CFD Analyses of Air-Ingress Accident for VHTRs

DISSERTATION

Presented in Partial Fulfillment of the Requirements for the Degree Doctor of Philosophy in the Graduate School of the Ohio State University

By
Tae Kyu Ham
Graduate Program in Nuclear Engineering

The Ohio State University
2014

Dissertation Committee:
Dr. Xiaodong Sun (Advisor)
Dr. Tunc Aldemir
Dr. Richard Christensen
Abstract

The Very High Temperature Reactor (VHTR) is one of six proposed Generation-IV concepts for the next generation of nuclear powered plants. The VHTR is advantageous because it is able to operate at very high temperatures, thus producing highly efficient electrical generation and hydrogen production. A critical safety event of the VHTR is a loss-of-coolant accident. This accident is initiated, in its worst-case scenario, by a double-ended guillotine break of the cross vessel that connects the reactor vessel and the power conversion unit. Following the depressurization process, the air (i.e., the air and helium mixture) in the reactor cavity could enter the reactor core causing an air-ingress event. In the event of air-ingress into the reactor core, the high-temperature in-core graphite structures will chemically react with the air and could lose their structural integrity. Complex multiple phenomena (i.e., diffusion, gravity-driven density-driven flow, natural circulation, and chemical reactions) are involved in an air-ingress accident; however, there is limited experimental data available to understand the air-ingress phenomena. Therefore, Ohio State University (OSU) designed a 1/8th scaled-down test facility to develop an experimental database for studying the mechanisms involved in the air-ingress phenomenon. The current research focuses on the analysis of the air-ingress phenomenon using the computational fluid dynamics (CFD) tool ANSYS FLUENT for better understanding of the air-ingress phenomenon.
CFD is the process of obtaining solutions to a set of coupled non-linear partial
differential equations by numerical approximation. Due to physical modeling and
discretization limitations, it is inevitable that the CFD solutions will have some errors.
Therefore, the uncertainty of the CFD physical models were investigated by benchmark
studies, and mesh refinement studies were performed to minimize the error from the
mesh structure and transient time step. The grid conversion index (GCI) and Richardson
extrapolation method were used to quantify the uncertainty of the simulation.

The anticipated key steps in the air-ingress scenario for guillotine break of VHTR
cross vessel are: 1) depressurization; 2) density-driven stratified flow; 3) local hot plenum
natural circulation; 4) diffusion into the reactor core; and 5) global natural circulation.
However, the OSU air-ingress test facility covers the time from depressurization to local
hot plenum natural circulation. Prior to beginning the CFD simulations for the OSU air-
ingress test facility, benchmark studies for the mechanisms which are related to the air-
ingress accident, were performed to decide the appropriate physical models for the
accident analysis. In addition, preliminary experiments were performed with a simplified
1/30th scaled down acrylic set-up to understand the air-ingress mechanism and to utilize
the CFD simulation in the analysis of the phenomenon.

Previous air-ingress studies simulated the depressurization process using simple
assumptions or 1-D system code results. However, recent studies found flow oscillations
near the end of the depressurization which could influence the next stage of the air-ingress accident. Therefore, CFD simulations were performed to examine the air-ingress mechanisms from the depressurization through the establishment of local natural circulation initiate. In addition to the double-guillotine break scenario, there are other scenarios that can lead to an air-ingress event such as a partial break were in the cross vessel with various break locations, orientations, and shapes. These additional situations were also investigated.

The simulation results for the OSU test facility showed that the discharged helium coolant from a reactor vessel during the depressurization process will be mixed with the air in the containment. This process makes the density of the gas mixture in the containment lower and the density-driven air-ingress flow slower because the density-driven flow is established by the density difference of the gas species between the reactor vessel and the containment.

In addition, for the simulations with various initial and boundary conditions, the simulation results showed that the total accumulated air in the containment collapsed within 10% standard deviation by: 1. multiplying the density ratio and viscosity ratio of the gas species between the containment and the reactor vessel and 2. multiplying the ratio of the air mole fraction and gas temperature to the reference value. By replacing the gas mixture in the reactor cavity with a gas heavier than the air, the air-ingress speed slowed down. Based on the understanding of the air-ingress phenomena for the GT-
MHR air-ingress scenario, several mitigation measures of air-ingress accident are proposed. The CFD results are utilized to plan experimental strategy and apparatus installation to obtain the best results when conducting an experiment. The validation of the generated CFD solutions will be performed with the OSU air-ingress experimental results.
Dedication

This document is dedicated to my parents.
Acknowledgments

I would like to express the deepest appreciation to my advisor Prof. Xiaodong Sun for the opportunity to join this research project. This work would not have been achievable without his guidance and support. I want to thank him for his patience in allowing me to resolve the many issues I encountered while working on this project.

I would also like to thank Prof. Richard Christensen for his encouragement and thoughtful criticism during all our weekly research meetings.

I am very thankful to my committee member Prof. Tunc Aldemir, for his comments and suggestions during my candidacy exam.

The support from the U.S. Department of Energy Nuclear Energy University Programs (NEUP) for this research is gratefully acknowledged. The computing resources provided by the Ohio Supercomputer Center (OSC) are also appreciated.

I would like to thank David Arcilesi, who has worked on the same research project as I have, for his help and for keeping me motivated through my graduate studies. Thanks to
Sai Mylavarapu, Benjamin Doup, Xinquan Zhou, Quiping Lu, and Minghui Chen for helping me in various ways. Thanks to Dr. In Hun Kim for his comments and discussions regarding to this project.

I want to express my gratitude and love to my parents and sister for their unfailing support. I thank my lovely sons, Kyle and Bryan, for their sweet smile. Finally, I want to express my deepest love and thanks to my wife, Borami Lee, for her continued support and help throughout this graduate period.
Vita

2005 ..............................................................B.S. Electrical Engineering, Ajou University, Suwon, South Korea

2008 ...............................................................M.S. Energy systems, Ajou University, Suwon, South Korea

2009 to present ..............................................Graduate Research Associate, Nuclear Engineering Program, Department of Mechanical and Aerospace Engineering, The Ohio State University

Fields of Study

Major Field: Nuclear Engineering
# Table of Contents

Abstract ........................................................................................................................................ ii

Dedication ................................................................................................................................ vi

Acknowledgments ...................................................................................................................... vii

Vita ............................................................................................................................................. ix

Fields of Study .......................................................................................................................... ix

Table of Contents .................................................................................................................... x

List of Tables ............................................................................................................................ xv

List of Figures ........................................................................................................................... xvii

Chapter 1: Introduction ............................................................................................................. 1

1.1. Research background ...................................................................................................... 1

1.2. Statement of the problem .............................................................................................. 7

1.3. Research objectives ....................................................................................................... 9

1.4. Dissertation outline ..................................................................................................... 11

Chapter 2: CFD results uncertainty quantification ................................................................. 14

2.1 Introduction ..................................................................................................................... 14
3.5. Depressurization of prototypic design, GT-MHR .................................................. 55

3.5.1. Confinement design .......................................................................................... 61

3.5.2. Compartment design effects ............................................................................. 66

3.6. Discussion ........................................................................................................... 71

Chapter 4: CFD as a design tool for the design of the OSU scaled down air-ingress test
facility .............................................................................................................................. 72

4.1. Small-scale experiments using acrylic set-up for density driven flow ............... 73

4.2. CFD model of the OSU air-ingress test facility ................................................ 81

4.3. Depressurization and flow oscillation .................................................................. 87

4.4. The proceeding stage of the depressurization .................................................... 93

4.4.1. Heat transfer from the reactor vessel internal structures ............................... 93

4.4.2. Consequences of flow oscillation on density-driven air-ingress ..................... 100

4.5. Local natural circulation ..................................................................................... 103

4.6. Discussion .......................................................................................................... 108

Chapter 5: CFD simulations of the OSU air-ingress test facility ................................. 110

5.1 Simulation matrix .................................................................................................. 110

5.2. Temperature decrease due to the adiabatic expansion during the depressurization
process ......................................................................................................................... 114

5.3. Local hot plenum natural circulation .................................................................. 118
5.4. Density difference of gas species between the containment and the vessel........ 122

5.5. Effects of Break location and size........................................................................ 125

5.6. Generalize the density-driven flow .................................................................... 127

5.7. Discussion ....................................................................................................... 135

Chapter 6: Mitigation measures ................................................................................. 136

6.1 Previous studies................................................................................................. 136

6.2. Reactor building design effect to air-ingress accident ..................................... 137

6.3. Filling inert gas in the reactor cavity................................................................. 139

6.3.1. Filling the reactor cavity with inert gas during normal operation .............. 139

6.3.2. Injecting argon gas near the end of depressurization stage ....................... 140

6.4 Physical barrier to air flow .............................................................................. 151

6.5. Discussion ....................................................................................................... 151

Chapter 7: Conclusion and future work ................................................................... 153

7.1 Conclusion....................................................................................................... 153

7.2 Future work ..................................................................................................... 156

References.............................................................................................................. 157

Appendix A: 1-D Depressurization analytical solution........................................... 164

Appendix B: Mass fraction contour of density driven flow benchmark study using different viscous model for the experiment of Grobelbauer et al. [30] ................. 170
Appendix C: side view of Air mass fraction contour for selected CFD simulations...... 174
List of Tables

Table 3.1. Experiment dimensions and depressurization time [34] .................................. 34
Table 3.2. FLUENT model mesh characteristics .............................................................. 37
Table 3.3. GCI (%) values for depressurization process .................................................. 40
Table 3.4. Gas combination used for the experiments ...................................................... 49
Table 3.5. Uncertainty quantification of the density driven flow ................................... 52
Table 3.6. CFD results of gas combinations ..................................................................... 54
Table 4.1. Dimensions of reactor prototype and scaled-down set-up ............................... 73
Table 4.2. FLUENT model mesh information .................................................................. 82
Table 4.3. GCI (%) values for OSU scale-down facility depressurization process .......... 83
Table 4.4. Solution verification results for density driven flow ....................................... 86
Table 4.5. Initial conditions of Figure 4.11 CFD simulations ......................................... 88
Table 4.6. Initial conditions of CFD simulations for various species concentration and pressure .................................................................................................................. 101
Table 4.7. Air mole fraction in the containment and the pressure vessel ....................... 102
Table 4.8. Air-ingress mass flow rate (g/s) ..................................................................... 103
Table 5.1. DEG test matrix ............................................................................................. 111
Table 5.2. Partial break test matrix .................................................................................. 113
Table 5.3. Pressure and temperature after depressurization ........................................... 115
Table 6.1. Inert gas density and price ............................................................................. 140
Table 6.2. Air concentration changes with injection and without injection (mol) ........ 143
List of Figures

Figure 1.1. GT-MHR below grade installation ................................................................. 4
Figure 1.2. Progression of an air-ingress scenario [13] ................................................. 5
Figure 2.1. Graphite oxidation regimes ....................................................................... 18
Figure 2.2. V&V overview – sources of error ............................................................... 21
Figure 3.1. Compressibility factor of helium [24] ......................................................... 32
Figure 3.2. Compressibility factor of air [25] ............................................................... 32
Figure 3.3. Depressurization test facility [34] ............................................................... 34
Figure 3.4. (a) Geometry of discharge model and (b) cut view of the mesh structure
through the center plane ............................................................................................. 36
Figure 3.5. Mesh refinement study with three different mesh sizes ............................. 39
Figure 3.6. Coarse mesh solution and uncertainty band ............................................... 41
Figure 3.7. Time refinement study with three different time steps ............................... 42
Figure 3.8. Depressurization results with various vessel and nozzle sizes .................... 44
Figure 3.9. Schematic diagram of gravity current ......................................................... 45
Figure 3.10. The heavy current plume height [29] ......................................................... 47
Figure 3.11. The heavy current velocity function of a density ratio ............................. 48
Figure 3.12. High density ratio density driven flow experimental set-up ..................... 49
Figure 4.18. Gas mixture compositions at the center of the hot duct at t = 5 s ................. 99
Figure 4.19. Amount of air in the vessel with various initial conditions ...................... 103
Figure 4.20. Fluent calculation of air mass fraction for a 1/8th scaled-down facility .... 105
Figure 4.21. Accumulated air in the experimental facility .............................................. 106
Figure 4.22. Density contour of isothermal (top) and non-isothermal (bottom) at 40 seconds ............................................................................................................................................. 107
Figure 5.1. Temperature decrease due to depressurization and temperature changes by the heat transfer from internal structures over time ........................................................ 116
Figure 5.2. Variation of streamlines according to time .................................................. 118
Figure 5.3. Air accumulation in the vessel over time ....................................................... 119
Figure 5.4. Air mole fraction of DEG 6 (left) and DEG 23 (right) at 48 sec ............... 120
Figure 5.5. Density contour of DEG 6 (left) and DEG 9 (right) at 48 sec ...................... 120
Figure 5.6. Temperature contour of DEG 6 at 48 sec ...................................................... 121
Figure 5.7. Flow velocity of DEG 6 (left) and DEG 23 (right) at 48 sec ....................... 121
Figure 5.8. The OSU air-ingress test facility thermo couple locations (red lines) ....... 122
Figure 5.9. Air mole fraction measurement at thermo couple location 3 ....................... 123
Figure 5.10. Air mole fraction measurement from the initial time the air plume reaches the thermocouple, divided by the initial air mole fraction in the containment ............. 124
Figure 5.11. Viscosity ratio division from the result of Figure 5.10 ............................. 124
Figure 5.12. Partial break size, shape, and orientation .................................................. 125
Figure 5.13. Partial break simulation of different shapes and orientations a) semicircle, b) slit, c) circumferential ........................................................................................................ 126
Figure 5.14. Accumulated air in the vessel for the change of air mole fraction in the containment (top) and the collapsed data by density, viscosity, and air concentration ratio (bottom)........................................................................................................................... 128

