************

```
*-------------------------------------------------------------------
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1100101   0.002827  6.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000
*Time dependent data to be pressure
*       ebt(fluid type, no-boron, initial thermodynamic state)
1100200  101
*       time    Temp (K) Quality
1100201  0.0      288.0   0.000001
*-------------------------------------------------------------------
*******Inlet Time Dependent Junction Properties*******
*-------------------------------------------------------------------
1200000  "inlet"  tmdpjun
*       from vol   to vol    f.area
1200101  110010002  125010001   0.0
*        vel/flow    trip*
1200200      1         0
*       time  liq.mflow  vap.mflow     int.vel.
1200201  0.0      10.0        0.0         0.0
*-------------------------------------------------------------------
*******Pipe Properties*******
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1250000  "pipe1"  pipe
*    no. of vols
1250001  6
*       flowArea  vol no.
1250101  0.002827    6
*       lengh   vol no.
1250301  6.0  6
*      az. ang vol no
```

1250501 0.0 6

* vt. ang vol no

1250601 0.0 6

* rough hyd vol no.

1250801 0.0058 0.0 6

* f loss r loss jun. no.

1250901 0.0 0.0 5

* vol flag vol no

1251001 0000000 6

* jun flag jun. no

1251101 0000000 5

* flag temp(K) vol.no.

1251201 101 288.0 0.000001 0.0 0.0 0.0 6

*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)

1251300 1

* liq.mflow vap.mflow int.vel. jun no.

1251301 0.0 0.0 0.0 5

*---

1270000 "sngljuno" sngljun

* from vol to vol f.area f.loss r.loss flag (fefvcahs)a=0 smooth, a=1 full abruot change

1270101 125060002 130010001 0.0 0.0 0.0 0000000

* flag liq.mflow vap.mflow int.vel.

1270201 1 0.0 0.0 0.0

*---

1300000 "sink" tmdpvol

* f.area l vol azi vert dz rough hyd d flags

1300101 0.002827 6.0 0.0 0.0 0.0 0.0 0.0 0.0 0000000

*Time dependent data to be pressure and temperature

1300200 101

* time Pressure(Pa) Quality

1300201 0.0 288.0 0.000001

```
*-------------------------------------------------------------------
*******Plot Requests*******
*----------------------------------------------------------------
20300021  mflowj 125010000 1
20300022  mflowj 125050000 1
*
20300011 p 125010000 1
20300012 p 125060000 1
*
20300031  tempf 125010000 1
20300032  tempf 125060000 1
. end
```

II.a. Connecting Pipes (Series Connection)

```
= Flow in 2 pipes of same length connected in series horizontally
*to investigates the pressure drop, mass flow rate and temp diff of both pipes
*--------------------------------------------------------------------
*********** Input Data *********
*----------------------------------------------------------------------
*Mass flow rate 10 kg/s
*Pipes diameter=6cm
*Pipes Length=30m each
*Inlet temp = 288K
*quality = 0.00001
*Inlet and outlet Pressures of both pipes are unknown
*---------------------------------------
***********Defining the problem*******
*----------------------------------------
100  new  transnt
102  si      si
```

```
105   1.0    2.0   1000.0
110  air
115  1.0
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201   100.0      1.0E-6    0.01     5        50      200       200
*-------------------------------------------------------------------
*******Inlet control volume boundary conditions*******************
*-------------------------------------------------------------------
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m) vol(m3)  azi  vert  dz rough hyd.d flags
1100101  0.002827    30.0     0.0     0.0  0.0  0.0 0.0001 0.06 0000000
*Time dependent data to be pressure
*       ebt(fluid, no-boron, initial thermodynamic state)
1100200  101
*       time   temp(K) Quality
1100201  0.0    288.0   0.00001
*----------------------------------------------------------------
*******Inlet Time Dependent Junction Properties*******
*----------------------------------------------------------------
1200000  "inlet"  tmdpjun
*       from vol   to vol    f.area
1200101  110010002  125010001   0.0
*        vel/flow    trip*
1200200     1        0
*        flag  liq.mflow  vap.mflow     int.vel.
1200201   0.0      10.00        0.0       0.0
*----------------------------------------------------------------
*******Pipe1Properties*******
*----------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1250000  "pipe1"  pipe
```

```
*       no. of vols
1250001  6
*       flowArea  vol no.
1250101  0.002827  6
*       lengh   vol no.
1250301   2.5    6
*       az.ang  vol no
1250501  0.0     6
*       vt.ang  vol no
1250601  0.0     6
*       rough   hyd vol no.
1250801  0.0018  0.0  6
*       f loss r loss jun. no.
1250901  0.0    0.0     5
*       vol flag  vol no
1251001  0000000    6
*       jun flag  jun. no
1251101  0000000    5
*      flag  temp(K)  quality                  vol.no.
1251201 101   288.0   0.00001    0.0   0.0   0.0  6
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
1251300    1
*    liq.mflow vap.mflow  int.vel.  jun no.
1251301  0.00    0.0        0.0        5
*-------------------------------------------------------------------
1270000  "sngljuno"  sngljun
*       from vol   to vol    f.area    f.loss    r.loss  flag (fefvcahs)a=0 smooth, a=1 full abruot change
1270101  125060002  135010001 0.002807   0.0     0.0     0000000
*     flag  liq.mflow  vap.mflow     int.vel.
1270201  1      10.0        0.0         0.0
*-------------------------------------------------------------------
```

*******Pipe2Properties*******

*6cm diameter 30 meter stainless steel pipe

1350000 "pipe1" pipe

*   no.of vols

1350001 6

*   flowArea vol no.

1350101 0.002827 6

*   lengh   vol no.

1350301 2.5   6

*   az. ang vol no

1350501 0.0   6

*   vt. ang vol no

1350601 0.0   6

*   rough   hyd vol no.

1350801 0.0018   0.0 6

*   f loss r loss jun. no.

1350901 0.0   0.0   5

*   vol flag vol no

1351001 0000000   6

*   jun flag jun. no

1351101 0000000   5

*   flag temp(K)   quality   vol.no.

1351201 101 288.0   0.00001   0.0   0.0   0.0 6

*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)

1351300   1

*   liq.mflow vap.mflow int.vel. jun no.

1351301 0.0   0.0   0.0   5

*--

1370000 "sngljuno" sngljun

*   from vol   to vol   f.area f.loss   r.loss   flag (fefvcahs)a=0 smooth, a=1 full abruot change

1370101 135060002 130010001 0.0   0.0   0.0   0000000

```
*    flag  liq.mflow  vap.mflow     int.vel.
1370201  1      10.0        0.0         0.0
*--------------------------------------------
1300000  "sink"  tmdpvol
*       f.area    l     vol   azi  vert  dz   rough hyd d flags
1300101  0.002827  100.0    0.0   0.0   0.0  0.0   0.0  0.0  0000000
*Time dependent data to be pressure and temperature
1300200  101
*       time  temp  quality
1300201  0.0   288.0  0.000
*-------------------------------------------------------------------
*******Plot Requests*******
*----------------------------------------------------------------
20300011 p 125010000 1
*20300012 p 125060000 1
*20300013 p 135010000 1
20300014 p 135060000 1
*
20300021  mflowj 125010000 1
20300022  mflowj 125050000 1
20300023  mflowj 135010000 1
20300024  mflowj 135050000 1
*
20300031  tempf 125010000 1
20300032  tempf 125060000 1
20300033  tempf 135010000 1
20300034  tempf 135060000 1
*
. end
```