Figure 5.15. Accumulated air in the vessel for the change of temperature in the containment (top) and the collapsed data by density, viscosity, and air concentration ratio ......................................................................................................................................... 129

Figure 5.16. Accumulated air in the vessel for the change of concentration in the containment with vessel temperature 973K (top) and the collapsed data by density, viscosity, and air concentration ratio .............................................................................................................................................. 130

Figure 5.17. Accumulated air in the vessel for the change of concentration in the containment with vessel temperature 673K (top) and the collapsed data by density, viscosity, and air concentration ratio .............................................................................................................................................. 131

Figure 5.18. Accumulated air in the vessel for the change of concentration in the containment with vessel temperature 373K (top) and the collapsed data by density, viscosity, and air concentration ratio .............................................................................................................................................. 132

Figure 5.19. Collapsed data of the data of Figure 5.14 Figure 5.18 ............................... 133

Figure 5.20. Accumulated air in the vessel DEG 1-6 and the collapsed data by density, viscosity, and air concentration ratio .............................................................................................................................................. 134

Figure 6.1. Air mass fraction change over time – air flow back to cavity 1 ................... 138

Figure 6.2. Compartmentalized CFD model with argon injection port at the bottom of cavity 1.............................................................................................................................................. 141

Figure 6.3 Mass fraction of air (inert gas injection) ....................................................... 142
Chapter 1: Introduction

1.1. Research background

A Very High Temperature Reactor (VHTR) is one of the Generation IV (Gen-IV) nuclear reactors proposed by the U.S. Department of Energy (DOE). The VHTR concept is a helium-cooled, graphite-moderated, thermal spectrum reactor. This reactor is an advanced concept reactor that has higher safety and reliability, and improved economic characteristics, as compared to the current nuclear power reactors [1-5]. The VHTR is designed to avoid fission product release under any conditions, even beyond design basis accidents, by utilizing passive safety systems and inherent materials characteristics to satisfy its safety design goal. Helium coolant is neurotically transparent, an inert gas, and is in a single phase under all conditions. The TRISO coated fuel particle maintains its strength at high temperatures and minimizes the radioactive fission product release. The graphite moderator has high temperature stability, long response time and large heat capacity. The reactor accomplishes the safety functions by a large negative temperature coefficient, passive heat removal systems, TRISO coated particle fuel and graphite moderator.
By utilizing the passive safety features of the VHTR and minimizing the reliance on the active safety systems, a VHTR achieves the reactor safety goal, maintaining plant protection and control of radionuclide release. Therefore, the main concern of the VHTR is the failure of its passive safety systems. One possible failure of the VHTR is the small oxidation stability of graphite. Since the main components in the VHTR reactor consists of graphite, an air-ingress accident could lead to potentially severe accidents when air enters into the reactor core.

Since a VHTR uses graphite as a core structure, if there is a break on the pressure boundary, the air (i.e., the air and helium mixture) in the reactor cavity could ingress into the reactor core causing an air-ingress event. The air-ingress accident could cause a temperature increase in the reactor core and degrade the graphite structural integrity by the chemical reaction between high temperature graphite structure and oxygen. Based on a phenomena identification and ranking table (PIRT) exercise in major topical areas of NGNP, the air-ingress accident is categorized as the most significant phenomenon in the thermal-fluid area [6]. In the early stages of the VHTR safety analysis, the break size of the air-ingress accident was limited to 33 cm$^2$, which was considered as the largest pipe break possible on the vessel [7]. In this event, air-ingress through a small pipe break is relatively limited; therefore, the amount of graphite oxidation is small with no significant damage to the reactor core. Ball [8] demonstrated the effectiveness of accident prevention and mitigating capabilities of the VHTR passive safety system, however, the degradation of the graphite structure by oxidation was not considered.
Moormann [9] presented that graphite burning was sustained when the heat production from graphite oxidation was greater than the heat removal. The graphite self-ignition temperature is between 650-700°C and the normal operating temperature at the core exit in VHTRs is higher than 850°C. The temperature of the graphite support column in the hot plenum always exceeds the graphite ignition temperature. In addition, several previous studies indicate that the worst-case scenario for an air-ingress accident is the double-ended guillotine break of the co-axial cross vessel that connects the reactor vessel to the power conversion unit [10-12].

In the VHTR, the reactor pressure vessel, which is located within a reactor cavity, is filled with air during normal operation as shown in Figure 1.1. During the normal operation, the pressures in the vessel and containment are 7 and 0.1 MPa, respectively. Once the break is initiated, the coolant (helium) of the reactor is discharged into the containment. Therefore, after the depressurization, the air-helium mixture in the cavity may enter the reactor pressure vessel through the break. Since the oxygen in air chemically reacts with high-temperature graphite that is used as moderator and structural material, this leads to potential damage of the core bottom graphite structures as well as release of carbon monoxide [12] unless mitigation actions are taken.
There are two hypothesized modes of air ingress: molecular diffusion and density-driven stratified flow. Generally, molecular diffusion is a slower process than the density-driven stratified flow. Therefore, if the density-driven stratified flow dominates during the accident, air will ingress into the reactor much faster. However, the two air-ingress mechanisms, molecular diffusion and density-driven stratified flow, may be important for different accident scenarios, specifically for different break sizes, shapes, and locations. Therefore, it is essential to understand the dominant mechanisms for each
scenario in the air-ingress process so that mitigation measures could be considered in the VHTR design and operation.

The progression of the anticipated density-driven stratified flow dominant air-ingress scenario is shown in Figure 1.2. The air-ingress accident is initiated by a cross-vessel break, which leads to a rapid reactor depressurization. Helium in the pressure vessel is rapidly discharged to the reactor cavity. After the depressurization, it is postulated that a pair of gravity-driven fronts would propagate along the upper and lower part of the hot-duct surface. An air-helium mixture flows into the pressure vessel through the bottom of the cross vessel and helium flows out of the pressure vessel through the top region of the cross vessel. These flows are induced by the density difference between the

![Figure 1.2. Progression of an air-ingress scenario [13]](image-url)
helium in the pressure vessel and air-helium mixture in the reactor cavity. This is referred to as the density-driven stratified air-ingress flow. After air fills the hot plenum, another counter-current flow would occur, which is induced by the density difference of the gas species between the heated mixture near the high-temperature support column structures and relatively cold mixture in the reactor cavity. This localized natural circulation event is referred to as hot plenum natural circulation, which is a convective flow in the hot plenum as shown in Figure 1.2-(3), and is distinct from global natural circulation that flows through the core as shown in Figure 1.2-(5).

When the buoyancy force is greater than the flow resistance through the core, part of air-helium mixture in the hot exit plenum will move upward into the reactor core. This flow into the reactor core will be further enhanced by the heat generated due to the graphite oxidation in the core. This will force the flow into the core if the buoyancy force in the lower plenum exceeds the energy barrier threshold for initiating the onset of global natural circulation. However, if the intruded flow does not have sufficient energy to overcome the hydrostatic head of the core fluids, the process will be dominated by a molecular diffusion process. Finally, if the buoyancy force is enough to generate the global natural convection flow, massive air-ingress begins.

For the demonstration of the diffusion dominant air-ingress accident, Takeda [10] developed an experimental apparatus that consists of a reverse U-shaped tube and a gas tank. One of the vertical sides of the u-tube is heated and the other vertical tube is cooled.
by water to measure the effect of diffusion and natural convection flow during air-ingress. Another set of tests were conducted with the Natural Convection in Core with Corrosion (NAKOK) experiments to assess air-ingress. From these experiments, it was found that after depressurization process, the main mechanism of air-ingress is molecular diffusion. Therefore, there is a grace period before natural circulation of air through the core occurs.

1.2. Statement of the problem

Since the cross vessel and its welding to the reactor vessel are designed, fabricated, tested and installed following ASME section III rules to ensure the 60 year plant life. The possibility of cross vessel failure is extremely low. However, the cross vessel is located between the reactor pressure vessel and the power conversion unit to connect the two units. If displacement of either units took place by seismic events or another external shock, an excess load would be concentrated on the cross vessel, which may cause to break it. Through the break air could enter the reactor and the graphite oxidation would likely occur. A previous study shows that approximately 25% of the mechanical strength of the graphite structure might be lost by a 4.5% graphite burn-off during the progress of an air-ingress event [14]. Therefore, despite the frequency of the cross vessel failure accident being much lower than design basis accidents (DBAs), an investigation of the air-ingress accident, which has the possibility of causing reactor core structure failure and large release of radioactive material, is necessary to ensure the safe operation of the VHTR.
There are limited experimental data that are readily available to understand the aforementioned air-ingress scenario; therefore, development and validation of air-ingress-related experimental models are a high priority to determine the dominant mechanism for each scenario. Since it is not practical to investigate all of the scenarios through actual experiments, a commercial computational fluid dynamics (CFD) tool, FLUENT, was used in this study to analyze the air-ingress accident of the gas-turbine modular helium reactor (GT-MHR), a prismatic VHTR design developed by General Atomics [15].

Since the CFD tool can be used to perform accident analysis and predict performance of the system without actual modification or installation of the system, it can allow for visualizing of the required information to aid in understand the phenomena. The growth in computing power available for the CFD analysis allow for it to be applied in various engineering applications. However, there is still a strong requirement to validate computational results with experimental results. CFD physical models may not accurately describe a physical phenomenon. Therefore, through an understanding of what is important for the analysis, good physical modeling can be acquired. In addition, physical models used in CFD often deliberately simplify the real-life physical problem to reduce the computational cost. Therefore, validation of the computational solution is required to identify how accurately the physical models express the real physical phenomena. There were studies validating the physical models for the air-ingress accident [16, 17], however, the studies qualitatively compared the experimental data with the computational solutions.
In this study, the CFD physical models for the phenomena proposed for the air-ingress accident will be validated by conducting benchmark studies. In addition, solution verification will be performed to quantify the uncertainty of the solution. The solutions will be used to understand the air-ingress phenomenon, to propose accident mitigation measures and to help determine the optimum placement of the experimental instrumentation to experimentally capture the important parameter.

1.3. Research objectives

The uncertainty of the physical models and numerical solutions will be quantified to provide verified solutions to understand the air-ingress phenomenon. The quantified results are an integral effect of the physical models used in the CFD simulations. It would support the accuracy of CFD simulation results and the ability of CFD as an analysis tool. This way, CFD results could provide additional insights into the air-ingress phenomenon that is difficult to discover through experimental or theoretical analysis alone. Additionally, air-ingress preventive measures will be proposed. In what follows, the objectives of the current research are outlined.

a. To perform preliminary CFD calculations to give design guidelines for the experimental set-up in order to determine which parameters should be measured in the experiments. The preliminary CFD simulation results show the fluid flows
inside the hot plenum so that the measurement sensor locations and the effectiveness of the design can be examined.

b. To develop a one-dimensional (1-D) analytical solution to predict the depressurization process, which will establish the initial conditions for the experimental runs. Even though the depressurization time scale is expected to be small, typically less than one second, a large amount of computational resources are required for three-dimensional (3-D) CFD simulations. Therefore, by calculating the depressurization process using an analytical solution, considerable computational resources can be saved while still being able to define the initial condition of the experiment.

c. To quantify the uncertainty of the numerical solutions. Input property error, computer round-off error, iterative error, and discretization error will be identified. The discretization error will be systematically analyzed using the Richardson extrapolation method and the grid convergence index (GCI) to quantify the uncertainty.

d. To perform CFD analyses of air-ingress accidents, for a wide range of possible scenarios, including breaks of varying types and orientation, including the double-ended guillotine break. Each break will be exposed to difference density conditions, which will cover the range of density differences expected in the air-
ingress accident. Note that the density difference is defined as the difference between the density of the hot vessel helium and the relatively cooler air-helium mixture in the containment.

e. To propose air-ingress mitigation measures to prevent air-ingress accident progression by understanding the air-ingress mechanism and the reactor systems.

1.4. Dissertation outline

In summary, the contributions that the present study made are as follows:

a. Developed CFD models and performed a benchmark study to provide uncertainty of the physical model for the analysis of the air-ingress accident analysis
b. Developed a depressurization model for the pressure vessel depressurization
c. Developed a mesh structure to minimize the uncertainty and the quantifying this uncertainty using Richardson’s extrapolation and GCI index
d. Generated CFD solutions for various air-ingress scenarios to be utilized in determining the optimum placement of the instrument to experimentally capture the important parameter
e. Proposed air-ingress mitigation measures

In Chapter 2, the research for the current study provides an introduction of the physical models and theories used in the air-ingress analysis, and provide uncertainty quantification methods.
In Chapter 3, the performance of the benchmark studies to identify the uncertainty of the physical models used for the air-ingress study are discussed. In addition, the researcher introduces a 1-D analysis of the depressurization for the depressurization process to verify the CFD solution, and discuss the investigation of the depressurization stage of the air-ingress phenomena to determine the initial condition for the OSU air-ingress test facility.

In Chapter 4, the investigations of the air-ingress phenomena using the OSU air-ingress test facility design are discussed. Before starting the analysis, the researcher introduced the small-scale experimental set-up to investigate how CFD can be used in the air-ingress analysis. In addition, the depressurization analysis performed to identify the pressure oscillation near the end of the depressurization process and the effects of the oscillation to the next air-ingress scenarios are investigated.

In Chapter 5, various air-ingress CFD simulation data are generated to understand the air-ingress mechanism. In addition, the generated data will be used to help determine how to install the experimental apparatus. The generated solution will be validated when the OSU test facility are built up.

In Chapter 6, the air-ingress mitigation measures are proposed. By injecting argon gas into the reactor cavity, the amount of the air ingress is reduced and a proposed
gate which closed by gravitational force would help prevent the air flow back to the reactor cavity.

In Chapter 7, conclusions based on the previous chapters are presented and recommendations for the future research are provided.
Chapter 2: CFD results uncertainty quantification

2.1 Introduction

CFD is the simulation of thermal-fluid systems to describe temperature, velocity, and fluid properties throughout a region of interest. CFD solves governing equations of the continuity, momentum, and energy equations using numerical methods. Since mass, momentum, and energy are always conserved, the governing equations are applicable for physical systems which adhere to the continuum hypothesis. Improvements of computer resources have allowed for significant progress in CFD. The simulation of fluid phenomena can be used to capture results from difficult full-scale model experiments.

However, CFD simulation physical models may not accurately describe a physical phenomenon. Therefore, through an understanding of what is important for the analysis, good physical modeling can be acquired. In addition, the physical models used in CFD often deliberately simplify the real-life physical problem to reduce the computational cost. Therefore, validation of the computational solution is required, if the correct physical models are used for the analysis. In this chapter, the physical models used in the air-ingress analysis and the uncertainty quantification for the numerical solutions are discussed.
2.2. Physical models for air-ingress analysis

Complex multiple phenomena, (i.e., diffusion, gravity-driven density-driven flow, natural circulation, and chemical reactions) are involved in air-ingress accidents.

2.2.1. Species transport and reaction model

FLUENT can define the species transports and chemical reactions in species models. When the model is used, FLUENT solves the conservation equations for each species to predict the mass fraction of each species, \( Y_i \), by solving a convection-diffusion equation for each species. The equation can be expressed as

\[
\frac{\partial}{\partial t} \left( \rho Y_i \right) + \nabla \cdot \left( \rho \vec{v} Y_i \right) = \nabla \cdot \vec{J}_i + R_i + S_i
\]  

(2.1)

where, \( R_i \) is the net rate of production of species \( i \) by chemical reaction and \( S_i \) is the rate of creation by addition from the dispersed phase plus other sources. The equation will be solved for \( N-1 \) species, where the \( N \) is the total number of species involved in the calculation. The \( N_{th} \) species will be calculated to conserve the mass. Therefore, the \( N_{th} \) species mass fraction is one minus the total mass fraction of \( N-1 \) species. To minimize the numerical error, the most abundant species need to be selected as the \( N_{th} \) species.

In the equation(2.1), \( \vec{J}_i \) is the diffusion flux of species \( i \). FLUENT used the dilute approximation express as,
where, $D_{i,m}$ is the mass diffusion coefficient for species $i$ in the mixture and $D_{r,i}$ is the thermal diffusion coefficient.

### 2.2.2. Heat transfer model

FLUENT solves the energy equation as follow,

$$
\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot \left( k \nabla T - \sum_i h_i \vec{J}_i + (\tau \cdot \vec{v}) \right) + S_h
$$

(2.3)

Where, $k$ is the conductivity, and $\vec{J}_i$ is the diffusion flux of species $i$. The first three terms of the right side of the equation represent energy transfer by conduction, diffusion, and viscous dissipation respectively. $S_h$ is the heat generation by chemical reaction, and other heat sources.

Energy $E$ can be defined as,

$$
E = h - \frac{p}{\rho} + \frac{v^2}{2}
$$

(2.4)
where enthalpy $h$ is defined as,

$$ h = \sum_i Y_i h_i $$

(2.5)

2.2.3. Graphite oxidation model

When air enters into the reactor the high temperature graphite structures will have a chemical reaction with oxygen. The situations could cause a significant temperature increase by exothermic reaction, damage of structural integrity and accumulation of explosive and toxic $CO$ gas in the reactor. Therefore, the graphite oxidation rate during experiment should be analyzed using FLUENT before the actual test.

The graphite oxidation is the reaction that the reactants can only come in and go out by diffusion due to the no-slip boundary condition at the wall. Therefore, the reactants mass flux by diffusion must equal the graphite reaction rate and the overall graphite reaction regime divided into three as shown in Figure 2.1 [20-22].
Regime 1: The temperature range is 673 to 900K. This regime is called the chemical kinetic regime. Because the reaction rate is slow, there is enough time for the reactant material to diffuse into graphite pores. The reaction surface of pores is much more than the external graphite surface, the oxidation happens mostly inside graphite. Therefore, the reaction in this regime degrades the mechanical strength of the graphite components.

Regime 2: The temperature range is 900 to 1123K. The diffusion into the pores becomes limited with temperature increase. Even though the reaction rate increases exponentially with increasing temperature, the inability of the reactants to diffuse into the graphite, limits the amount of internal oxidation. Therefore, the graphite reaction eventually shifts to the external surface as the temperature increases.
Regime 3: The temperature range is over 1,123K. Since the reaction is fast, the reaction occurs at the external surface only. Since the available reactant is the limiting factor for this regime, only surface corrosion of the graphite components is observed.

2.3. Uncertainty quantification

Uncertainty quantification procedures are the primary means of assessing accuracy in computational simulations. The uncertainty quantification of the numerical solution is divided in two parts, verification and validation.

Verification is the process of determining if the numerical solution is correct. The verification process consists of two types: code verification and solution verification. Code verification is related to mistakes in the source code and algorithms. By using a widely-used commercial CFD code, ANSYS FLUENT, this code verification is excluded in this study. The solution verification is the process of assuring the accuracy of input data and estimating the numerical solution error.

Validation is the process of determining if the right equations are used for the solution. The American Society of Mechanical Engineers Verification and Validation (ASME V&V) standard defines validation as estimating the modeling error within an uncertainty range [23]. This is obtained by comparing the simulation solutions and experimental measurement at a particular validation point.
The discrepancy between the simulation result (S) and experimental data (D), called comparison error (E), can be defined as,

\[ E = S - D \tag{2.6} \]

Because the simulation error is \( \delta_S = S - \text{True Value} \) and experiment error is \( \delta_{\text{exp}} = D - \text{True Value} \),

\[ E = \delta_S - \delta_{\text{exp}} \tag{2.7} \]

The simulation error consists of modeling, simulation input, and numerical error. Then the Eq. (2.7) can be decomposed as,

\[ \delta_{\text{model}} = E - (\delta_{\text{input}} + \delta_{\text{num}} - \delta_{\text{exp}}) \tag{2.8} \]

\( \delta_{\text{input}} \) and \( \delta_{\text{num}} \) is defined from the solution verification process, and \( \delta_{\text{exp}} \) is defined from measurement. Figure 2.2 shows the relationships between errors in the simulations and measurement.
If the concepts of error and uncertainty used in experimental data analysis defining a validation uncertainty, $u_{val}$, to express uncertainty of the modeling error, then the modeling error falls within the range,

$$
\delta_{model} = E \pm u_{val}, \text{ where } u_{val} = \sqrt{u_{input}^2 + u_{num}^2 + u_{exp}^2} \quad (2.9)
$$

Therefore, the input, numerical, and experimental uncertainty should be defined precisely to obtain the correct modeling error.

2.3.1. Input uncertainty

CFD uses the experimentally-determined input parameters that contain uncertainty. Therefore, it is required to have an uncertainty contribution from each
parameter. By defining the individual contribution in the simulation, the parameters that require reducing the uncertainty can be determined. The input uncertainty propagation for a simulation result \( S \) with \( n \) uncorrelated parameters is,

\[
\begin{align*}
    u^2_{\text{input}} &= \sum_{i=1}^{n} \left( \frac{\partial S}{\partial X_i} u_{X_i} \right)^2
\end{align*}
\]  

(2.10)

where \( S, u_{X_i}, \) and \( X_i \) are simulation results, corresponding standard uncertainty in input parameter \( X_i \), and input parameter, respectively. In the Eq. (2.10), \( \frac{\partial S}{\partial X_i} \) represents the change of a result to a unit change in each parameter. Thus, it is termed the sensitivity coefficient. The term “finite difference” used here refers to the parameter space not the finite difference in the space/time discretization algorithm. The procedure is first run on the simulation with fixed parameter \( X_i \), and the second run is with a perturbed value \( (X_i + \Delta X_i) \) for input parameter \( X_i \). Then a finite difference approximation in parameter space is used to compute the sensitivity coefficient.

\[
\begin{align*}
    \frac{\partial S}{\partial X_i} &= \frac{S(X_1, X_2, \ldots, X_i + \Delta X, \ldots + X_n) - S(X_1, X_2, \ldots, X_i, \ldots X_n)}{\Delta X}
\end{align*}
\]  

(2.11)
2.3.2. Numerical uncertainty

There are three uncertainty contributors in CFD calculation: round-off error, iterative error, and discretization error. The round-off error is a consequence of the representation of floating point number. For single-precision computations, residual can drop to six orders of magnitude before hitting round-off. Residuals in FLUENT double-precision simulations can drop up to 12 orders of magnitude. Therefore, round-off errors are not significant when compared with other errors.

All numerical approaches, solving non-linear equations, estimate a new solution estimate by using an iterative method. Therefore, inevitably, any iterative/numerical solution procedure will only give a solution which is converged relative to some criteria. If the gradient is large, then the new estimate may be worse than the old estimate and if the convergence criteria are too large, then the solutions would have large error. In principle, it can be reduced to machine precision. This may not always be possible in complex flow and transient calculations. It is suggested that iterative error roughly reduced to two to three orders of magnitude below the discretization error is sufficient to have a negligible influence.

Discretization errors occur from the approximation of the governing flow equations to algebraic expressions in a discrete domain of space and time. The numerical solution asymptotically approaches the exact solution and the error approaches zero as the grid space and the transient time step decrease. However, it is difficult to achieve
numerical solutions that are completely accurate, due to limited computational resources. Usually, this is the main contribution to the numerical error. This error is mainly dependent on mesh quality, size, and time step. Therefore, the discretization error of the numerical solution is quantified by using the GCI Index and the Richardson extrapolation method, instead of obtaining the exact numerical solution with infinitely small grid space and time step.

2.3.3. Experimental uncertainty

Any experimental measurement has error, which results in a difference between the measured value and the true value. Because the true value is unknown, the total error in a measurement cannot be known, therefore, only its uncertainty can be estimated. The experimental measurement error consists of systematic error and random error.

The experimental result that is determined from $J$ measured variables is,

$$r = r(X_1, X_1, \ldots, X_i, \ldots, X_J)$$

(2.12)

Then the standard uncertainty of the result, $u_r$, is,

$$u_r = \sqrt{h_r^2 + s_r^2}$$

(2.13)
where $b_r$ is the systematic uncertainty of the result

$$b_r^2 = \sum_{i=1}^{n} \left( \frac{\partial R}{\partial X_i} b_i \right)^2$$  \hspace{1cm} (2.14)

and $s_r$ is the random uncertainty of the result

$$s_r^2 = \sum_{i=1}^{n} \left( \frac{\partial R}{\partial X_i} s_i \right)^2$$  \hspace{1cm} (2.15)

where $b_i$ is the systematic uncertainties of the measurements, and $s_i$ is the random uncertainties of the measurements.