II.b. Connecting Pipes (Parallel Connection)

```
*To investigates the flow parameter changes in a 2 Parallel connected Pipes
*-------------------------------------------------------------------
*********** Input Data**********
*------------------------------------------------------------------
*Mass flow rate 20 kg/s
*Pipe 1 diameter=0.06 m
*Pipe 2 diameter=0.06 m   *change to 0.03 for the unequal pipes
*Pipe Length=6 m
*Inlet temp = 513K
*quality = 0.00001
*Pressure = 8MP
*----------------------------------------
***********Defining the problem*******
*-----------------------------------------
= Multi-Channel
100  new  transnt
102  si      si
*============================================================
==================*
*                 cpu time remaining card
*============================================================
==================*
*crdno  time1  time2
105   1.0    2.0
110  air
115  1.0
*
```

```
*=====================================================================
==================*
*                time step control cards
*=====================================================================
==================*
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     100.0   1.0E-9     0.001    5       5000        5000     2000
*---------------------------------------------------------------------
*******Inlet control volume boundary conditions******************
*---------------------------------------------------------------------
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1100101   0.0028      6.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000*check
*Time dependent data to be pressure
*       ebt(fluid type, no-boron, initial thermodynamic state)
1100200  103
*       time       Pa                          K
1100201  0.0        8000000.0   513.0
*---------------------------------------------------------------------
*******Inlet Time Dependent Junction Properties*******
*---------------------------------------------------------------------
1200000  "inlet"  tmdpjun
*       from vol   to vol    f.area
1200101  110010002  150010001  0.0028
*        vel/flow    trip*
1200200     1        0
*       time  liq.mflow  vap.mflow     int.vel.
1200201  0.0      0.0        0.0         0.0
1200202  0.0      20.0        0.0         0.0
*---------------------------------------------------------------------
*********Definition of the branch 150*********************
```

```
* This branch connects the 110TDV --->> pipe 125 and pipe 126
*----------------------------------------------------------------------------
*---------------------------------------------------------------------
*******branch sngl volume boundary conditions******************
*----------------------------------------------------------------------
1500000  "inlet"  snglvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1500101   0.0028      6.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000*check
*Time dependent data to be pressure
*       ebt(fluid type, no-boron, initial thermodynamic state)
1500200  3     8000000.0   513.0
*---------------------------------------------------------------------
******* single junction from sngl vol to pipe 125 properties*******
*---------------------------------------------------------------------
1270000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1270101  150010002  125010001  0.0028    0.0     0.0      0000000
*     flag  liq.mflow  vap.mflow     int.vel.
1270201  1      0.0         0.0         0.0
*---------------------------------------------------------------------
******* single junction from sngl vol to pipe 126 properties*******
*---------------------------------------------------------------------
1280000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1280101  150010002  126010001  0.0028    0.0     0.0      0000000
*     flag  liq.mflow  vap.mflow     int.vel.
1280201  1      0.0         0.0         0.0
*---------------------------------------------------------------------
*******Pipe Properties (left) *******
```

```
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1250000  "pipe1"  pipe
*     no. of vols
1250001  12
*        flowArea  vol no.
1250101  0.0028    12
*        lengh   vol no.
1250301  0.5    12
*      az. ang vol no
1250501  0.0    12
*      vt. ang vol no
1250601  0.0    12
*       rough      hyd vol no.
1250801  0.0018    0.0  12
*       f loss r loss jun. no.
1250901  0.0    0.0     11
*       vol flag  vol no
1251001  0000000    12
*       jun flag  jun. no
1251101  0000000    11
*      flag  Pressure(P)  temp(K)              vol.no.
1251201 103   8000000.0   513.0 0.0  0.0  0.0  12
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
1251300          1
*     liq.mflow vap.mflow  int.vel.  jun no.
1251301  0.0   0.0         0.0        11
*-------------------------------------------------------------------
*******Pipe (126)Properties (right)*******
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1260000  "pipe1"  pipe
```

```
*     no. of vols
1260001  12
*        flowArea  vol no.
1260101  0.00071    12
*        lengh   vol no.
1260301  0.5    12
*      az. ang vol no
1260501  0.0    12
*      vt. ang vol no
1260601  0.0    12
*       rough      hyd vol no.
1260801  0.0018    0.0  12
*       f loss r loss jun. no.
1260901  0.0    0.0     11
*       vol flag  vol no
1261001  0000000    12
*       jun flag  jun. no
1261101  0000000    11
*      flag  Pressure(P)  temp(K)              vol.no.
1261201 103  8000000.0   513.0  0.0  0.0  0.0  12
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
1261300          1
*     liq.mflow vap.mflow  int.vel.  jun no.
1261301  0.0   0.0         0.0        11
*-------------------------------------------------------------------
******* Branch component (inlet)*******
*-------------------------------------------------------------------
*********Definition of the branch 160*********************
* This branch connects the pipe 125 and pipe 126 --->> 127sngljn
*--------------------------------------------------------------------------
*
1600000  "branPRZ"  branch
```

```
*       no.juns.   ctl
1600001  3         1
*       area      len  volume  h-ang  v-ang  delz   rough  dhy   ctl
1600101  0.0028  0.1  0.0     0      0.0    0.0    0.0    0.0    0
*       ctl  pressure    temperature
1600200  3  8000000.0   513.0
*       from        to          area    floss   rloss    flag
1601101  125120002   160010001    0      0      0       100
1602101  126120002   160010001    0      0      0       100
1603101  160010002   130010001    0      0      0       100
*       flowf   flowg   velj
1601201  0.0     0.0     0
1602201  0.0     0.0     0
1603201  0.0     0.0     0
*
*-------------------------------------------------------------------
******* Exit time dependent volume properties*******
*-------------------------------------------------------------------
1300000  "sink"  tmdpvol
*       f.area    l     vol   azi  vert  dz   rough hyd d flags
1300101  0.0028 6.0   0.0   0.0   0.0  0.0   0.0  0.0  0000000
*Time dependent data to be pressure and temperature
1300200  103
*       time      Pa                    K
1300201  0.0  8000000.0   513.0
*------------------------------------------------------------------
*******Plot Requests*******
*---------------------------------------------------------------
*20300021  mflowj 125010000 1
*20300022  mflowj 125110000 1
*
20300011 p 125010000 1
```

```
20300012 p 125120000 1
20300013 p 126010000 2
20300014 p 126120000 2
*
*20300031  mflowj 126010000 1
*20300032  mflowj 126110000 1
*
20300051  mflowj 125010000 1
20300052  mflowj 125110000 1
20300053  mflowj 126010000 1
20300054  mflowj 126110000 1
*
*20300041  tempf 125010000 1
*20300042  tempf 125120000 1
*20300043  tempf 126010000 1
*20300044  tempf 126120000 1
*
. End
```

III. Flow Rate Determination with Set Pressure Drop

```
= To determine flow rate in a pipe with set Temperature and pressure drop.
*------------------------------------------------------------------
*********** Input Data **********
*-----------------------------------------------------------------
*Mass flow rate 10 kg/s
*Pipe diameter=6cm = 0.06 m
*Pipe Length=30 m
*Inlet temp = 303 K
*quality = 0.00001
*---------------------------------------
```

```
***********Defining the problem*******
*----------------------------------------
100  new  transnt
102  si      si
105   1.0     2.0
110  air
115  1.0
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     50.0   1.0E-6     0.1     5        20         50     2000
*---------------------------------------------------------------------
*******Inlet control volume boundary conditions******************
*---------------------------------------------------------------------
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1100101   0.002827  15.0     0.0       0.0  0.0     0.0   0.0 0.0  0000000
*Time dependent data to be pressure
*       ebt(fluid type, no-boron, initial thermodynamic state)
1100200  103
*       time  Pressure(pa) temp(K)
1100201  0.0   1160000.0   288.0
*---------------------------------------------------------------------
1200000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1200101  110010002  125010001  0.0    0.0     0.0       0000000
*     flag  liq.mflow  vap.mflow     int.vel.
1200201  1      0.0         0.0          0.0
*******Pipe Properties*******
*---------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1250000  "pipe1"  pipe
```

```
*    no. of vols
1250001  6
*       flowArea  vol no.
1250101  0.002827    6
*       lengh   vol no.
1250301  2.5  6
*      az. ang vol no
1250501  0.0    6
*      vt. ang vol no
1250601  0.0    6
*      rough      hyd vol no.
1250801  0.0050    0.0  6
*      f loss r loss jun. no.
1250901  0.0    0.0     5
*      vol flag  vol no
1251001  0000000    6
*      jun flag  jun. no
1251101  0000000    5
*     flag   temp(K)                  vol.no.
1251201 101   288.0  0.0  0.0 0.0   0.0  6
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
1251300          1
*    liq.mflow vap.mflow  int.vel.  jun no.
1251301  0.0   0.0        0.0        5
*-------------------------------------------------------------------
1270000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1270101  125060002  130010001  0.0    0.0     0.0      0000000
*     flag  liq.mflow  vap.mflow     int.vel.
1270201  1      0.0        0.0          0.0
*-------------------------------------------------------------------
```

```
1300000  "sink"  tmdpvol
*       f.area    l     vol   azi  vert  dz   rough hyd d flags
1300101  0.002827 15.0   0.0    0.0    0.0  0.0   0.0  0.0  0000000
*Time dependent data to be pressure and temperature
1300200  103
*       time   pressure  temp(K)
1300201  0.0    1060000.0 288.0
*----------------------------------------------------------------------
*******Plot Requests*******
*------------------------------------------------------------------
20300021  mflowj 125010000 1
20300022  mflowj 125050000 1
*
20300011 p 125010000 1
20300012 p 125060000 1
*
20300031  tempf 125010000 1
20300032  tempf 125060000 1
. end
```