### 2.3.4. Richardson extrapolation

The average 3-D grid space $h$ is calculated by,

$$h_i = \left[ \frac{V}{N_i} \right]^{1/3}, \quad i = 1, 2, 3$$  \hspace{1cm} (2.16)

where, subscripts 1, 2, and 3 represent the fine, medium, and coarse grids, respectively, and $V$ and $N$ are the volume and the number of cells of the model, respectively.
Numerical solution $f$ can be expressed by the series expansion,

$$f_{\text{numerical}} = f_{\text{exact}} + Ah^p + HOT$$

(2.17)

where $p$ is the order of convergence, $h$ is grid space for spatial discretization and time step for temporal discretization, $A$ is a constant, and $HOT$ are higher order terms. As the grid space is refined, the numerical solution asymptotically approaches close to the exact solution and the error approaches zero. However, it is hard to achieve exactly accurate numerical solutions because CFD solves coupled non-linear equations.

$$\text{Error} = f_{\text{exact}} - f_{\text{numerical}} \approx Ah^p$$

(2.18)

The numerical solution may be even further from exact solutions due to the limited computing resources. Therefore quantifying the numerical error is important. In this paper, the discretization error is quantified with the Richardson extrapolation method. The Richardson extrapolation method is a method of estimating higher-order solutions and quantifying the error range systematically by a series of solutions located in the asymptotic range. The following equations are the series expansions with two different $h$ values. Subscript 1 and 2 represents fine and medium grid space, respectively.
By combining these two equations to remove the constant $A$, a higher-order solution can be obtained. However, it cannot be guaranteed if the two solutions are within an asymptotic range. Therefore, one more calculation is required to obtain the observed order of convergence with coarse mesh. Then, observed order of convergence can be calculated iteratively as Eq. (2.21) with grid ratio $r_{21} = h_2 / h_1$, $r_{32} = h_3 / h_2$.

\[
\log\left(\frac{f_3 - f_2}{f_2 - f_1}\right) + \log\left(\frac{r_2^{p_1} - 1}{r_2^{p_2} - 1}\right) = \log(r_{21})
\]

(2.21)

Once the observed order of convergence is calculated, an extrapolated value can be calculated and the relative error of the extrapolated solutions can be estimated with the error of two grid solutions, as follows.

\[
f_{\text{extrapolated}} \approx \frac{r_2^{p_1} f_1 - f_2}{r_2^{p_1} - 1}
\]

(2.22)
2.3.5. Grid convergence index

The grid convergence index (GCI) provides an estimate of the discretization error of the finest grid solution. The GCI is based on a generalized Richardson extrapolation involving the comparison of discrete solutions for the CFD solution and the extrapolated value, multiplied by a safety factor which is obtained through empirical studies. As shown in Eq. (2.24), the GCI value is the multiplied safety factor, \( F_s \), from the relative error, Eq. (2.23). \( F_s = 3 \) is recommended for unstructured mesh refinement to represent the GCI with a 95% confidence interval [26].

\[
E_{\text{fine}} = \frac{f_{\text{extrapolated}} - f_1}{f_1} = \frac{f_1}{1 - r_{21}^p}
\]  \hspace{1cm} (2.23)

\[
GCI_{12} = F_s \left| \frac{f_2 - f_1}{f_1} \right|, \quad GCI_{23} = F_s \left| \frac{f_3 - f_2}{r_{23}^p - 1} \right|
\]  \hspace{1cm} (2.24)

The range of convergence for the solutions can be checked by the two GCI values,

\[
C = \frac{r^p GCI_{12}}{GCI_{23}}
\]  \hspace{1cm} (2.25)
2.4. Summary

In this section, the method of the uncertainty estimation by using the GCI index and the Richardson extrapolation method is presented. Among the types of errors associated with CFD analysis, discretization error is the main contributor to numerical error. Discretization error occurs from the approximation of the governing flow equations to algebraic expressions in a discrete domain of space and time. Since the air-ingress accident is a time-dependent problem, the uncertainty from the discretization errors from both the grid space and the time step should be investigated. The refinement studies to quantify the uncertainty are conducted in the following sections according to the method described in this chapter.
Chapter 3: CFD application for air-ingress accident

3.1. Introduction

Oh et al. suggests that the anticipated sequence of events during an air-ingress accident following a guillotine break of the GT-MHR cross vessel is as follows: 1) depressurization, 2) density-driven stratified flow, 3) local hot plenum natural circulation, 4) diffusion into the reactor core, and 5) global natural circulation [13]. The scope of this research covers the depressurization, density-driven stratified flow, and the local hot plenum natural circulation. The FLUENT physical model application for the air-ingress accident mechanism will be investigated in this chapter.

3.2. Material properties of gas species

The pressure of helium, which is used as a coolant in the reactor, changes from 7 MPa to atmospheric pressure during the air-ingress depressurization process. Since ideal gas law was used for the density calculations in the CFD simulations, and kinetic-theory, which is valid when the ideal gas law is applicable, was used for the thermal conductivity, viscosity, and diffusion coefficient, the compressibility factor of the helium and air should be checked. The compressibility factor is the ratio of the molar volume of a gas
to the molar volume of an ideal gas. It shows how the gas behavior is deviated from the ideal gas behavior. The compressibility factor is expressed as,

\[
Z = \frac{V_m}{(V_m)_{\text{ideal}}} = \frac{PV_m}{RT}
\]  

(3.1)

where, \( V_m \), \((V_m)_{\text{ideal}}\), \( P \), \( V_m \), \( R \), and \( T \) are the molar volume, molar volume of the ideal gas, pressure, gas constant, and temperature, respectively.

CFD simulation for the air-ingress study The helium remains ideal gas law in a wide range of pressure and temperature. Since FLUENT uses ideal gas law to calculate the density for compressible condition, the compressible factor need to be remains to unit.

Figure 3.1 and 2 show the compressibility factor of helium and air with various pressure and temperature. The compressibility factor of helium is less than 3% deviation at 7 MPa and 300 K and the compressibility factor of air is within the 1% error. The compressibility factor tends to approach to 1 as the gas pressure decreases and the temperature increases. Therefore, the 3% error would be the maximum for the density calculation using ideal gas law in FLUENT.
Figure 3.1. Compressibility factor of helium [24]

Figure 3.2. Compressibility factor of air [25]
The properties of thermal conductivity, viscosity, and the diffusion coefficient for the gas species are calculated using kinetic-theory. The kinetic-theory in FLUENT uses the empirically determined Lennard-Jones potential energy function that has showed good agreement when the ideal gas law is applicable.

3.3. Depressurization

Once the accident is initiated, the coolant (helium) of the reactor is discharged into the containment. The reactor pressure vessel is located in a cavity which is filled with air during normal operation; the discharged helium will be mixed with air. The mixture concentration and temperature is the key parameter for the next scenario, therefore, it should be calculated precisely. However, CFD calculations for the highly-pressurized case, coupled with the energy equation converge slowly and require large computational time. Therefore, previous studies [32, 33] assumed the depressurization terminated when the pressure equalizes between the confinement and the vessel and started the analysis from the termination of blowdown. The species concentration changes by the depressurization were not precisely calculated.

3.3.1 Benchmark study of depressurization

The gas discharge model by Dutton and Coverdill [34] is used for the depressurization process benchmark study. In that study, the experiments are performed with a combination of two pressure vessel sizes and four nozzle diameters. The facility is
initially filled with air at 1 MPa and 297 K. The depressurization time for different vessel size and nozzle diameter combinations are summarized in Table 3.1.

![Figure 3.3. Depressurization test facility [34]](image)

**Table 3.1. Experiment dimensions and depressurization time [34]**

<table>
<thead>
<tr>
<th>Vessel Volume (cm$^3$)</th>
<th>4,920</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle diameter (mm)</td>
<td>1.63</td>
</tr>
<tr>
<td>Discharge time (s)</td>
<td>22</td>
</tr>
<tr>
<td>Vessel Volume (cm$^3$)</td>
<td>29,100</td>
</tr>
<tr>
<td>Nozzle diameter (mm)</td>
<td>1.63</td>
</tr>
<tr>
<td>Discharge time (s)</td>
<td>132</td>
</tr>
</tbody>
</table>

The experimental results show the depressurization times are between two theoretically calculated times based on isothermal and isentropic processes. The depressurization process that follows the isothermal process is slower. This allows sufficient time for heat to be transferred from the vessel wall to the gas in order to
maintain the gas temperature. On the other hand, the fast depressurization follows an isentropic process. The two limiting cases, i.e., a large volume vessel with the smallest nozzle and a small volume vessel with the largest nozzle, followed the isothermal and the isentropic depressurization process, respectively.

The nondimensional pressure solutions for the isentropic and isothermal cases are expressed in Eqs. (3.2) and (3.3), respectively. The fluid pressure inside the vessel \( P \) is scaled with respect to the initial fluid pressure inside the vessel \( P_i \). The quotient of these two pressures results in a nondimensional pressure expressed as \( P^+ = P/P_i \). In this study, the nondimensional time is defined as \( t^+ = t/t_{\text{char}} \) where \( t_{\text{char}} = V/A_i a_i \). The symbols \( \gamma, V, A_i, \) and \( a_i \) denote the heat capacity ratio, vessel free volume, nozzle throat area, and speed of sound of the gas at the initial temperature, respectively. The derivation of the equation is shown in Appendix A.

Choked isentropic solution:

\[
P^+ = \left[ 1 + \left( \frac{\gamma - 1}{2} \right) \left( \frac{\gamma + 1}{2} \right) \right]^{-\frac{-(\gamma+1)}{2(\gamma-1)}} t^+
\]

(3.2)

Choked isothermal solution:

\[
P^+ = \exp \left[ -\left( \frac{\gamma + 1}{2} \right)^{-\frac{-(\gamma+1)}{2(\gamma-1)}} t^+ \right]
\]

(3.3)
The geometry and meshes of the benchmark CFD model are generated by using ANSYS DesignModeler and ANSYS Meshing, respectively. An illustration of the geometry and a cut view of the polyhedral mesh structure are shown in Figure 3.4.

3.3.2 Determining CFD mesh structure and transient time step

A study is performed to investigate the effects of the mesh density and the transient time step. As summarized in Table 3.2, three sets of mesh are selected for the small vessel with a 1.63 mm nozzle diameter. The grid space refinement ratio, which is the average cell distance of the coarser cell to the current average cell distance, is set to 1.23. The minimum recommended by Roache for the Richardson extrapolation and GCI
index method is 1.1 [26]. The value of 1.1 is empirically obtained so that a higher ratio may give more accurate results due to an increase in the number of meshes. However, a higher ratio caused by an increase in the number of meshes increases the difficulty of performing a refinement study for this model. In addition, the increased number of meshes, decrease the average cell distance so that the transient time step might need to be decreased to achieve convergence. Therefore, to achieve a balance between the computational accuracy and the computational cost, 1.23 is selected.

<table>
<thead>
<tr>
<th></th>
<th>No. of cells</th>
<th>Ave. cell distance (mm)</th>
<th>Refinement ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>57,911</td>
<td>4.39</td>
<td></td>
</tr>
<tr>
<td>Medium</td>
<td>108,031</td>
<td>3.57</td>
<td>1.23</td>
</tr>
<tr>
<td>Fine</td>
<td>205,407</td>
<td>2.89</td>
<td>1.23</td>
</tr>
</tbody>
</table>

For the FLUENT model development, the small vessel with a 1.63 mm nozzle diameter is selected because the experimental results show its depressurization process is placed between the two extreme cases, the isentropic and the isothermal process.

A density-based solver is developed to solve high speed compressible flow while the pressure-based solver is developed for low-speed incompressible flows. Although a pressure-based solver can also solve the compressible flow problem, it is believed that a density-based solver will give more accurate solutions for a highly-pressurized vessel depressurization problem. However, it is difficult to achieve convergence with a density-based solver. A density-based solver solves all the governing equations simultaneously,
i.e., continuity, momentum, energy, and species transport. In addition, it is recommended for a density-based solver that the Courant number not exceed five while the pressure-based solver is a stable implicit solver. The Courant number is proportional to the transient time step and fluid velocity, and inversely proportional to grid space as shown in Eq. (3.4). Since the depressurization velocity of the gas is mostly constant at the choked condition, if the grid space decreases the transient time step needs to be reduced in order to maintain a similar Courant number. Therefore, a pressure-based solver is selected to solve the depressurization problem.

\[
Co = \frac{\Delta t \times u}{h}
\]  

(3.4)

where \( Co, h, \) and \( u \) denote the Courant number, grid space, velocity of the gas phase, respectively.

For the CFD simulations, the ideal gas law and a piecewise polynomial function are used to calculate the helium density and its specific heat, respectively. To calculate the other thermo-physical properties, kinetic theory is utilized. Standard \( k-\varepsilon \) model is used for viscous model. A \( 2^{nd} \) order upwind scheme is used for the convective term and a \( 1^{st} \) order scheme is used for the transient time step.
During the depressurization, choked flow is observed through the pipe break. The initial condition of the air in the vessel is 1 MPa and 297 K. The choked velocity can be calculated as,

$$c = \sqrt{c_\gamma R T}$$  

where $c$, $\gamma$, $R$, and $T$ denote the speed of sound, the ratio of specific heat, the specific gas constant, and absolute temperature, respectively. The maximum depressurization velocity or choked velocity at the pipe break is 345 m/s for air at 297 K. The time step for the grid refinement is set to 1 ms. The corresponding Courant number for the fine, medium, and coarse mesh are 78, 96 and 119, respectively. Unlike a density-based solver, a pressure-based solver gives a converged solution with high Courant numbers. Figure 3.5 shows the two limiting cases for three different types of mesh. The uncertainty from the fine and coarse mesh can be obtained by the GCI index as summarized in Table 3.3.