IV. Heat Structure Model

Wall and fluid temperatures of a heated single pipe

```
= Flow in Single heated horizontal pipe
*to investigates the pressure drop, mass flow rate and Fluid/Wall temp difference
*-------------------------------------------------------------------
***********Input Data**********
*------------------------------------------------------------------
*Mass flow rate = 30 kg/s
*Pipe diameter=6cm = 0.06 m
```

```
*Pipe Length=30 m
*Inlet temp = 320 K
*Pipe roughness = 0.0018
*---------------------------------------
***********Defining the problem*******
*----------------------------------------
100  new  transnt
102  si       si
105   1.0     2.0
110  air
115  1.0
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     5000.0   1.0E-6     0.1     5        20         50     2000
*-------------------------------------------------------------------
*******Inlet control volume boundary conditions******************
*--------------------------------------------------------------------
*
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1100101   0.002827  30.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000
*Time dependent data to be pressure
*       ebt(fluid type, no-boron, initial thermodynamic state)
1100200  101
*       time    Temp (K) Quality
1100201  0.0      320.0  0.0
*
*-------------------------------------------------------------------
*******Inlet Time Dependent Junction Properties*******
*-------------------------------------------------------------------
1200000  "inlet"  tmdpjun
*       from vol   to vol    f.area
```

```
1200101  110010002  125010001   0.0
*        vel/flow    trip*
1200200      1         0
*       time  liq.mflow  vap.mflow     int.vel.
1200201  0.0      30.0        0.0         0.0
*
*-------------------------------------------------------------------
*******Pipe Properties*******
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1250000  "pipe1"  pipe
*    no. of vols
1250001  6
*       flowArea  vol no.
1250101  0.002827    6
*       lengh   vol no.
1250301  5.0  6
*     az. ang vol no
1250501  0.0    6
*     vt. ang vol no
1250601  0.0    6
*      rough      hyd vol no.
1250801  0.0018    0.0  6
*      f loss r loss jun. no.
1250901  0.0    0.0     5
*      vol flag  vol no
1251001  0000000    6
*      jun flag  jun. no
1251101  0000000    5
*     flag  temp(K)                         vol.no.
1251201 101  320.0  0.000001   0.0  0.0  0.0  6
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
```

```
1251300         1
*    liq.mflow vap.mflow  int.vel.  jun no.
1251301  0.0   0.0        0.0       5
*
*---------------------------------------------------------------------------
* Definition of the single junction 127
*--------------------------------------------------------------------
1270000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full abruot change
1270101  125060002  130010001  0.0    0.0     0.0      0000000
*      flag  liq.mflow  vap.mflow     int.vel.
1270201  1      0.0         0.0         0.0
*
*---------------------------------------------------------------------------
* Definition of the Time Dependent Volume 130
*--------------------------------------------------------------------
1300000  "sink"  tmdpvol
*       f.area    l     vol   azi  vert  dz   rough hyd d flags
1300101  0.002827 30.0   0.0   0.0   0.0  0.0   0.0  0.0  0000000
*Time dependent data to be pressure and temperature
1300200  101
*       time   Pressure(Pa) Quality
1300201  0.0      320.0  0.0
*
*-------------------------------------------------------------
***********Heat Structure added to 125******
*-------------------------------------------------------------
*        nh  np  geotype  steadyi inner_radius(left)
11250000  6    6    2       0      0.03
*        flag    mesh_format
11250100  0       1
```

```
*        np-1    outer_radius(right)
11250101  5       0.1
*        comp    np-1
11250201  125      5
*        source  np-1
11250301  1.0      5
11250400  0
*        temperature mesh_point
11250401  320.0        6
*left_BC  volume_num  increment  BCtype  SA type  height   hs.num
11250501  125010000    010000    1     1       5.0     6
*Right_BC volume_num  increment  BCtype  SA type  height   hs.num
11250601  0           0          0     1       5.0     6
*Heat    src.type   src.mult  dir.left   dir.rt  hs.num
*        src.type  src.mltp. mod.heat.multp.
11250701  125         10.0      10.0           0  6
*left    hy_dia  Convective heat  transfer factors       hs.num
*       ht.hyd.d. htd.l.fr.        grd          boil.fctr(1.0 no power)
11250801  0.03      10.0    10.0  0.0 0.0  0.0   0.0    0.0001   6
*                      htd l.bk.      grdloss
*--------------------------------------------------------------
*  stainless steel
20112500        tbl/fctn  1  1
*              temperature      thermal conductivity
20112501        273.15          42.5578
20112502        1199.8166667     82.3129
*              temperature      heat capacity
20112551        273.1500        3827850.0
20112552        422.0389        3962160.0
20112553        477.5944        4096471.0
20112554        533.15          4230782.0
20112555        588.7056        4331514.0
```

20112556	644.2611	4432113.0
20112557	699.8167	4499403.0
20112558	810.9278	4633713.0
20112559	1366.483	5372421.0

```
*
20212500 power
20212501 0.0   0.0
20212502 10.0  1000.0
20212503 100.0 10000.0
20212504 500.0  100000.0
*-----------------------------------------------------------------
*******Plot Requests*******
*--------------------------------------------------------------
20300021  mflowj 125010000 1
20300022  mflowj 125050000 1
*
20300011 p 125010000 1
20300012 p 125060000 1
*
20300031  tempf 125010000 1
20300032  httemp 125000101 1
20300033  tempf 125060000 1
20300034  httemp 125000601 1
. End
```

V. Candu Fuel Bundle

```
*To investigates the flow parameter changes in Fuel element Based on the set Operational condition
*---------------------------------------------------------------
*********** Input Data**********
*--------------------------------------------------------------
```

```
*Mass flow rate 20 kg/s
*Pipe diameter=8mm = 0.06 m
*Pipe Length=6 m
*Inlet temp = 513K
*quality = 0.00001
*Pressure = 8MP (reactor operating pressure)
*----------------------------------------------------------------
***********Defining the problem*******
*----------------------------------------------------------------
= Operating condition of fuel element
100  new  transnt
102  si      si
*==========================================================*
*                 cpu time remaining card
*==========================================================*
*crdno  time1  time2
105   1.0    2.0
110   air
115   1.0
*==========================================================*
*                 time step control cards
*==========================================================*
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     100.0   1.0E-9    0.001    5       5000        5000     2000
*----------------------------------------------------------------
*******Inlet control volume boundary conditions******************
*----------------------------------------------------------------
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1100101   0.0043      6.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000*check
*Time dependent data to be pressure
```

```
*       ebt(fluid type, no-boron, initial thermodynamic state)
1100200  103
*       time      Pa                K
1100201  0.0       8000000.0   513.0
*------------------------------------------------------------------
*******Inlet Time Dependent Junction Properties*******
*------------------------------------------------------------------
1200000  "inlet"  tmdpjun
*       from vol   to vol    f.area
1200101  110010002  125010001  0.0043
*        vel/flow    trip*
1200200     1        0
*       time  liq.mflow  vap.mflow     int.vel.
1200201  0.0      0.0        0.0         0.0
1200202  0.0      20.0        0.0         0.0
*------------------------------------------------------------------
*******Pipe Properties*******
*------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1250000  "pipe1"  pipe
*    no. of vols
1250001  12
*       flowArea  vol no.
1250101  0.0043    12
*       lengh   vol no.
1250301  0.5    12
*     az. ang vol no
1250501  0.0    12
*     vt. ang vol no
1250601  0.0    12
*      rough      hyd vol no.
1250801  0.0018    0.0  12
```

```
*      f loss r loss jun. no.
1250901  0.0    0.0     11
*      vol flag  vol no
1251001  0000000    12
*      jun flag  jun. no
1251101  0000000    11
*           flag  Pressure(P)  temp(K)              vol.no.
1251201 103  8000000.0    513.0   0.0  0.0  0.0  12
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
1251300          1
*    liq.mflow vap.mflow  int.vel.  jun no.
1251301  0.0   0.0        0.0        11
*-------------------------------------------------------------------
1270000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1270101  125120002  130010001  0.0043    0.0     0.0       0000000
*     flag  liq.mflow  vap.mflow     int.vel.
1270201  1      0.0         0.0          0.0
*-------------------------------------------------------------------
1300000  "sink"  tmdpvol
*       f.area    l     vol   azi  vert  dz   rough hyd d flags
1300101  0.0043 6.0   0.0   0.0   0.0  0.0   0.0  0.0  0000000
*Time dependent data to be pressure and temperature
1300200  103
*       time      Pa                     K
1300201  0.0      8000000.0  513.0
*-----------------------------------------------------------
***********Heat Structure added to 125******
*-----------------------------------------------------------
*        nhs  np  geotype(Cyl)  steadyi inner_radius(left)
11250000  12   6     2             0         0.0   *Check
```

```
11250100  0  1
*        np-1   outer_radius(right)
11250101  5      0.0065
*        comp    np-1
11250201  55      5
*        src.value
11250301  1.0        5
11250400  0
*                    temp   mesh_pt
11250401  513.0     6
11250501   0          0      0   1  0.5  12
11250601  125010000 010000    1    1  18.5  12
*        src.type  src.mltp.    mod.heat.multp.
11250701  140          0.08333      0.0    0.0  12
*       ht.hyd.d.   htd.l.fr.  grd          boil.fctr(1.0 no power)
11250800  0
11250801  0.0123       10.0      10.0  0.0 0.0  0.0   0.0    1.0    12
11250900  0
11250901  0.0123       10.0      10.0  0.0 0.0  0.0   0.0    1.0    12
*-----------------------------------
****** Material********************
*----------------------------------
20105500 c-steel
*------------------------------------------------
************ Power Table************
*----------------------------------------------------
20214000 power
20214001 0.0  0.0
20214002 10.0 1000000.0
*-----------------------------------------------------------------
*******Plot Requests*******
*--------------------------------------------------------------
```

20300021 mflowj 125010000 1

20300022 mflowj 125110000 1

*

20300011 p 125010000 1

20300012 p 125120000 1

*

20300031 tempf 125010000 1

20300032 httemp 125000101 1

20300033 tempf 125120000 1

20300034 httemp 125001201 1

*

20300041 tempf 125010000 1

20300042 tempf 125120000 1

. End

VI. Pump Model

***To investigates the flow parameter in a PUMP powered flow of 3 Pipe Network**

*---

*********** Input Data**********

*---

*Mass flow rate 20 kg/s

*Pipe diameter=8mm = 0.06 m

*Pipe Length=6 m

*Inlet temp = 513K

*quality = 0.00001

*Pressure = 8MP (reactor operating pressure)