![Figure 3.5. Mesh refinement study with three different mesh sizes](image-url)
Table 3.3. GCI (%) values for depressurization process

<table>
<thead>
<tr>
<th>t⁺</th>
<th>GCI₁₂</th>
<th>GCI₂₃</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0.3</td>
<td>2.48</td>
<td>1.67</td>
</tr>
<tr>
<td>0.6</td>
<td>0.69</td>
<td>1.47</td>
</tr>
<tr>
<td>0.9</td>
<td>0.81</td>
<td>1.52</td>
</tr>
<tr>
<td>1.2</td>
<td>0.66</td>
<td>1.36</td>
</tr>
<tr>
<td>1.5</td>
<td>0.34</td>
<td>1.59</td>
</tr>
<tr>
<td>1.8</td>
<td>0.58</td>
<td>0.97</td>
</tr>
<tr>
<td>2.1</td>
<td>0.07</td>
<td>0.23</td>
</tr>
</tbody>
</table>

From the fine and medium mesh results, the uncertainty values are less than 3%. The uncertainties of the fine and medium mesh are relatively low. Since finer mesh requires more computational resources, coarser mesh would be beneficial in benchmark study if the uncertainty is reasonably low.

The uncertainty of the coarse mesh can be obtained by modifying Eq. (3.6) which is expressed with medium and coarse meshes as,

$$f_{exact} \approx \frac{r_{23}^p f_2 - f_1}{r_{23}^p - 1}$$

$$= f_2 + \frac{f_2 - f_3}{r_{23}^p - 1} = f_3 + \frac{r_{23}^p (f_2 - f_3)}{r_{23}^p - 1}$$

(3.6)

Then the relative error of the coarse mesh solution can be obtained as,
By applying the safety factor value as the GCI value, the uncertainty of the coarse mesh solution can be obtained. Figure 3.6 shows the coarse mesh solution with uncertainty bars.

By assuming the errors from the time step and the grid space are independent, temporal discretization error estimation can be performed with the coarse mesh. The time steps of the depressurization are set to 1, 0.5, and 0.25 ms. The time refinement

$$E_{3}^{coarse} = \frac{f_{extrapolated} - f_{3}}{f_{1}} \approx \frac{r_{23}^{p} f_{2} - f_{3}}{r_{23}^{p} - 1}$$

(3.7)
results are shown in Figure 3.7. The GCI values for this time refinement study are on the order of $10^{-5}$. The values are relatively small compared to the spatial discretization; therefore, the uncertainty from the time step can be ignored for these time steps.

![Figure 3.7. Time refinement study with three different time steps](image)

The coarse mesh structure gives a maximum error of 3.7% at the beginning and then decreases to less than 1% eventually. Therefore, the coarse mesh and 1 ms time step, which gives a maximum uncertainty of 3.7% for a small vessel, is selected for the benchmark study. In the generating the mesh structure for a large vessel, it maintains the same mesh structure by maintaining same cell distance and expands the size of the vessel. Therefore, further mesh refinement study is not performed for the large vessel. The number of meshed for large vessel is 177,476.
3.3.3 Depressurization results

Three cases are selected for CFD benchmarks: 1) the small vessel with the largest diameter nozzle, 2) the large vessel with the smallest diameter nozzle, and 3) the small vessel with the smallest diameter nozzle. Cases 1 and 2 are the two limiting cases which follow the isentropic and isothermal depressurization processes, respectively. At some point during its depressurization, case 3 follows both of the limiting cases. It initially follows the isentropic process then eventually starts to follow the isothermal process until the depressurization is completed.

The simulation is performed from the initial pressure (1 MPa) to the unchoked pressure (0.189 MPa). The CFD results in Figure 3.8 show that the depressurization of case 1 and case 2 closely follow the experimental results of the limiting cases given by Dutton and Coverdill [34]. Furthermore, the entire depressurization of case 3 occurs within the limits of the isothermal and isentropic case. The depressurization of case 1 is a faster process than the depressurization of case 2 and 3. For those cases where the depressurization occurs more quickly, a lesser amount of heat transfer from the wall occurs. Hence, in case 1, the depressurization closely follows the isentropic case since there is little time for significant heat transfer from the wall to occur. On the other hand, in case 2, the depressurization closely follows the isothermal case since there is sufficient time for heat transfer from the vessel wall to maintain the temperature of the gas in the
vessel constant. Therefore, the fast depressurization process follows the isentropic process, while the slow process follows the isothermal process.

![Depressurization results with various vessel and nozzle sizes](image)

Figure 3.8. Depressurization results with various vessel and nozzle sizes

From this study, it shows that the FLUENT pressure-based solver could solve the highly-pressurized tank depressurization problem with relatively low uncertainty. In addition, implicit transient formulation of a pressure-based solver generates a stable result even with large time steps resulting in large Courant number.

### 3.4. Density-driven flow

Density driven flow is established when there are density variations horizontally by gravitational force. The heavier current produces a higher pressure on the lighter
current; therefore, the heavier fluid propagation occurs through the bottom of the lighter gas. The density driven flow occurs in gases when there are density differences by temperature or by gas species mixture concentration difference.

Yih [27] and Benjamin [28] proposed the lock density driven flow model by gravitational force. When the density of the heavy current $\rho_2$ and the light current $\rho_1$ are flow as shown in Figure 3.9, the velocity can be given as,

$$ U = \sqrt{\frac{g(\rho_2 - \rho_1)H}{2(\rho_2 + \rho_1)}} $$  \hspace{1cm} (3.8)

where, $g$ and $H$ are acceleration due to gravity and pipe height.
For the Boussinesq case, the densities are close enough and the energies are conserved during the gravity-driven flow. The plume height is half of the pipe height.

Then it can be expressed as the Froude number by using reduced gravity $g' = g \frac{(\rho_2 - \rho_1)}{\rho_2}$

$$F_r = \frac{U}{\sqrt{g'H}} = \frac{1}{2}$$

(3.9)

However, the air-ingress density-driven case is the flow between high temperature helium and cold air, which is not a Boussinesq case. Lowe et al. [29] show the Boussinesq model prediction is not correct when the density ratio is 0.681. Therefore, the air-ingress accident case, in which the density ratio is less than 0.16, cannot apply the Eq. (3.9).

Lowe et al. [29] proposed the heavy current velocity model,

$$U_H = \sqrt{(1 - \gamma)gH \left[ \frac{1}{\gamma} \left( 2 - \frac{h_H}{H} \right) \left( 1 - \frac{h_H}{H} \right) \right]^{1/2}}$$

(3.10)

where $\gamma$ and $h$ are the density ratio and heavy current plume height, respectively. The theoretical height of the plume can be obtained from the solid line in Figure 3.10.
However, the equation over predicts the velocity of the high density current experimental data. As shown in Figure 3.11, the solid line which is a theoretical solution, deviates from the experimental results of Grobelbauer et al. [30] which is the open square box in the figure. Grobelbauer et al. [30] had high density ratio gas density-driven flow experiment. The experiments cover the density ratio from 0.904 to 0.046. The experiment also contains the air-helium experiment. CFD benchmark study for this model is performed with the experimental data of Grobelbauer et al. [30]
The experimental set-up is shown in Figure 3.12. The cross section of the channel is 0.3x0.3 m² and the left side of the gate is 3 m and right side of the gate is 0.8 m long. In the left side, there are seven hot-wires probes installed at the bottom of the channel. The probe only detects the arrival time of the density driven flow. The gas combinations used for the experiment are summarized in Table 3.4.
Table 3.4. Gas combination used for the experiments

<table>
<thead>
<tr>
<th>No</th>
<th>Heavy gas</th>
<th>Light gas</th>
<th>$\rho_{\text{heavy}}$</th>
<th>$\rho_{\text{light}}$</th>
<th>$\gamma$</th>
<th>Velocity (m/s)</th>
<th>Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>CO$_2$</td>
<td>Argon</td>
<td>1.77</td>
<td>1.60</td>
<td>0.904</td>
<td>0.21</td>
<td>74</td>
</tr>
<tr>
<td>2</td>
<td>Argon</td>
<td>Air</td>
<td>1.60</td>
<td>1.16</td>
<td>0.724</td>
<td>0.44</td>
<td>1,679</td>
</tr>
<tr>
<td>3</td>
<td>R-22</td>
<td>Argon</td>
<td>3.52</td>
<td>1.60</td>
<td>0.456</td>
<td>0.76</td>
<td>7,874</td>
</tr>
<tr>
<td>4</td>
<td>R-22</td>
<td>Air</td>
<td>3.52</td>
<td>1.16</td>
<td>0.330</td>
<td>0.88</td>
<td>14,857</td>
</tr>
<tr>
<td>5</td>
<td>Air</td>
<td>Helium</td>
<td>1.16</td>
<td>0.16</td>
<td>0.138</td>
<td>1.24</td>
<td>58,763</td>
</tr>
<tr>
<td>6</td>
<td>Argon</td>
<td>Helium</td>
<td>1.60</td>
<td>0.16</td>
<td>0.100</td>
<td>1.35</td>
<td>90,509</td>
</tr>
<tr>
<td>7</td>
<td>R-22</td>
<td>Helium</td>
<td>3.52</td>
<td>0.16</td>
<td>0.046</td>
<td>1.69</td>
<td>93,831</td>
</tr>
</tbody>
</table>

Figure 3.12. High density ratio density driven flow experimental set-up

The 2-D CFD model was developed as shown in Figure 3.13. To quantify the discretization error from the mesh structure, a mesh refinement study was performed with three set of meshes, 663, 2652, and 10608. The left side of the gas was filled with helium and the right side was filled with air at room temperature and atmospheric pressure.
Since the fluid height is not constant and depends on the density ratio, accumulated air concentration was selected to compare for each model.

The fluent specification and model used in this calculation are listed as follows:

- **Solver**
  - Solver : Pressure-Based
  - Time : Transient
  - Pressure Velocity coupling: PISO
  - Transient Formulation : 1st Order Implicit

- **Discretization**
  - Pressure : Standard
  - Momentum : 2nd order upwind
  - Species : 2nd order upwind
  - Energy : 2nd order upwind

- **Viscosity model**:
  - K-ε standard turbulence

- **Species transport model**
  - Mixture material: Mixturetemplate
  - 2 species: Air and Helium
  - Density : Ideal-gas
  - Specific heat : mixing-law
  - Thermal conductivity : Ideal-gas-mixing-law
  - Viscosity : Ideal-gas-mixing-law
  - Mass Diffusivity : Kinetic theory
Figure 3.14 shows the mesh refinement study results, which show how fast the heavy fluid flow to the left channel. As the mesh refined, the solution asymptotically converged. By applying the Richardson extrapolation and the GCI index, the extrapolated solution and uncertainty of the mesh structure can be obtained as summarized in Table 3.5. The uncertainty of the mesh structure ranges from 1 to 3 % of the results and Figure 3.15 shows how the fine mesh results are close to the extrapolated values with an uncertainty bar.
Table 3.5. Uncertainty quantification of the density driven flow

<table>
<thead>
<tr>
<th></th>
<th>Asymp</th>
<th>fine</th>
<th>medium</th>
<th>coarse</th>
<th>P</th>
<th>GCI12</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.04</td>
<td>8.448</td>
<td>8.382</td>
<td>8.171</td>
<td>7.279</td>
<td>2.079259</td>
<td>0.98%</td>
</tr>
<tr>
<td>1.84</td>
<td>8.096</td>
<td>7.948</td>
<td>7.647</td>
<td>6.732</td>
<td>1.601989</td>
<td>2.33%</td>
</tr>
<tr>
<td>1.64</td>
<td>7.309</td>
<td>7.198</td>
<td>6.938</td>
<td>6.067</td>
<td>1.741873</td>
<td>1.93%</td>
</tr>
<tr>
<td>1.44</td>
<td>6.364</td>
<td>6.321</td>
<td>6.158</td>
<td>5.375</td>
<td>2.261678</td>
<td>0.85%</td>
</tr>
<tr>
<td>1.24</td>
<td>5.520</td>
<td>5.416</td>
<td>5.220</td>
<td>4.652</td>
<td>1.533183</td>
<td>2.39%</td>
</tr>
<tr>
<td>1.04</td>
<td>4.527</td>
<td>4.484</td>
<td>4.360</td>
<td>3.876</td>
<td>1.96377</td>
<td>1.19%</td>
</tr>
<tr>
<td>0.84</td>
<td>3.596</td>
<td>3.543</td>
<td>3.433</td>
<td>3.092</td>
<td>1.627684</td>
<td>1.86%</td>
</tr>
<tr>
<td>0.64</td>
<td>2.644</td>
<td>2.613</td>
<td>2.535</td>
<td>2.268</td>
<td>1.78087</td>
<td>1.53%</td>
</tr>
<tr>
<td>0.44</td>
<td>1.707</td>
<td>1.688</td>
<td>1.635</td>
<td>1.435</td>
<td>1.919609</td>
<td>1.40%</td>
</tr>
<tr>
<td>0.24</td>
<td>0.782</td>
<td>0.763</td>
<td>0.730</td>
<td>0.645</td>
<td>1.402845</td>
<td>3.23%</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 3.15. Extrapolated value with uncertainty bar

In addition, to determine the turbulence model uncertainty, five different viscosity model in FLUENT were investigated (i.e., laminar, Reynolds stress model (RSM),...
standard k-ε, and k-ε realizable model were investigated. Figure 3.16 shows the mass fraction contour for the five different viscos physical model for the air-helium case. It shows different heavy current locations for the different viscosity model at the chosen time step. However, there is limited information how the front speed measured with the experimental apparatus of Grobelbauer et al. [30]. Therefore, the front speed of the CFD results displays the speed at the 10% of the heavy current arrival time and 90 % arrival time and range the velocity. The complete set of the contour results are shown in Appendix B and the front velocity and the relative error from the experimental results are summarized in Table 3.6. The values in parenthesis represent the negative values.