*---------------------------------------

***********Defining the problem*******

*---------------------------------------

= Single Channel Heating

```
100  new  transnt
102  si      si
*=====================================================*
*                  cpu time remaining card
*=====================================================*
*crdno  time1  time2
105   1.0     2.0
110  air
115  1.0
*======================================================*
*                  time step control cards
*=========================================================*
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     100.0   1.0E-9     0.001    5       5000        5000     2000
*-------------------------------------------------------------------
*******Component 130 = TDV 1 Inlet control BC*****************
*--------------------------------------------------------------------
1300000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*        fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1300101   0.0043      6.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000*check
*Time dependent data to be pressure
*        ebt(fluid type, no-boron, initial thermodynamic state)
1300200  103
*        time       Pa                             K
1300201  0.0      2100000.0   300.0
*-------------------------------------------------------------------
*******Component 110 = Inlet Sngljun connecting TDV 1 to Pipe 1****
*-------------------------------------------------------------------
1100000  "sngljuno"  sngljun
*        from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
```

```
1100101  130010002  100010001  0.0043    0.0     0.0       0000000
*      flag  liq.mflow  vap.mflow     int.vel.
1100201  1      0.0         0.0          0.0
*-------------------------------------------------------------------
*******Component 100 = Pipe 1*******
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1000000  "pipe1"  pipe
*     no. of vols
1000001  12
*        flowArea  vol no.
1000101  0.0043    12
*        lengh   vol no.
1000301  0.5    12
*      az. ang vol no
1000501  0.0    12
*      vt. ang vol no
1000601  0.0    12
*       rough      hyd vol no.
1000801  0.0    0.0  12
*       f loss r loss jun. no.
1000901  0.0    0.0     11
*       vol flag  vol no
1001001  0000000    12
*       jun flag  jun. no
1001101  0000000    11
*      flag  Pressure(P)  temp(K)             vol.no.
1001201 103 2010000.0    300.0   0.0  0.0  0.0  12
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
1001300          1
*     liq.mflow vap.mflow  int.vel.  jun no.
1001301  0.0   0.0         0.0        11
```

*--

*********Centrifugal PUMP Properties ****************

*--

*crdno name type

1500000 CentPMP pump

*crdno area length volume h-ang v-ang delz ctl

1500101 0.0043 2.0 0.0 0.0 0.0 0.0 0 * [12]t = 0, Thermal stratification is not used in a pump component.

*crdno from area floss rloss flag

1500108 100010000 0.0043 0.0 0.0 00000 * Pump Inlet(Suction)Junction Card Reduce loss factor to increase flow

*crdno to area floss rloss flag

1500109 200000000 0.0043 0.0 0.0 0000000 * Pump Outlet (Discharge) Junction Card

*crdno ctl pressure temp

1500200 103 10000000.0 300.0* Initial Conditions

*crdno ctl flowf flowg velj

1500201 1 20.0 0.0 0.0 * Pump Inlet (Suction) Junction Initial Conditions

1500202 1 20.0 0.0 0.0 * Pump Outlet (Discharge) Junction Initial Conditions

*crdno id 2faz 2fazd tork pvel ptrip rvrs

1500301 -2 -1 -3 -1 0 0 1 * Pump Index and Option Card

*crdno rpvel initv rflo rhead rtork momi rdens

1500302 186.9 1.0 0.1451 24.6888 234.56 0.001 995.0 * Pump Description Card

*crdno rmotk tf2 tf0 tf1 tf3

1500303 0.0 0.0 0.0 0.0 0.0 * Pump Description Card

* pump rotational velocity table

*crdno trip no- alphanumeric part of variable request code - numeric part of variable request code

1506100 0 time

*crdno srch-var pump-vel

1506101 100.0 186.9

*1326102 5.15 0.5 * VARYING pump speed

```
*1326103   7.35    0.5
*1326104   20.0   0.0
*-------------------------------------------------------------------
*******Component 200 = Pipe 2*******
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
2000000  "pipe1"  pipe
*     no. of vols
2000001  12
*        flowArea  vol no.
2000101  0.0043    12
*        lengh   vol no.
2000301  0.5    12
*      az. ang vol no
2000501  0.0    12
*      vt. ang vol no
2000601  0.0    12
*       rough      hyd vol no.
2000801  0.00    0.0  12
*       f loss r loss jun. no.
2000901  0.0    0.0     11
*       vol flag  vol no
2001001  0000000    12
*       jun flag  jun. no
2001101  0000000    11
*              flag  Pressure(P)  temp(K)              vol.no.
2001201   103  2000000.0   300.0     0.0  0.0  0.0  12
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
2001300          1
*     liq.mflow vap.mflow  int.vel.  jun no.
2001301 0.0   0.0         0.0        11
*-------------------------------------------------------------------
```

```
*******Component 120****
*---------------------------------------------------------------------
1200000  "sngljuno"  sngljun
*       from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1200101  200120002  300010001  0.0043    0.0     0.0       0000000
*      flag  liq.mflow  vap.mflow     int.vel.
1200201  1      0.0         0.0          0.0
*---------------------------------------------------------------------
*******Component 300= Pipe 3*******
*---------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
3000000  "pipe1"  pipe
*     no. of vols
3000001  12
*       flowArea  vol no.
3000101  0.0043    12
*       lengh   vol no.
3000301  0.5    12
*      az. ang vol no
3000501  0.0    12
*      vt. ang vol no
3000601 90.0    12
*       rough      hyd vol no.
3000801  0.0    0.0  12
*       f loss r loss jun. no.
3000901  0.0    0.0     11
*       vol flag  vol no
3001001  0000000    12
*       jun flag  jun. no
3001101  0000000    11
*      flag  Pressure(P)  temp(K)              vol.no.
```

```
3001201 103 2000000.0    300.0   0.0  0.0  0.0  12
*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)
3001300          1
*     liq.mflow vap.mflow  int.vel.  jun no.
3001301  0.0   0.0         0.0        11
*---------------------------------------------------------------------
*******Component 160 = Outlet Sngljun connecting TDV 2 to Pipe 2****
*---------------------------------------------------------------------
1600000  "sngljuno"  sngljun
*        from vol   to vol    f.area f.loss   r.loss     flag (fefvcahs)a=0 smooth, a=1 full
abruot change
1600101  300120002  140010001  0.0043    0.0     0.0       0000000
*      flag  liq.mflow  vap.mflow     int.vel.
1600201  1      0.0         0.0          0.0
*---------------------------------------------------------------------
*******Component 140 = TDV 2 Outlet control BC*********************
*---------------------------------------------------------------------
1400000  "sink"  tmdpvol
*        f.area    l     vol   azi  vert  dz   rough hyd d flags
1400101  0.0043 6.0   0.0   0.0   0.0  0.0   0.0  0.0  0000000
*Time dependent data to be pressure and temperature
1400200  103
*        time      Pa                       K
1400201  0.0   2000000.0  300.0
*---------------------------------------------------------------------
*******Plot Requests*******
*------------------------------------------------------------------
20300021  mflowj 100010000 1
20300022  mflowj 100110000 1
20300023  mflowj 200010000 1
20300024  mflowj 200110000 1
*
```

```
20300011 p 100010000 1
20300012 p 100120000 1
20300013 p 200010000 1
20300014 p 200120000 1
*
20300031  tempf 100010000 1
20300032  tempf 100120000 1
20300033  tempf 200010000 1
20300034  tempf 200120000 1
*
20300041 p 300010000 1
20300042 p 300120000 1
*
20300051  mflowj 300010000 1
20300052  mflowj 300110000 1
*
. End
```

VII. 4 Pipe Loop Model

```
=Four Pipe Simple Loop with a Pump and Heater Input Model
*In this model uniform heat sources are also applied onto the pipes 1 and 3
*100          new    stdy-st
100           new    transnt
105     5.0    6.0   500.0
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201    1000.00  1.0-6    0.1       3        10       100    5000
*==============================================
* plot requests
*==================================================
20300011  mflowj  100010000 1
```

```
20300012  mflowj  150010000 1
20300013  mflowj  140010000 1
20300014  mflowj  160010000 1
*
20300031  httemp  100000201 1
20300032  httemp  150000201 1
*
20300021       p  100020000 1
20300022       p  150020000 1
20300023       p  140020000 1
20300024       p  160020000 1
*
20300041   tempf  100020000 1
20300042   tempf  150020000 1
20300043   tempf  140020000 1
20300044   tempf  160020000 1
*=================================================
*                    trips
*=================================================
*501   time      0    gt   null   0     501.0  1 * for pump + logical
*502   tempf 100020000 gt null 0 365.0 l -1.*for accumulator trip when temp at 100
exceeds 510K
*-------------------------------------------------------------------
*                    component 160
*================================================
*crdno      name       type
1600000    pipe_no4       pipe
*crdno     no.vols.
1600001        2
*crdno     area                        vol.no.
1600101    0.0196                         2
*crdno     length                      vol.no.
```

```
1600301       2.0                          2
*crdno     h-ang                        vol.no.
1600501      0.0                           2
*crdno     v-ang                        vol.no.
1600601      0.0                           2
*crdno     rough     dhy                  vol.no.
1600801        0       0                   2
*crdno     ctl                          vol.no.
1601001      0                             2
*crdno     ctl                          jun.no.
1601101      0                             1
*6rdno     ctl       pressure     temp      vol.no.
1601201      3          1.5+6    350.0  0 0 0    2
*crdno     i.c.
1601300      1
*crdno     flowf     flowg       velj      jun.no.
1601301    0.0      0.0           0          1
*------------------------------------------------------------------
*                    component 170
*=========================================
* pump
*     westinghouse homologous pump curves
*     flow area = 0.07 sqm, volume = 0.14 cum
*     ratings:  speed = 369 rpm               flow = 0.81 cum/sec
*               head = 27.5 m              torque = 500.0 nm
*             mom.i. = 1.43 kg*sqm          density = 614.0 kg/cum
*     actual speed = 369 rpm
*crdno       name        type
1700000      pump        pump
*crdno     area   length volume h-ang   v-ang   delz   ctl
1700101    0.07      2.0      0     0      0.0     0.0     0
*crdno     from          area     floss     rloss     flag
```

```
1700108   160020002      0      0      0     0
*crdno    to         area    floss    rloss    flag
1700109   100010001      0      0      0     0
*crdno    ctl   pressure    temp
1700200     3      1.5+6    350.0
*crdno    ctl    flowf   flowg   velj
1700201     1    0.0    0.0      0
1700202     1    0.0    0.0      0
*crdno     id   2faz   2fazd   tork   pvel   ptrip   rvrs
1700301    -2     -1     -3     -1     0     0      1
*crdno    rpvel  initv  rflo   rhead   rtork   momi   rdens
1700302   369.0   1.0   0.81    27.5   500.0   1.43   614.0
*crdno    rmotk   tf2   tf0    tf1    tf3
1700303      0  153.0  0.003  14.5      0

* pump rotational velocity table
1706100  0 time
1706101  100.0 0.0 150.0 100.0
*==============================================
*                  component 100
*================================================
*crdno      name       type
1000000    pipe_no1      pipe
*crdno    no.vols.
1000001        2
*crdno    area                       vol.no.
1000101    0.0196                         2
*crdno    length                      vol.no.
1000301      2.0                        2
*crdno    h-ang                       vol.no.
1000501     0.0                         2
*crdno    v-ang                       vol.no.
```

```
1000601    90.0                     2
*crdno    rough    dhy              vol.no.
1000801      0      0               2
*crdno    ctl                       vol.no.
1001001     0                       2
*crdno    ctl                       jun.no.
1001101     0                       1
*crdno    ctl      pressure    temp     vol.no.
1001201     3        1.5+6    350.0  0 0 0    2
*crdno    i.c.
1001300     1
*crdno    flowf    flowg      velj     jun.no.
1001301    0.0     0.0         0         1
*================================================
**********Heat Structure added to Pipe 100***************
*=====================================================
*       NVOL NP Geo Stdy In.Rad.
11000000  2   5  2   0  0.025
*                   M.Loc Form
11000100  0    1
11000101  4  0.05
11000201  100  4
*    Power curve
11000301  1.0    4*flat power profile assumed
*     in.temp. NP
11000401  350.0 5
*          left    incr.  typ area    NH
11000501  100010000 010000  1  1  2.0  2
*          right    incr.  typ area      NH
11000601       0      0       0   1   2.0  2
*               power peaking factor
*         src.table  src.mltp.   mod.multp. NH
```

```
11000701  100        1.0        1.0    0.0  2
*       ht.hyd.d. heated length  grid          boil.fctr NH
11000801   0.0      10.0 10.0  0.0 0.0  0.0   0.0    1.0   2

*  stainless steel
20110000  tbl/fctn  1  1  *  stainless steel
*
*          temperature   thermal conductivity
20110001       273.15           42.5578
20110002       1199.8166667      82.3129
*
*          temperature   heat capacity
20110051       273.1500        3827850.0
20110052       422.0389        3962160.0
20110053       477.5944        4096471.0
20110054       533.15          4230782.0
20110055       588.7056        4331514.0
20110056       644.2611        4432113.0
20110057       699.8167        4499403.0
20110058       810.9278        4633713.0
20110059       1366.483        5372421.0
20210000 power
20210001  0.0  0.0
20210002  0.0  20000.0
*=====================================================
*                   component 120
*=====================================================
*crdno       name        type
1200000      bndbrnch      branch
*crdno   no.juns.    ctl
1200001        3       1
*crdno   area   length volume h-ang  v-ang    delz    rough    dhy   ctl
```

```
1200101  0.0196  2.0    0.0 180.0    0.0     0.0      0.0    0.0     0
*crdno   ctl    pressure     temp
1200200    3       1.5+6    350.0
*crdno   from         to          area     floss     rloss     flag
1201101  100020002    120010001       0        0        0      100
1202101  120010002    140010001       0        0        0      100
1203101  120010006    130010001       0        0        0      100
*crdno   flowf    flowg    velj
1201201  0.0     0.0       0
1202201  0.0     0.0       0
1203201  0.0     0.0       0
*=======================================================
*                    component 140
*==================================================
*crdno       name        type
1400000     pipe_no2       pipe
*crdno     no.vols.
1400001          2
*crdno     area                        vol.no.
1400101    0.0196                          2
*crdno     length                       vol.no.
1400301       2.0                         2
*crdno     h-ang                        vol.no.
1400501    140.0                          2
*crdno     v-ang                        vol.no.
1400601     0.0                          2
*crdno     rough     dhy                  vol.no.
1400801        0       0                   2
*crdno     ctl                         vol.no.
1401001      0                            2
*crdno     ctl                         jun.no.
1401101      0                            1
```

```
*crdno    ctl      pressure    temp     vol.no.
1401201     3         1.5+6    350.0  0 0 0    2
*crdno    i.c.
1401300     1
*crdno    flowf    flowg       velj     jun.no.
1401301    0.0      0.0          0          1
*===============================================
*                  component 200
*===============================================
*crdno       name        type
2000000     jun2_3     sngljun
*crdno    from         to          area     floss     rloss     flag
2000101   140020002    150010001       0        0        0      000
*crdno    ctl     flowf    flowg    velj
2000201     1     0.0     0.0       0
*=============================================
*                  component 150
*===============================================
*crdno       name        type
1500000    pipe_no3       pipe
*crdno    no.vols.
1500001        2
*crdno    area                        vol.no.
1500101    0.0196                        2
*crdno    length                      vol.no.
1500301      2.0                        2
*crdno    h-ang                       vol.no.
1500501      0.0                        2
*crdno    v-ang                       vol.no.
1500601    -90.0                        2
*crdno    rough    dhy                vol.no.
1500801       0      0                  2
```

```
*crdno    ctl                    vol.no.
1501001    0                        2
*crdno    ctl                    jun.no.
1501101    0                        1
*crdno    ctl     pressure    temp     vol.no.
1501201    3        1.5+6    350.0  0 0 0    2
*crdno    i.c.
1501300    1
*crdno    flowf    flowg      velj     jun.no.
1501301   0.0     0.0         0         1
*-------------------------------------------------------------------
*Pipe 150 wall added by heat structure
*============================================
*       NH NP Geo Stdy In.Rad.
11500000  2  5  2   0  0.025
*                 M.Loc Form
11500100  0    1
11500101  4  0.05
11500201  150  4
*    Power curve shape
11500301  1.0    4*flat power profile assumed
*     wall in.temp.(K)
11500401  350.0 5
*         left    incr. typ area  NH
11500501  150010000 010000  1   1  2.0  2
*         right    incr.  typ area      NH
11500601      0      0       0   1   2.0  2
*             power peaking factor
*        src.table  src.mltp.   mod.multp. NH
11500701  150         1.0        1.0    0.0  2
*       ht.hyd.d. heated length  grid         boil.fctr NH
11500801   0.0      10.0 10.0  0.0 0.0  0.0   0.0    1.0   2
```

```
*  stainless steel
20115000  tbl/fctn  1  1  *  stainless steel
*
*          temperature   thermal conductivity
20115001       273.15          42.5578
20115002       1199.8166667     82.3129
*
*          temperature   heat capacity
20115051       273.1500        3827850.0
20115052       422.0389        3962160.0
20115053       477.5944        4096471.0
20115054       533.15          4230782.0
20115055       588.7056        4331514.0
20115056       644.2611        4432113.0
20115057       699.8167        4499403.0
20115058       810.9278        4633713.0
20115059       1366.483        5372421.0

20215000 power
20215001   0.0    0.0
20215002   0.0    0.0
*------------------------------------------------------------------
*                  component 210
*==============================================
*crdno      name        type
2100000     junct3_4     sngljun
*crdno    from         to          area     floss     rloss     flag
2100101   150020002    160010001      0        0         0      000
*crdno    ctl    flowf    flowg    velj
2100201     1     0.0     0.0       0
*------------------------------------------------------------------
```

```
*                component 130
*==================================================
*crdno     name      type
1300000   boundry      pipe
*crdno   no.vols.
1300001       4
*crdno   area                     vol.no.
1300101   1.45-3                      1
1300102    0.6                       4
*crdno   length                    vol.no.
1300301     8.0                      1
1300302     1.5                      4
*crdno   v-ang                     vol.no.
1300601    90.0                      4
*crdno   rough   dhy               vol.no.
1300801      0     0                 4
*crdno   ctl                      vol.no.
1301001     0                        4
*crdno   ctl                      jun.no.
1301101     0                        3
*crdno   ctl     pressure   quals     vol.no.
1301201     2       1.5+5     0.0  0 0 0    4
*crdno   i.c.
1301300     1
*crdno   flowf   flowg     velj     jun.no.
1301301     0       0         0        3
*-------------------------------------------------------------------
*                component 131
*=======================================================
*crdno     name      type
1310000   boundry     sngljun
*crdno   from       to        area    floss    rloss    flag
```

```
1310101   130010000    132000000     0      0      0      0
*crdno    ctl    flowf    flowg    velj
1310201    1       0       0       0
*-------------------------------------------------------------------
*                  component 132
*===============================================
*crdno       name        type
1320000     boundry       tmdpvol
*crdno   area   length volume h-ang  v-ang    delz    rough    dhy   ctl
1320101   0.6     3.0     0    0     0      0       0     0    0
*crdno   ctl
1320200    2
*crdno   time     pressure     quals
1320201   0.0       1.5+6      0.0
*-------------------------------------------------------------------
. End of input.
```