Figure 3.16. Mass fraction of helium-air simulation at 2.2 sec with 5 different turbulence models: 1. laminar, 2. Reynolds stress model (RSM), 3. standard k-ε, and 4. k-ε realizable
Table 3.6. CFD results of gas combinations

<table>
<thead>
<tr>
<th>No.</th>
<th>Viscosity model</th>
<th>Laminar</th>
<th>RSM</th>
<th>Standard</th>
<th>k-ε realizable</th>
<th>Exp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>CO2-Argon</td>
<td>0.22-0.21</td>
<td>0.18-0.16</td>
<td>0.16-0.14</td>
<td>0.18-0.17</td>
<td>0.21</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>(14.3)- (23.8)</td>
<td>(23.8)- (33.3)</td>
<td>(14.3)- (19.0)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Argon-Air</td>
<td>0.47-0.43</td>
<td>0.40-0.33</td>
<td>0.32-0.28</td>
<td>0.36-0.33</td>
<td>0.44</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>(9.09)- (25.0)</td>
<td>(27.27)- (36.36)</td>
<td>(18.18)- (25.0)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>R22-Argon</td>
<td>0.85-0.8</td>
<td>0.72-0.69</td>
<td>0.62-0.58</td>
<td>0.66-0.62</td>
<td>0.76</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>(-5.26)-(9.21)</td>
<td>(18.42)-(23.68)</td>
<td>(13.16)-(18.42)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>R22-Air</td>
<td>1.12-1.06</td>
<td>0.91-0.86</td>
<td>0.78-0.72</td>
<td>0.88-0.82</td>
<td>0.88</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td>(11.36)-(18.18)</td>
<td>(1.136)-(6.82)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>Air-Helium</td>
<td>1.36-1.3</td>
<td>1.26-1.18</td>
<td>1.08-1.00</td>
<td>1.19-1.09</td>
<td>1.24</td>
</tr>
<tr>
<td>10</td>
<td></td>
<td>(14.81)-(24.0)</td>
<td>(4.20)-(13.76)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>Argon-Helium</td>
<td>1.67-1.56</td>
<td>1.44-1.36</td>
<td>1.18-1.09</td>
<td>1.31-1.22</td>
<td>1.35</td>
</tr>
<tr>
<td>12</td>
<td></td>
<td>(12.16)-(23.67)</td>
<td>(2.96)-(9.63)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>R22-Helium</td>
<td>1.94-1.83</td>
<td>1.74-1.66</td>
<td>1.40-1.29</td>
<td>1.43-1.34</td>
<td>1.69</td>
</tr>
<tr>
<td>14</td>
<td></td>
<td>(17.16)-(23.67)</td>
<td>(15.38)-(20.71)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3.6 shows the flow velocity of the CO2-Argon and the Argon-Air within the range of laminar model and the R22-Argon is between the laminar and turbulent regime. The last of the simulations are within the turbulent regime. There have agreement with
the Reynolds number summarized in Table 3.4. The contour plots for the other gas combinations are shown in Appendix B.

Based on these results, RSM models would give best results for the density-driven air-ingress analysis. However, the RSM model for 3-D model would require excessive computational times and the unstable convergence make hard to get the simulation results, therefore, k-ε realizable model was selected for the air-ingress analysis.

3.5. Depressurization of prototypic design, GT-MHR

The normal operating pressure and the outlet temperature of the prototypical design GT-MHR are 7 MPa and 850°C. However, the OSU air-ingress test facility uses reduced pressure and temperature. The test vessel is designed to maintain the hydraulic similarity of the prototypic design (i.e., the GT-MHR), and can withstand reduced pressure and temperature up to 0.343 MPa and 973 K, respectively. Therefore, to decide the initial condition of the test facility, a volume-conserved CFD model was developed.

There are two zones separated distinctly for the heating, ventilating, and air conditioning (HVAC) system in the reactor building. As shown in Figure 1.1, the pressure vessel and the power conversion units are located underground. This part of the building is constructed as cylindrical silo and isolated from the rectangular part of the building above the ground. The wall structures, doors and any other barriers are designed
to separate the fluid flow from the silo portion and the rectangular part of the reactor building.

In the event of the large pipe break in the closed portion of the reactor building, the gases in the closed portion could move to any compartment through the entire building and released to atmosphere through relief valves. The containment concept allows the initial release from reactor building since the TRISO coated particle would hold the radioactive materials and contains the higher radioactive materials if the accident progresses over a long time.

Figure 3.17. Simplified containment CFD model

To study the depressurization effect, a simplified 3-D CFD calculation was performed. Figure 3.17 shows the simplified model. Instead of using the prototypic geometry, a regular hexahedron design with conserved volume was used. Since the fluid
freely flow through each compartment during depressurization process, the containment volume is combined. The vessel and containment volume are set to 265 and 25,000 m³, respectively, the same as the prototypic design.

During normal operation, the loads to the cross vessel are reactor operating pressure and temperature variation, which is same as the other pressure boundary, however, large load might be applied to the cross vessel in an abnormal condition, such as seismic events and building settlement. The cross vessel restrains the reactor vessel and power conversion unit, therefore, displacement of the reactor vessel and PCU would make the cross vessel a weak point in a pressure boundary and the cross sectional area of the break 3.7 m² was used in the analysis.

Figure 3.18. Cross vessel structure
The fluent specification and model used in this calculation are listed as follows:

- **Solver**
  - Solver : Pressure-Based
  - Time : Transient
  - Pressure Velocity coupling: PISO
  - Transient Formulation : 1st Order Implicit

- **Discretization**
  - Pressure : Standard
  - Momentum : 1st order upwind
  - Species : 1st order upwind
  - Energy : 1st order upwind

- **K-ε standard turbulence**

- **Species transport model**
  - Mixture material: Mixturetemplate
  - 2 species: Air and Helium
  - Density : Ideal-gas
  - Specific heat : mixing-law
  - Thermal conductivity : Ideal-gas-mixing-law
  - Viscosity : Ideal-gas-mixing-law
  - Mass Diffusivity : Kinetic theory

The simplified prototype model assumes that the depressurization of the guillotine break would be a fast process, so that the heat transfer could be ignored and the process would follow an isentropic process. As shown in section 3.2, the solution verification using the Richardson extrapolation would requires lots of computational resources, so the result of the CFD solution is verified by the 1-D analysis solution which showed good agreement with the depressurization benchmark study in the previous section.
Two CFD simulations are performed to check if the heat transfer from the vessel wall would have influence during the depressurization process. Both cases are initiated with the pressure vessel filled with 100% helium at 1023K and 7 MPa and the containment filled with 100% air at 300K and 0.1 MPa. The difference is one case used constant wall temperature so that heat fluxes exist from the internal structure to the fluid and the other case used zero heat flux from the internal structure.

![Depressurization Time vs Break Size](image)

Figure 3.19. Depressurization time depends on the break size

The design pressure of the OSU air-ingress test facility 0.3 MPa is higher than the critical pressure 0.206 MPa. Since the initial depressurization process choked flow, the depressurization time takes from the prototype initial pressure 7 MPa to the design pressure, 0.3 MPa, of OSU air-ingress test facility can be calculated by using the Eq. (3.2)
and (3.3). Figure 3.19 shows the expected depressurization time depends on the pressure boundary break size.

![Depressurization Curve](chart.png)

**Figure 3.20.** Comparison of depressurization using 3-D CFD and 1-D analytical solution from 7 to 0.3 MPa

For the break of the GT-MRH cross vessel guillotine break case, the depressurization is terminated in 200 ms which agrees with the CFD solution in Figure 3.20. The solid black line in Figure 3.20 is a 1-D solution, and solid triangle and open square are the CFD results without and with heat flux from the vessel wall. Both CFD cases follow the 1-D isentropic process. This verification study confirms the depressurization of the simplified CFD model for the air-ingress depressurization study.
without internal structure, gives correct results for the prediction of the air-helium mixture composition in the reactor containment building.

Once the guillotine break of the cross vessel is initiated, the coolant (helium) of the reactor is discharged into the containment. The reactor pressure vessel is located in a cavity which is filled with air during normal operation; the depressurized helium will be mixed with air. The mixture concentration and temperature is the key parameter for the subsequent scenario, therefore, it should be calculated precisely.

There are two reactor building design characteristics that could influence the air-ingress accident, reactor building ventilation and multiple compartments (i.e., confinement design and compartmentalization).

3.5.1. Confinement design

Different from current the commercial nuclear power plant, the VHTR reactor building uses a vented, low-pressure containment. The GT-MHR design report indicates that the reactor building is designed to vent whenever the internal pressure exceeds 1 psid [1]. Therefore, if reactor depressurization happens, the discharged coolant gas and air mixture would flow out to outside the containment. It makes the available air that could ingress into the reactor.
The amount of air in the reactor building and helium in the vessel are 1016 and 221 kmol during normal operation, respectively. When the depressurization is terminated, the amount of helium coolant discharged from the vessel is around 200 kmol. If the ventilation is considered, the same amount of helium and air mixture would be discharged to outside the containment. If the discharged gas is assumed 100% air, there is 20% less air that could ingress into the reactor.

To study the containment ventilation effect, a simplified 3-D CFD calculation was performed. Instead of using the prototypic geometry, a regular hexahedron design with conserved volume was used with the same mesh structure as in the previous section and located the vent at the containment as in Figure 3.21.

Figure 3.21. CFD model of volume conserved with ventilation at top right
Figure 3.22. Various ventilation locations

Since the vent location and size of the GT-MHR are not provided in the design report, a case study was performed with three different ventilation locations as shown in Figure 3.22. The vent in Figure 3.22.(a) faces to the reactor vessel breach with higher elevation, the vent in Figure 3.22.(b) faces to the breach and same elevation, and the vent in Figure 3.22.(c) is located at the same elevation as (a) without facing the breach.

Figure 3.23 shows the calculation result of the vent located at the top right of the containment. Initially, the reactor pressure vessel was filled with 100% helium at 1023 K and 7 MPa. The containment was filled with 100% air at 300 K and 0.1 MPa. The boundary condition of the vent was set to “pressure outlet” with 300 K and 0.1 MPa. Figure 3.23 (a) shows that the air and helium are initially separated in the containment and vessel. When the depressurization is initiated, the helium is discharged as shown in Figure 3.23 (b). The discharge continues to fill the pressures inside the vessel and the containment is equalized. Figure 3.23 (c) represents the point at which the pressures are equalized. At this time, 0.27 seconds after initiation of accident, 0.33% of air and 1.22% of helium were released into the atmosphere.
During the early stage of the depressurization, the discharged helium is not well mixed as shown in Figure 3.23 (b) - (d). Therefore, the discharged mixture concentration is determined by the mixture concentration at the vent location. As shown in Figure 3.24, cases (a) and (b) release more helium than air. At the beginning of the depressurization, discharged helium directly hit the wall; therefore, cases (a) and (b) have a higher increase at the beginning of the accident. On the other hand, case (c) does not discharge helium until the helium–air mixture reaches to the vent located at the top left of the containment.
Figure 3.24. Percentage of released gases to atmosphere

Figure 3.25 shows the changes of air mole fraction in the volume of the reactor building. It combines the reactor vessel and containment. It shows that the vent located at the bottom right gives a higher air mole fraction due to the initially discharged helium during the depressurization process. However, the air mole fraction change from the initial stage is less than 1% which is quite lower than 100% air or 100% helium discharge case. Even though the vent size would affect the air concentration change, the
current assumption, which is that the vent is the same as the double-ended guillotine break, is already big, therefore, the vent size and location would not affect much on the density change.

![Air mole fraction change due to the ventilation locations](image)

**Figure 3.25.** Air mole fraction change due to the ventilation locations

### 3.5.2. Compartment design effects

Figure 1.1 shows the reactor containment building that has multiple compartments. Because each compartment is not air sealed and the compartments are connected to each other, the fluid could flow to other compartments. A simplified CFD model is made by dividing it into two parts, cavity 1 and 2, as shown in Figure 3.26. The volume of cavity 1 and cavity 2 were set to 530 m³ and 24,470 m³ to conserve the containment volume. A double-ended guillotine break of the cross vessel was considered.
for this analysis. The discharged helium would fill the compartment in the vessel and power conversion unit (PCU) first. Therefore, the cavity volume was set to the combined volume of the cavity volume of PCU and reactor vessel.

Figure 3.26. GT-MHR CFD model (compartmentalized)

Figure 3.27 shows the simulation results achieved thus far. Initially, the cavity was filled with 100% air at 0.1 MPa and 300 K, and the vessel was filled with 100% air at 7 MPa and 300 K. Once the double-ended guillotine break of the cross vessel is initiated, the discharged helium would fill the compartment where the breach is located and move the air-helium mixture to other compartments during the depressurization process. The contour color changes in Figure 3.27 show how the air in cavity 1 moves to the other compartment. Figure 3.28 (left) shows the air concentration in cavity 1 decreased to less than 10% of the initial concentration when the depressurization is terminated.
Different from the previous single-containment design analysis, the depressurization took longer for the compartment design, since the discharged helium from the vessel mixed at the cavity 1 and then move to cavity 2. Cavity 1 in the compartment design behaves as a buffer region to dilute the air concentration and decrease the depressurization time. Figure 3.28 shows the pressure changes (top) and air concentration changes (bottom) for the compartment design. Because the depressurized helium fills cavity 1 first, and the pressure difference between cavity 1 and the pressure vessel is low, the depressurization process is slower than the single-containment design. Even though there is a time difference from one containment design to the other, both depressurization time scales are less than one second. However, the compartment design results show that the amount of air left in cavity 1 is less than 10% of the initial amount. Even though the air would move back to cavity 1 because the break location is at the low ground, the compartment design reduces the amount of air that could ingress to the reactor vessel. The amount air flow back to the cavity 1 will be discussed in Chapter 7.
Figure 3.27. Mass fraction of air (compartments)
Figure 3.28. Pressure changes of each compartment (top) and air concentration changes in cavity 1 (bottom)
3.6. Discussion

For the physical model for the analysis of the density driven flow and the depressurization flow stages were investigated. Based on the physical model of the density driven flow, k- ε realizable gave a 5 % difference from the experimental results. In addition, pressure-based solver for the depressurization analysis gave a reasonably-accurate solution. By using the pressure-based solver instead of the density-based solver, the computational resource could be saved without compromising the accuracy of the results. For the analysis of the containment building effect gave the initial condition for the OSU air-ingress test facility. The localized natural circulation will be discussed in Chapter 4 by utilizing the OSU design.
Chapter 4: CFD as a design tool for the design of the OSU scaled down air-ingress test facility

Realizing the need of having experimental data to better understand the air-ingress phenomena, the OSU has developed an air-ingress test facility. The test facility is a 1/8th geometric scaled-down facility of the GT-MHR hot duct and hot plenum as shown in Figure 4.1. To gain insights on the progress of the air-ingress accidents, 3-D CFD simulations are performed.

Figure 4.1. Air-ingress to the hot plenum (blue color-arrow) and enlarged view of hot duct-hot plenum system
4.1. Small-scale experiments using acrylic set-up for density driven flow

Small-scale experiments were performed prior to designing and constructing the actual test facility. A 1/30th reduced length scale model of GT-MHR demonstration setup was constructed. Figure 4.2 shows a schematic of the setup. This experiment is the first step to obtain a better understanding of flow patterns that could occur during a cross vessel break accident. The reactor core was highly simplified. This set-up can test the effects of different types of cross vessel break and creep by changing the flange that houses the break plate or slit.

The set-up has a gas inlet at the top to inject gas to pressurize the facility if needed. The dimensions of the scaled-down set-up are listed in Table 4.1.

<table>
<thead>
<tr>
<th></th>
<th>Prototype (m)</th>
<th>1/30 length scaled (in)</th>
<th>Available (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vessel ID</td>
<td>7.2</td>
<td>9.449</td>
<td>9.5</td>
</tr>
<tr>
<td>Barrel OD</td>
<td>6.31</td>
<td>8.281</td>
<td>8.0</td>
</tr>
<tr>
<td>Cold Duct ID</td>
<td>2.29</td>
<td>3.005</td>
<td>3.0</td>
</tr>
<tr>
<td>Hot Duct ID</td>
<td>1.43</td>
<td>1.877</td>
<td>1.75</td>
</tr>
<tr>
<td>Vessel Height</td>
<td>23.7</td>
<td>38.88</td>
<td>38.88</td>
</tr>
<tr>
<td>Core Height + Lower Plenum</td>
<td>13.4</td>
<td>17.56</td>
<td>17.56</td>
</tr>
</tbody>
</table>

For the better understanding of the density-driven flow and the feasibility test of applying a Particle Imaging Velocimetry (PIV) system to the future OSU high-temperature air-ingress test facility, air-helium experiments were performed for flow
visualization using a Planar Laser-Induced Fluorescence (PLIF) or PIV system. In the experiments, helium gas with seeding particles was used to fill the experimental apparatus. The seeding particles (vegetable oil) were generated with an aerosol generator with a mean size of about 1 micrometer.

The drag coefficient on spherical particle with small Reynolds number in a continuous viscous flow the drag coefficient can be expressed as,
Then the frictional force $F_d$ is,

$$ F_d = C_D \frac{1}{2} \rho_g V^2 A = 6 \pi \mu r V $$

(4.2)

The density of the olive oil aerosol particle is 924 kg/m$^3$. It is much heavier than helium or air density, 1.16 and 0.16 kg/m$^3$ at room temperature and atmospheric pressure. Then, the buoyancy force can be ignored.

$$ F_g = (\rho_p - \rho_g) \frac{4}{3} \pi r^3 g \approx \rho_p \frac{4}{3} \pi r^3 g $$

(4.3)

Where, $\mu, \rho_p, \rho_g, V, A, r, g$ are viscosity of gas, density of particle, density of gas, velocity of particle, projected area of particle, radius or particle, and acceleration of gravity, respectively.

When the forces $F_d$ and $F_g$ are balanced, the terminal velocity can be calculated as,

$$ V_T = \frac{2 \rho_p r^2 g}{9 \mu} $$

(4.4)
The terminal velocity of the vegetable aerosol is $9.57 \times 10^{-8}$ m/s. The vegetable oil could be used as a tracer material since the settlement of the vegetable oil does not influence the helium gas velocity measurement.

Figure 4.3. Test facility components
Figure 4.4 shows the results of the particle measurement with the PIV system. In the figure, the test set-up regions appeared red and could not measure the aerosol particles effectively in the cross pipe regions. This is due to the reflections of the laser from the curvature and transparency of the acrylic pipes. From this measurement, air-ingress velocity was not measured with the acrylic pipe. However, particles are measured at the outside pipe and low curvature region. These results give confidence that the PIV system could be applied to the OSU high-temperature air-ingress test facility, which uses plat quartz windows and metallic pipes. Therefore, instead of using particle measurements to measure density-driven flow, simple shadowography was used for the velocity measurement for the better understanding of density-driven flow.
Figure 4.5 shows the shadowgraph at around the pipe opening. Even though this experiment cannot show the complete air-ingress phenomena, it can show idea how the air intrudes into the vessel. Since the vegetable oil aerosol particle was injected into the vessel where the helium was filled, the shadow graph show the opaque region is the flow which helium contained and the clear pars are the region which air intrudes. Figure 4.5 shows the heavy air flew through the bottom of the pipe and the light gas flow out through the top of the break.

During the measurement, a vortex was measured as shown in the red circle in Figure 4.5. The displacement of this vortex in 3 milliseconds are 1.3 mm as shown in Figure 4.6. It represents the flow velocity at that location is 0.433 m/s.
To investigate how FLUENT could simulate this experiment, CFD model was developed as shown in Figure 4.7 (left) and the velocity vector(right) where the vortex displacement was measured. The CFD simulation was performed initially outside the vessel (red region) and the vessel (blue region) was filled with 100% air and helium,
respectively. The velocity vector as shown in the Figure 4.7 (right) indicates the velocity is between 0.36 and 0.48, which correspond to the experiment result. Since the acrylic set-up experiment was the experiment to obtain the idea for the density driven flow through the GT-MHR cross vessel, mesh refinement was not performed to minimize the uncertainty from the mesh structure. However, this result gives the confidence how to utilize the CFD result to understand the air-ingress accident. Different from the experiment, CFD simulation could show the information anywhere in the computational domain. Therefore, considerable understanding of the air-ingress phenomena can be obtained by performing the CFD simulation. It helps to design the experimental facility and determine the measurement location to obtain the optimize results to understand the air-ingress phenomena.

![CFD model of acrylic set-up and velocity vector](image)

Figure 4.7. CFD model of acrylic set-up (left) and the velocity vector at the location where the experiment was performed.
4.2. CFD model of the OSU air-ingress test facility

Figure 4.8 shows a cut-view of the reactor vessel with the air-ingress path to the hot plenum. The volume of the pressure vessel and the containment are 0.083 and 0.5 m$^3$, respectively. The FLUENT options and settings used for this model are summarized as follows:

- **Solver**
  - Pressure-Based
  - Double precision
  - Transient: 1$^{st}$ order implicit

- **Discretization**
  - Gradient: Least squares cell based
  - Pressure-Velocity Coupling: PISO
  - Density: 2$^{nd}$ order upwind
  - Momentum: 2$^{nd}$ order upwind
  - Turbulent Kinetic Energy: 2$^{nd}$ order upwind
  - Turbulent Dissipation Rate: 2$^{nd}$ order upwind
  - Species: 2$^{nd}$ order upwind
  - Energy: 2$^{nd}$ order upwind

- **Viscous Model**
  - k-ε turbulence

- **Species transport model**
  - Mixture material: Mixture template
  - Two species: Air and Helium

- **Species transport model**
  - Density: ideal-gas
  - Heat Capacity: mixing law
  - Thermal Conductivity: ideal-gas-mixing law
  - Viscosity: ideal gas mixing law
Three sets of mesh are used for the mesh refinement study and are summarized in Table 4.2. This study investigates two different phenomena: depressurization and density-driven air-ingress flow. The gas flow velocity during the depressurization stage is three orders of magnitude faster than the other stages of the air-ingress accident. Therefore, two separate simulations are performed to determine the mesh size and the transient time step.

Table 4.2. FLUENT model mesh information

<table>
<thead>
<tr>
<th></th>
<th>No. of cells</th>
<th>Ave. cell distance (mm)</th>
<th>Refinement ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine</td>
<td>423,984</td>
<td>11.12</td>
<td></td>
</tr>
<tr>
<td>Medium</td>
<td>154,756</td>
<td>14.14</td>
<td>1.27</td>
</tr>
<tr>
<td>Coarse</td>
<td>75,546</td>
<td>18.27</td>
<td>1.29</td>
</tr>
</tbody>
</table>
In the pressurized case, the test vessel is filled with helium to a pressure of 0.3 MPa and at a temperature of 1023 K, and the containment with air at 0.1 MPa and 300 K. In the non-pressurized case, the vessel is filled with helium to a pressure of 0.1 MPa and at a temperature of 1023 K, and the conditions in the containment are identical to those in the pressurized case.

The maximum depressurization velocity or choked velocity at the pipe break is 1,724 m/s for helium gas at 1,023 K. Using the same time step (750 μs) from the analysis performed in Sec. 3-2, the depressurization analysis on the scaled-down test facility results in a Courant number of 116. Consequently, the simulation does not converge to a solution. It might be due to the complicated geometry and the high speed of helium depressurization. The simulation converges to a solution when the time step is 50 μs. Therefore, 50, 25, and 12.5 μs are selected for a temporal refinement study.

Table 4.3. GCI (%) values for OSU scale-down facility depressurization process

<table>
<thead>
<tr>
<th>Time (ms)</th>
<th>Spatial</th>
<th>Temporal</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$GCI_{12}$</td>
<td>$GCI_{23}$</td>
</tr>
<tr>
<td>0</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0.1</td>
<td>12.2</td>
<td>13.8</td>
</tr>
<tr>
<td>0.2</td>
<td>9.3</td>
<td>13.9</td>
</tr>
<tr>
<td>0.5</td>
<td>3.8</td>
<td>6.4</td>
</tr>
<tr>
<td>1</td>
<td>0.4</td>
<td>1.6</td>
</tr>
<tr>
<td>1.5</td>
<td>0.01</td>
<td>0.2</td>
</tr>
<tr>
<td>2</td>
<td>0.1</td>
<td>0.7</td>
</tr>
<tr>
<td>2.5</td>
<td>0.7</td>
<td>1.7</td>
</tr>
<tr>
<td>3</td>
<td>0.5</td>
<td>1.4</td>
</tr>
<tr>
<td>4</td>
<td>0.01</td>
<td>0.07</td>
</tr>
<tr>
<td>5</td>
<td>0.08</td>
<td>0.2</td>
</tr>
</tbody>
</table>
Table 4.3 summarizes the GCI values for the depressurization process. Unlike the benchmark study, the temporal discretization error is on the same order as the spatial discretization error. Therefore, the temporal discretization error cannot be ignored in the overall uncertainty.

![Temperature as a function of pressure](image)

**Figure 4.9. Temperature as a function of pressure**

The depressurization of a cross vessel guillotine break terminates within 100 ms. It is a fast process and follows an isentropic process. Figure 4.9 compares a CFD simulation utilizing a fine mesh and time step with the analytical solution, Eq. (3.2), the depressurization and temperature relations follow an isentropic process. The maximum relative difference between the theoretical results and CFD results are maintained at less than 0.2 %. This result shows the stability of the FLUENT calculation for the pressurized
case. In addition, it shows that the fine mesh structure is sufficient for this study, since it closely matches the analytical solution.