VIII. Heat Exchanger Model

```
*Determination of Heat transfer parameters in a HX
*-----------------------------------------------------------------
***********Input Data**********
*-----------------------------------------------------------------
*Pipe diameter=6cm = 0.06m
*Pipe Length=6m
*Inlet temp = 513K (hot leg)
*Inlet temp = 483K (cold leg)
*quality = 0.00001
*Mass flow rate = 20 kg/s (hot leg)
*Mass flow rate = 40 kg/s (cold leg)
*Pressure = 10MP (reactor operating pressure)
```

```
*-------------------------------------------------
***********Defining the problem*******
*--------------------------------------------------
= Single Channel Heating
100  new  transnt
102  si      si
*=====================================================*
*************cpu time remaining card*************
*=====================================================*
*crdno  time1  time2
105   1.0     2.0
110  air
115  1.0
*====================================================*
*                 time step control cards
*====================================================*
*crdno end time  min dt   max dt  control  minor ed  major ed  restart
201     100.0   1.0E-9    0.01    5        5000        5000     2000
*-------------------------------------------------------------
*******Inlet control TDV BC for the HOT side of HX*********
*-------------------------------------------------------------
1100000  "inlet"  tmdpvol
*flow area of inlet volume is selected the same as the flow area of pipe
*       fa(sq.m)  length(m)  vol(m3)  azi  vert    dz   rough hyd.d flags
1100101   0.002827      2.0     0.0       0.0  0.0     0.0   0.00 0.0  0000000*check
*Time dependent data to be pressure
*       ebt(fluid type, no-boron, initial thermodynamic state)
1100200  103
*       time       Pa                          K
1100201  0.0        10000000.0   513.0
*-------------------------------------------------------------------
*******Inlet Time Dependent Junction Properties-
```

```
*-------------------------------------------------------------------
1200000  "inlet"  tmdpjun
*       from vol   to vol    f.area
1200101  110010002  100010001  0.002827
*        vel/flow    trip*
1200200      1         0
*       time  liq.mflow  vap.mflow     int.vel.
1200201  0.0     20.0         0.0          0.0
*-------------------------------------------------------------------
*******Condenser HOT side component 100*******
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
1000000  "pipe1"  pipe
*     no. of vols
1000001  12
*       flowArea  vol no.
1000101  0.002827    12
*       lengh   vol no.
1000301  0.5    12
*     az. ang vol no
1000501  0.0    12
*     vt. ang vol no
1000601  0.0    12
*      rough      hyd vol no.
1000801  0.00    0.0  12
*      f loss r loss jun. no.
1000901  0.0    0.0     11
*      vol flag  vol no
1001001  0000000    12
*      jun flag  jun. no
1001101  0000000    11
*     flag  Pressure(P)  temp(K)              vol.no.
```

1001201 103 10000000.0 513.0 0.0 0.0 0.0 12

*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)

1001300 1

* liq.mflow vap.mflow int.vel. jun no.

1001301 0.0 0.0 0.0 11

*--

*******Outlet Single Junction 130*******-HOT-outlet

*--

1300000 "sngljuno" sngljun

* from vol to vol f.area f.loss r.loss flag (fefvcahs)a=0 smooth, a=1 full abruot change

1300101 100120002 190010001 0.002827 0.0 0.0 0000000

* flag liq.mflow vap.mflow int.vel.

1300201 1 0.0 0.0 0.0

*--

*******Outlet TDV 190 **************HOT-outlet

*--

1900000 "OUTLET" tmdpvol

* f.area l vol azi vert dz rough hyd d flags

1900101 0.002827 6.0 0.0 0.0 0.0 0.0 0.0 0.0 0000000

*Time dependent data to be pressure and temperature

1900200 103

* time Pa K

1900201 0.0 10000000.0 513.0

*--

*******150 TDV-COLD-inlet BC*********

*--

1500000 "Source" tmdpvol

*flow area of inlet volume is selected the same as the flow area of pipe

* fa(sq.m) length(m) vol(m3) azi vert dz rough hyd.d flags

1500101 0.002827 6.0 0.0 0.0 0.0 0.0 0.00 0.0 0000000*check

*Time dependent data to be pressure

```
*       ebt(fluid type, no-boron, initial thermodynamic state)
1500200  103
*       time      Pa                              K
1500201  0.0       10000000.0   483.0
*-------------------------------------------------------------------
*******160 TDJ-inlet BC*******************TDJ-COLD-inlet
*-------------------------------------------------------------------
1600000  "inlet"  tmdpjun
*       from vol   to vol    f.area
1600101   180010001 300120002    0.002827
*        vel/flow    trip*
1600200      1         0
*       time  liq.mflow  vap.mflow     int.vel.
1600201  0.0      20.0        0.0         0.0
*-------------------------------------------------------------------
*******300 Pipe Properties*******COLD PIPE
*-------------------------------------------------------------------
*6cm diameter 30 meter  stainless steel pipe
3000000  "pipe1"  pipe
*    no. of vols
3000001  12
*       flowArea  vol no.
3000101  0.002827    12
*       lengh   vol no.
3000301  0.5    12
*     az. ang vol no
3000501  0.0    12
*     vt. ang vol no
3000601  0.0    12
*      rough      hyd vol no.
3000801  0.0000    0.0  12
*      f loss r loss jun. no.
```

3000901 0.0 0.0 11

* vol flag vol no

3001001 0000000 12

* jun flag jun. no

3001101 0000000 11

* flag Pressure(P) temp(K) vol.no.

3001201 103 10000000.0 483.0 0.0 0.0 0.0 12

*pipe junc.cont.word(mflow will be entered as inital values at pipe junctions)

3001300 1

* liq.mflow vap.mflow int.vel. jun no.

3001301 0.0 0.0 0.0 11

*---

1700000 "sngljuno" sngljun

* from vol to vol f.area f.loss r.loss flag (fefvcahs)a=0 smooth, a=1 full abruot change

1700101 300010001 150010002 0.002827 0.0 0.0 0000000

* flag liq.mflow vap.mflow int.vel.

1700201 1 0.0 0.0 0.0

*---

1800000 "sink" tmdpvol

* f.area l vol azi vert dz rough hyd d flags

1800101 0.002827 6.0 0.0 0.0 0.0 0.0 0.0 0.0 0000000

*Time dependent data to be pressure and temperature

1800200 103

* time Pa K

1800201 0.0 10000000.0 483.0

*---

*Condenser Heat component 200 (31 tubes, 0.016 id, wall = 0.001m thick, 1.7m long):

*---

Condenser Heat Structure component 100 HX to 100 and 300*************

*---

* nhs np geotype(Cyl) steadyi inner_radius(left)

11000000 12 6 2 0 0.025 *Check

* flag mesh format

11000100 0 1

* np-1 outer_radius(right)

11000101 5 0.03

* comp np-1

11000201 55 5

* source np-1

11000301 1.0 5

* temp mesh point

11000400 0

* temp mesh_pt

11000401 513.0 6

*left BC volume_num increment BCtype SA type height hs.num(Hot side)

11000501 100010000 010000 1 1 6.0 12

*left BC volume_num increment BCtype SA type height hs.num(cold side)

11000601 300120000 -010000 1 1 6.0 12

* src.type src.mltp. mod.heat.multp.

11000701 140 0.08333 0.0 0.0 12 * src.mltp 0.08333=1/12 where 12 is number of vol.

* ht.hyd.d. htd.l.fr. grd boil.fctr(1.0 no power)

11000800 0

11000801 0.0123 10.0 10.0 0.0 0.0 0.0 0.0 1.0 12 *Check

11000900 0

11000901 0.0123 10.0 10.0 0.0 0.0 0.0 0.0 1.0 12

*-----------------------------------

****** Material********************

*----------------------------------

20105500 c-steel

*

20214000 power

20214001 0.0 0.0

```
20214002 10.0 1.0
*-----------------------------------------------------------------
*******Plot Requests*******
*------------------------------------------------------------------
20300021  mflowj 100010000 1
20300022  mflowj 100110000 1
20300023  mflowj 300010000 1
20300024  mflowj 300110000 1
*
20300011 p 100010000 1
20300012 p 100120000 1
20300013 p 300010000 1
20300014 p 300120000 1
*
20300041  tempf 100120000 1
20300042  tempf 100010000 1
20300043  tempf 300010000 1
20300044  tempf 300120000 1
*
*20300031  httemp 100000101 1
*20300032  httemp 100001201 1
*20300033  httemp 300000101 1
*20300034  httemp 300001201 1
. End
```

Bibliography

[1] United Nations Framework Convention on Climate Change, "COP28 Declaration," 2024. [Online]. Available: https://www.un.org/press/en/2024/cop28-declaration

[2] United Nations Framework Convention on Climate Change, "The Paris Agreement," 2015. [Online]. Available: https://unfccc.int/process-and-meetings/the-paris-agreement/the-paris-agreement.

[3] World Nuclear Association, "The Nuclear Fuel Report: Global Scenarios for Demand and Supply Availability 2017-2040," 2017. [Online]. Available: https://www.world-nuclear.org/our-association/publications/online-reports/the-nuclear-fuel-report.aspx

[4] World Nuclear Association, "The Nuclear Fuel Report: Global Scenarios for Demand and Supply Availability 2017-2040," 2017. [Online]. Available: https://www.world-nuclear.org/our-association/publications/online-reports/the-nuclear-fuel-report.aspx

[5] M. Oe, Y. Takebayashi, H. Sato, & M. Maeda, "Mental health consequences of the three mile island, chernobyl, and fukushima nuclear disasters: a scoping review", International Journal of Environmental Research and Public Health, vol. 18, no. 14, p. 7478, 2021. https://doi.org/10.3390/ijerph18147478

[6] A. Satô and Y. Lyamzina, "Diversity of concerns in recovery after a nuclear accident: a perspective from fukushima", International Journal of

Environmental Research and Public Health, vol. 15, no. 2, p. 350, 2018. https://doi.org/10.3390/ijerph15020350

[7] J. Song and T. Kim, "Severe accident issues raised by the fukushima accident and improvements suggested", Nuclear Engineering and Technology, vol. 46, no. 2, p. 207-216, 2014. https://doi.org/10.5516/net.03.2013.079

[8] International Atomic Energy Agency, "Safety Standards Series No. SSR-2/1 (Rev. 1)," 2018. [Online]. Available: https://www.iaea.org/publications/8937/safety-of-nuclear-power-plants-design

[9] US Nuclear Regulatory Commission, "Regulatory Guide 1.70: Standard Format and Content of Safety Analysis Reports for Nuclear Power Plants," 2016. [Online]. Available: https://www.nrc.gov/reading-rm/doc-collections/reg-guides/power-reactors/rg/01-070/

[10] N. Aksan, F. D'Auria, and H. Glaeser, "Thermal-hydraulic phenomena for water cooled nuclear reactors," Scientific Consultant, Haldenstrasse 35, CH-5415 Nussbaumen, Switzerland; DESTEC/GRNSPG, University of Pisa, Via Livornese 1291, San Piero a Grado, 56100 Pisa, Italy; Consultant, Herbststrasse 9, 85386 Eching, Germany.

[11] M. M. El-Wakil, *Nuclear heat transport*. American Nuclear Society, 1983.

[12] U.S. Nuclear Regulatory Commission, "RELAP5/MOD3.3 Code Manual," Idaho National Laboratory, 2001.

[13] N. E. Todreas and M. S. Kazimi, *Nuclear systems volume I: Thermal hydraulic fundamentals*. CRC Press, 2011.

[14] U.S. Nuclear Regulatory Commission, "RELAP5/MOD3.3 Code Manual," Idaho National Laboratory, 2001.

[15] International Atomic Energy Agency, "Safety Standards Series No. SSR-2/1 (Rev. 1)," 2018. [Online]. Available: https://www.iaea.org/publications/8937/safety-of-nuclear-power-plants-design

[16] U.S. Nuclear Regulatory Commission, "RELAP5/MOD3.3 Code Manual," Idaho National Laboratory, 2001.

[17] Innovative Systems Software, "RELAP5-3D Code Manual," ISS, Idaho Falls, 2015.

[18] M. Kim, H.-K. Kim, H.-J. Kim, S. H. Hwang, I. S. Hong, and C. H. Kim, "Enhancement of safety analysis capability for a CANDU-6 reactor using RELAP-CANDU/SCAN coupled code system," Nuclear Technology, vol. 156, no. 2, pp. 159-167, 2006. doi: 10.13182/NT06-A3782.

[19] I. Prisecaru, D. Dupleac, and L. Biro, "A study of RELAP5 capability to perform the analysis of CANDU6 reactors accidents," in Proc. 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, Oct. 2-6, 2005, Paper 298.

[20] B. D. Chung, W. J. Lee, and Y. S. Bang, "Improvements of RELAP5/MOD3.2.2 Models for the CANDU Plant Analysis," *U.S. Nuclear Regulatory Commission*, NUREG/IA-0189, Oct. 2000.

[21] D. Dupleac, A. Prisecaru, and L. Biro, "A study of RELAP5 capability to perform the analysis of CANDU6 reactors accidents," *Nuclear Engineering and Design*, vol. 239, no. 9, pp. 1773-1782, Sept. 2009. doi: 10.1016/j.nucengdes.2009.05.007.

[22] A. K. Trivedi, D. R. Novog, and C. Allison, "Implementation of Solar Salt as Fluid in ASYST4.1 and Validation for a Natural Circulation Loop," in Proc.

28th Int. Conf. on Nuclear Engineering (ICONE-28), Avignon, France, Oct. 2-6, 2005, Paper 298.

[23] A. Ezzidi, M. Naitoh, and C. M. Allison, "Need of a next generation severe accident code," Jurnal Teknologi Reaktor Nuklir TRI DASA MEGA, vol. 21, no. 3, p. 119, Nov. 2019. doi: 10.17146/tdm.2019.21.3.5630.

[24] D. Dupleac, R.-M. Nistor-Vlad, C. M. Allison, and M. Perez Ferragut, "Development of the ASYST VER 3.x LWR/HPWR Best Estimate Integral Code," in Proc. 2021 10th Int. Conf. on Energy and Environment (CIEM), Oct. 2021. doi: 10.1109/CIEM52821.2021.9614801.

[25] C. M. Allison, "Introduction to AFSAS, ADTP, and ASYST (Adaptive SYStem Thermal-hydraulics) Development of best estimate methods for reactor and advanced fluid systems safety analysis and training," Dec. 2019. doi: 10.13140/RG.2.2.30036.35204.

[26] C. M. Allison, J. K. Hohorst, A. Ezzidi, and R. Pericas, "Development and preliminary assessment of the new ASYST - ISA Integral Analysis BEPU Code using the PBF SFD-ST Bundle Heating and Melting Experiment, a typical BWR under Fukushima-Daiichi-accident-like thermal hydraulic conditions and PWR for a steam line break in the containment," in Proc. ICAPP 2020, Abu Dhabi, UAE, Mar. 15-19, 2020.

[27] N. Vododokhov, A. Trivedi, and D. R. Novog, "ASYST4.1 validation for gas cooled SMR applications using the HTTF experiment," Nuclear Engineering and Design, vol. [volume number], [page numbers], 2023. doi: 10.1016/j.nucengdes.2023.112643.

[28] G. L. Mesina, "A History of RELAP Computer Codes," Idaho National Laboratory, 2525 Fremont Avenue, Idaho Falls, Idaho 83402.

[29] A. D. Shin, C. W. Huh, I. G. Kim, and S. M. Bajorek, "Assessment of TRACE 4.050 using UPTF bypass tests," Transactions of the American Nuclear Society, vol. 93, pp. 538-538, 2005.

[30] D. Bestion, "System code models and capabilities," 2008.

[31] S. G. Shin and J. I. Lee, "Comparative study of constitutive relations implemented in RELAP5 and TRACE – Part I: Methodology & wall friction," Nuclear Engineering and Technology, vol. 54, no. 9, pp. 3526-3539, 2022. Available: https://doi.org/10.1016/j.net.2022.03.036.

[32] R. Szilard, H. Zhang, D. B. Kothe, and P. J. Turinsky, "The Consortium for Advanced Simulation of Light Water Reactors," Jan. 2011.

[33] Institute of Thermal Energy Technology and Safety, "Research Projects," Karlsruhe Institute of Technology, [Online]. Available: https://www.ites.kit.edu/671.php. [Accessed: 13-Jul-2024].

[33] D. Anders, R. Berry, D. Gaston, R. Martineau, J. Peterson, H. Zhang, H. Zhao, and L. Zou, "RELAP-7 Level 2 Milestone Report: Demonstration of a Steady State Single Phase PWR Simulation with RELAP-7," Idaho National Laboratory, May 2012.

[34] A. Hainoun, "Towards Standard Methodology in the Safety Analysis of Research Reactors," presented at the International Conference on Research Reactors: Safe Management and Effective Utilization, Rabat, Morocco, Nov. 14-18, 2011.

[35] K. Ardron, "System Codes for Reactor Licensing – Part 1: Code Applications," presented at the Imperial College – Nuclear Thermalhydraulics Course, Feb. 2014.

[36] INTERNATIONAL ATOMIC ENERGY AGENCY, "Milestones in the Development of a National Infrastructure for Nuclear Power," IAEA Nuclear Energy Series No. NG-G-3.1 (Rev. 1), Vienna, 2015.

[37] U.S. Nuclear Regulatory Commission, "Safety Codes," [Online]. Available: https://www.nrc.gov/about-nrc/regulatory/research/safetycodes.html. [Accessed: 13-Jul-2024].

[38] K. Andrzejewski, T. Kulikowska, M. M. Bretscher, N. A. Hanan, and J. E. Matos, "Methods and Codes for Neutronic Calculations of the MARIA Research Reactor," Institute of Atomic Energy, Poland, and Argonne National Laboratory, USA.

[39] Institute of Thermal Energy Technology and Safety, "Research Projects," Karlsruhe Institute of Technology, [Online]. Available: https://www.ites.kit.edu/671.php. [Accessed: 13-Jul-2024].

[40] Argonne National Laboratory, Mathematics and Computer Science Division, "Multiphysics Simulations: Challenges and Opportunities," ANL/MCS-TM-321 Rev. 1.1, Rev. 1.1.

[41] T. Zhang, "Chapter 5 - Nuclear physics deterministic code," in Nuclear Power Plant Design and Analysis Codes, J. Wang, X. Li, C. Allison, and J. Hohorst, Eds., Woodhead Publishing Series in Energy, Woodhead Publishing, 2021, pp. 97-122. doi: 10.1016/B978-0-12-818190-4.00005-X.

[42] T. Xenofontos, G.-K. Delipei, P. Savva, M. Varvayanni, J. Maillard, J. Silva, and N. Catsaros, "Testing the new stochastic neutronic code ANET in simulating safety important parameters," Annals of Nuclear Energy, vol. 103, pp. 85-96, 2017. doi: 10.1016/j.anucene.2017.01.012.

[43] International Atomic Energy Agency, Benchmarking Severe Accident Computer Codes for Heavy Water Reactor Applications, IAEA-TECDOC-1727, IAEA, Vienna, 2013.

[44] Idaho National Laboratory, "Code Descriptions," [Online]. Available: https://neams.inl.gov/code-descriptions/. [Accessed: 13-Jul-2024].

[45] C. Pereira, R. Schirru, and A. S. Martinez, "Genetic Algorithms Applied to Nuclear Reactor Design Optimization," in Fuzzy Systems and Soft Computing in Nuclear Engineering, Jan. 2000, pp. 217-237. doi: 10.1007/978-3-7908-1866-6_14.

[46] G. Yesilyurt, K. T. Clarno, I. C. Gauld, and J. Galloway, "MODULAR ORIGEN-S FOR MULTI-PHYSICS CODE SYSTEMS," presented at the International Conference on Mathematics and Computational Methods Applied to Nuclear Science and Engineering (M&C 2011), Rio de Janeiro, RJ, Brazil, May 8-12, 2011, on CD-ROM, Latin American Section (LAS) / American Nuclear Society (ANS), ISBN 978-85-63688-00-2.

[47] Z. XingGuan, J. Cai, and H. C. Chow, "Physics Codes and Methods for CANDU Reactor," Shanghai Nuclear Engineering Research & Design Institute, Atomic Energy of Canada Limited.

[48] S. Davies, D. Litskevich, U. Rohde, A. Detkina, B. Merk, P. Bryceet al.,

"Dyn3d and ctf coupling within a multiscale and multiphysics software development (part i)", Energies, vol. 14, no. 16, p. 5060, 2021.

[49] A. Bikeev, "Precision calculations of some neutron-physical characteristics of vver-1000 using the mcu code", Atomic Energy, vol. 118, no. 4, p. 234-237, 2015. https://doi.org/10.1007/s10512-015-9985-6

[50] K. Chen, C. Tang, Y. Qian, L. Qi, J. Zhao, & Y. Lei, "Safety analysis of representative accidental transients for a 100 mwe small modular lbe-cooled reactor", Frontiers in Energy Research, vol. 10, 2023. https://doi.org/10.3389/fenrg.2022.1088706

[51] M. Petri and W. Pasedag, "U.s. department of energy sponsored in-depth safety assessments of vver and rbmk reactors", 10th International Conference on Nuclear Engineering, Volume 2, 2002. https://doi.org/10.1115/icone10-22586

[52] H. Chen, C. Li, H. Xing, & C. Fang, "The ramp;d of htr-stac program package: source term analysis codes for pebble-bed high-temperature gas-cooled reactor", Science and Technology of Nuclear Installations, vol. 2018, p. 1-9, 2018. https://doi.org/10.1155/2018/7389121

[53] Xenofontos, T., Panayiota, S., Varvayanni, M., Maillard, J., Silva, J., & Catsaros, N. (2021). Criticality search of an accelerator driven system using the anet code. World Journal of Nuclear Science and Technology, 11(01), 65-72. https://doi.org/10.4236/wjnst.2021.111004

[54] G. Giudicelli, C. Permann, D. Gaston, B. Feng, & A. Abou-Jaoude, "The virtual test bed (vtb) repository: a library of multiphysics reference reactor

models using neams tools", International Conference on Physics of Reactors 2022 (PHYSOR 2022), 2022. https://doi.org/10.13182/physor22-37606

[55] G. Stefani and T. Conti, "Consolidation of the new nuclear calculation methodology of the iea-r1 reactor", Revista Internacional De Ciências, vol. 7, no. 2, 2018. https://doi.org/10.12957/ric.2017.29930

[56] T. Abuqudaira, P. Tsvetkov, & P. Sabharwall, "Assessment of nuclear sensors and instrumentation maturity in advanced nuclear reactors",, 2023. https://doi.org/10.2172/2324979

[56] Y. Wang, W. Chen, L. Zhang, X. Zhao, Y. Gao, & V. Dinavahi, "Small modular reactors: an overview of modeling, control, simulation, and applications", IEEE Access, vol. 12, p. 39628-39650, 2024. https://doi.org/10.1109/access.2024.3351220

[57] C. Forsberg, P. Peterson, & P. Pickard, "Molten-salt-cooled advanced high-temperature reactor for production of hydrogen and electricity", Nuclear Technology, vol. 144, no. 3, p. 289-302, 2003. https://doi.org/10.13182/nt03-1

[58] R. Sinha and A. Kakodkar, "Design and development of the ahwr—the indian thorium fuelled innovative nuclear reactor", Nuclear Engineering and Design, vol. 236, no. 7-8, p. 683-700, 2006. https://doi.org/10.1016/j.nucengdes.2005.09.026

[59] NSTXL, "Military Uses for Nuclear Power," [Online]. Available: https://nstxl.org/military-uses-for-nuclear-power/. [Accessed: Jul. 14, 2024].

[60] IAEA, "How research reactors help make medical imaging possible," [Online]. Available: https://www.iaea.org/newscenter/news/how-research-reactors-help-make-medical-imaging-possible. [Accessed: Jul. 14, 2024].

[61] World Nuclear Association, "Nuclear Power Reactors," [Online]. Available: https://world-nuclear.org/information-library/nuclear-fuel-cycle/nuclear-power-reactors/nuclear-power-reactors. [Accessed: Jul. 14, 2024].

[62] R. L. Murray, "Nuclear Reactors," in Kirk-Othmer Encyclopedia of Chemical Technology, John Wiley & Sons, Inc., 2007. doi: 10.1002/0471238961.1805010313211818.a01.pub3.

[63] E. De Sanctis, S. Monti, and M. Ripani, "Nuclear Reactors," in Energy from Nuclear Fission, Undergraduate Lecture Notes in Physics, Springer, Cham, 2016. doi: 10.1007/978-3-319-30651-3_4.

[64] R. Chaplin, "Genealogy of CANDU Reactors," in The Essential CANDU, Chapter 2, UNENE, 2016.

[65] IEER, "Basics of Nuclear Physics and Fission," [Online]. Available: https://ieer.org/resource/factsheets/basics-nuclear-physics-fission/. [Accessed: Jul. 14, 2024].

[66] J. R. Lamarsh and A. J. Baratta, Introduction to Nuclear Engineering, 3rd ed., Upper Saddle River, NJ: Prentice Hall, 2001.

[67] A. Tehrani, "Executive Beyond Design Basis Analysis: Developments in UK's Approach and Perspective," presented at the IAEA International Expert's Meeting on Severe Accident, Mar. 2014.

[68] INTERNATIONAL ATOMIC ENERGY AGENCY, Application of the Principle of Defence in Depth in Nuclear Safety to Small Modular Reactors, INSAG Series No. 28, IAEA, Vienna, 2024. doi: 10.61092/iaea.w9s3-1k5y.

[69] V. de Vasconcelos, W. A. Soares, A. C. Lopes da Costa, and A. L. Raso, "Deterministic and Probabilistic Safety Analyses," in Advances in System

Reliability Engineering, M. Ram and J. P. Davim, Eds. Academic Press, 2019, pp. 43-75. ISBN 9780128159064.

[70] A. Wielenberg, L. Lovasz, P. Pandazis, A. Papukchiev, L. Tiborcz, P. J. Schöffel, C. Spengler, M. Sonnenkalb, and A. Schaffrath, "Recent improvements in the system code package AC2 2019 for the safety analysis of nuclear reactors," Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Schwertnergasse 1, 50667 Cologne, Germany.

[72] V. G. Snell, "Reactor Safety Design and Safety Analysis," in The Essential CANDU, Chapter 13, UNENE, 2016.

[71] Argonne National Laboratory, "System Analysis Module," [Online]. Available: https://www.anl.gov/nse/system-analysis-module. [Accessed: Jul. 14, 2024].

[73] F. D'Auria, N. Debrecin, and H. Glaeser, "The technological challenge for current generation nuclear reactors," Nuclear Energy and Technology, vol. 5, no. 3, pp. 183-199, 2019. doi: 10.3897/nucet.5.38117.

[74] IAEA, "Open-Source Nuclear Code for Reactor Analysis: ONCORE," [Online]. Available: https://www.iaea.org/topics/nuclear-power-reactors/open-source-nuclear-code-for-reactor-analysis-oncore. [Accessed: Jul. 14, 2024].

[75] J. Cimbala and Y. Cengel, Fluid Mechanics: Fundamentals and Applications, 4th ed. Boston, MA: McGraw-Hill Education, 2020.

[76] C. Young, P.E., Moody Chart for Estimating Friction Factors. [Online]. Available: https://engineerexcel.com/moody-chart/. [Accessed: May 14, 2024]

[77] Forsberg, C., Renault, C., Brun, C., Merle-Lucotte, E., & Ignatiev, V. (2007). Liquid salt applications and molten salt reactors. Revue Générale Nucléaire, (4), 63-71. https://doi.org/10.1051/rgn/20074063

[78] T. Abuqudaira, "Fuel flow modeling under thermal fluidics considerations within operational domains of molten salt reactors," 2023. [Online]. Available: https://doi.org/10.13182/nureth20-40919. [Accessed: Aug. 1, 2024].

[79] J. Swetye, Reading a Centrifugal Pump Curve: Pump Sizing 101. Grundfos, Feb. 19, 2018. [Online]. Available: https://www.pumpsandsystems.com/pumps/reading-centrifugal-pump-curve. [Accessed: Aug. 1, 2024].

[80] H. Dwivedi, H. Goswami, D. Singh, and S. P. Singh, "CFD Analysis of Fluid Flow in a Heat Exchanger Using ANSYS Software," Niet Journal of Engineering and Technology, Sep. 2018. Available: https://www.researchgate.net/publication/361481990. [Accessed: Aug. 24, 2024].