In the non-pressurized case, the initial conditions are the same except the vessel pressure is the same as with the containment pressure. The density-driven flow speed is expected to be three orders of magnitude slower than the depressurization speed. Initially, 50 ms is attempted and convergence is not achieved. When 25 ms is the time step, convergence is achieved with the prescribed initial conditions. Therefore, 25, 12.5, and 6.25 ms are used for the density-driven flow. Table 4.4 summarizes the GCI values and Figure 4.10 shows the amount of air ingressed into the pressure vessel as a function of time for the density-driven flow. It shows that the spatial discretization errors are 7 % at the beginning of the air-ingress and are reduced to 1 % after 2 seconds.

Based on the discretization error analysis, by using the fine mesh for the geometrical structure, and 50 µs and 25 ms for depressurization and density-driven flow transient time steps, respectively would give 12.9 and 6.9 % uncertainty initially for depressurization and density-driven flow. It decreases to less than 1% at the end of the time step in this study.
Table 4.4. Solution verification results for density driven flow

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>$GCI_{12}$</th>
<th>$GCI_{23}$</th>
<th>$GCI_{12}$</th>
<th>$GCI_{23}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0.20</td>
<td>6.9</td>
<td>21.5</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>0.40</td>
<td>4.6</td>
<td>21.0</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>0.60</td>
<td>2.4</td>
<td>16.4</td>
<td>0.9</td>
<td>1.4</td>
</tr>
<tr>
<td>0.80</td>
<td>4.1</td>
<td>13.2</td>
<td>0.7</td>
<td>1.3</td>
</tr>
<tr>
<td>1.00</td>
<td>3.7</td>
<td>15.7</td>
<td>0.5</td>
<td>1.0</td>
</tr>
<tr>
<td>1.25</td>
<td>4.4</td>
<td>17.4</td>
<td>0.7</td>
<td>1.1</td>
</tr>
<tr>
<td>1.50</td>
<td>3.6</td>
<td>19.2</td>
<td>0.4</td>
<td>0.8</td>
</tr>
<tr>
<td>1.75</td>
<td>1.9</td>
<td>17.0</td>
<td>0.4</td>
<td>0.8</td>
</tr>
<tr>
<td>2.00</td>
<td>1.3</td>
<td>12.0</td>
<td>0.5</td>
<td>0.7</td>
</tr>
</tbody>
</table>

Figure 4.10. Amount of air in the vessel
4.3. Depressurization and flow oscillation

In the event of a pipe break, the pressurized hot helium coolant from the reactor discharges into the containment. This depressurization process continues until the pressure in the vessel is equalized with the containment pressure. After the depressurization, for a short time, rapid oscillations occur at the break. Martineau [35] show flow oscillations of the air-helium mixture in the reactor cavity into the reactor vessel before the establishment of air-ingress flow into the vessel by either diffusion or density-driven stratified flow. They investigate the flow oscillations at the end of the depressurization process with 1-D and 2-D shock tube simulations using a finite element code [35]. However, the consequences of the flow oscillations for the next stage of the air-ingress accident are not presented. Since the density driven air-ingress flow is established near the end of the depressurization process by the density difference between the helium and the relatively cool air-helium mixture, the mixing of gas species by flow oscillations could have an effect on the initiation and progress of the density driven air-ingress.

Previous studies [6,32,33] investigated the air-ingress accident from the termination of the depressurization. The simulation initiated the air-ingress from the equalized pressure in the vessel and containment. The main focus was to identify the air-ingress by the molecular diffusion and density-driven stratified flow. Therefore, the possible flow oscillation was not investigated that shown in reference [35]. However,
OSU test facility will initiate the test with the vessel slightly pressurized. Therefore, the flow reversal phenomenon could be validated experimentally. Before the experiment, the preliminary CFD simulation was performed. The two different simulation conditions are summarized in Table 4.5.

Table 4.5. Initial conditions of Figure 4.11 CFD simulations

<table>
<thead>
<tr>
<th></th>
<th>Case (a)</th>
<th>Case (b)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Vessel</td>
<td>Containment</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>300</td>
<td>300</td>
</tr>
<tr>
<td>Pressure (psig)</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Species concentration</td>
<td>100% He</td>
<td>100% Air</td>
</tr>
</tbody>
</table>

The only difference in the simulations is the initial pressure in the vessel to check the pressure effect. When initiate the accident, case (a) started air-ingress from the beginning of the accident. However, there is depressurization stage for case (b). Case (a) represents the previous assumption where the depressurization was terminated when the pressure of vessel and containment was equalized with 100% air in the containment and 100% helium in the vessel. The first contour in Figure 4.11 case (a) shows the initial condition when the pressure is equalized. The second, third, and fourth contours in Figure 4.11 case (a) show how the air-helium mixture flows into the reactor. As in the previous studies, after the depressurization, density-driven stratified flow is observed. Air in the containment enters into the vessel through the bottom part.

However, case (b) that initially pressurized vessel shows a different phenomenon. The first contour in Figure 4.11 (b) is when the pressure of the vessel and containment
are equalized. Different from the case (a), it shows the different species composition at the breach due to the discharged helium. In addition, as it progresses, a sudden change of mass fraction is observed. As shown in the second and third contour plots in Figure 4.11 case (b), the gas mixture is not just through bottom of the vessel before starting the density-driven flow as shown in the fourth contour in case (b). It can be distinguished by see the change of the color. Because the driving force of the density-driven flow is induced by the density differences between the flow interfaces. The assumption of case (a) would not appropriate for the air-ingress analysis. Therefore, more investigation is required to estimate the gas mixture composition at the end of blowdown and to investigate the impact of this flow reversal to the next stage.

Figure 4.12 shows the flow the velocity measurement at the center of the plane of the hot duct break surface. This graph shows that there is flow oscillation and it makes the difference in Figure 4.11 cases (a) and (b). As Martineau and Berry [35] demonstrated, the flow oscillations are observed near the end of the depressurization process. This flow oscillation phenomenon is analyzed using a simple 1-D model as shown in Figure 4.13 to explain the flow oscillation.
1. Starting point of air entering into vessel

2. After 10 ms

3. After 200 ms

4. After 1000 ms

Case (a)                   Case (b)

Figure 4.11. Mass fraction of air (a) without depressurization process and (b) with depressurization process
Due to the blowdown process being a very fast process, it can be considered as an isentropic process by ignoring heat transfer from component and energy loss. By
combining Newton’s second law of motion and Hooke’s law, the oscillation frequency can be predicted as follows,

\[
\frac{\Delta P}{P_0} = -\gamma \frac{\Delta V}{V_0} = -\gamma \frac{A\Delta x}{V_0} \tag{4.5}
\]

\[
\frac{d^2x}{dt^2} = \frac{F}{m} = \frac{\Delta PA}{\rho AL} = -\frac{\gamma AP_0}{\rho V_0 L} \Delta x \tag{4.6}
\]

\[
F = m \frac{d^2x}{dt^2} = -kx, \quad f = \frac{1}{2\pi} \sqrt{\frac{k}{m}} \tag{4.7}
\]

\(P_0\) and \(\Delta P\) are the pressure in the vessel when it equalizes with the containment pressure, and the pressure change due to the momentum of the discharged gas, respectively. \(V_0\) is the volume of the helium when the pressure equilibrium is achieved, and \(\Delta V\) is the volume change of the helium due to its pressure change. The \(\Delta V\) can be expressed as \(A\Delta x\) with break area \(A\) and moving distance \(\Delta x\). \(\gamma\), \(\rho\), \(L\), and \(c\) are the ratio of specific heat, species density, hot duct length, and speed of sound, respectively. By combining Newton’s second law of motion and Hooke’s law, the flow oscillation frequency can be predicted as,

\[
f(\text{Hz}) = \frac{1}{2\pi} \sqrt{\frac{\gamma AP_0}{\rho V_0 L}} = \frac{c}{2\pi} \sqrt{\frac{A}{V_0 L}}. \tag{4.8}
\]
Since the gas composition during the flow oscillation is difficult to estimate for the frequency calculation, the frequencies of the two limiting cases, 100% helium and 100% air, are calculated. Eq. (4.8) gives 78.7 and 205.6 Hz for 100% air and 100% helium, respectively. These two frequencies will provide a lower and upper limit of the expected range of oscillation frequencies. The frequency of the oscillation from the CFD result is around 120 Hz. This frequency falls within the calculated range; however, this flow oscillation cannot be measured and validated from the OSU experimental facility. Therefore, instead of investigating the flow oscillation, the details of the accumulated effects by the flow oscillation will be discussed in the next section.

4.4. The proceeding stage of the depressurization

Figure 4.11 case (b) shows the air concentration in the containment is diluted by the depressurized helium. In addition, due to the flow reversal inside the vessel, the helium also mixed with the outside air-helium mixture. In this section, the overall effect of the air-helium mixture during the depressurization process and the following flow oscillation to the subsequent air-ingress stage is investigated.

4.4.1. Heat transfer from the reactor vessel internal structures

During the depressurization, the gas inside the pressure vessel will have an adiabatic expansion and decrease in temperature. The average helium temperature of the prototypic vessel at the end of the depressurization is 220 K; however, the internal helium
temperature recovers to the internal structure temperature within a few seconds [36]. To better understand the temperature recovery after the depressurization, a CFD calculation is performed. Initially, the vessel and the containment are filled with 100% helium and 100% air at atmospheric pressure, respectively. The temperature of the internal vessel structure is set to 1023 K, and the helium temperature in the vessel and air in the containment are both set to 300 K. Arcilesi et al. [36] show the temperature of the gas in the vessel after blowdown is 220 K. However, the temperature recovery time will not change significantly if the final depressurization temperature changes from 220 to 300 K. Filling the vessel and containment at room temperature would be beneficial to generate experimental data for the validation work. Therefore, the air and helium temperature are initially set to 300 K.

Figure 4.14 shows the air mass fraction, temperature and density contour over time. Once the simulations are initiated, the density-driven flow can be observed in Figure 4.14 (a) at 0.2 sec. At this time, the temperature has not reached thermal equilibrium with the internal structure and the density difference is mainly due to the composition difference of air and helium. As the air-ingress progresses, air fills the vessel and the temperature inside the vessel reaches an equilibrium state. Figure 4.14 (b) shows the gas almost reaches a steady temperature profile at 5 sec after the initiation.
Figure 4.14. a) air mass fraction, b) temperature, c) density contour at 0.2, 2, 5, and 8 sec from top to bottom

Figure 4.15 and Figure 4.16 show the heat transfer coefficient of the support rod’s surface at 0.2 and 5 sec, respectively. As Arcilesi et al. [37] presented in the scaling analysis of the OSU air-ingress test facility, the natural convection heat transfer coefficient from the support rod to the air-helium gas mixture is about 5-15 W/(m²-K) depending on the far-field temperature and the air mole fraction. The FLUENT result shows that when the vessel is filled with cold helium the maximum heat transfer coefficient is about 16 W/(m²-K) and average about 10 W/(m²-K) at 0.2 sec as shown in
Figure 4.15. In Figure 4.16, most of the heat transfer is occurring when the air-ingress flow established and the air plume flows at the bottom of the rod which is at 5 sec. As shown in Figure 4.14 (b), the cold air plume flows along the bottom of the vessel and a hot gas mixture occupies the top of the vessel at 5 sec. Therefore, most of the heat transfer takes place at the bottom of the vessel. The heat transfer characteristics of the simulations are consistent with the calculations based on experimental correlations.

Figure 4.15. Heat transfer coefficient at the surface of graphite rods along the height at 0.2 sec
As shown in Figure 4.14 (c), the gas mixture density inside the vessel remains low due to the increasing temperature of the air-helium mixture even after the initial air-ingress. Since the higher temperature gas has higher specific volume, the amount of air would be lower depends on the internal gas mixture temperature. Therefore, additional simulations are performed to see how much air will be accumulated as a function of the internal structure temperature. It uses the same initial conditions as the previous simulation except the internal structure temperature with 300, 500, 700, and 900 K. As shown in Figure 4.17, the amount of air accumulated in the vessel decreases as the vessel internal structure temperature increases.
Figure 4.17. Accumulated air over the vessel internal structure temperature

Figure 4.18 shows the gas mixture composition that flows along the hot duct height at 5 sec after the initiation of the simulation. Figure 4.18 (a) shows the air mole fraction is higher at the bottom and gradually decreases toward the upper part of the duct. Figure 4.18 (b) shows the flow velocity along the hot duct height. The positive direction is into the reactor vessel. By multiplying these two values, Figure 4.18 (c) shows how much air enters and leaves the vessel. The heavier air flows through the bottom of the duct and the lighter air-helium mixture flows through the upper part of the duct. While nearly all pure air enters into the reactor through the bottom of the hot duct regardless of the vessel internal structure temperature, the air mole fraction along the height of the duct decreases less rapidly the higher the vessel internal structure temperature. The inlet velocity profile through the bottom of the hot duct is nearly identical for all internal
structure temperatures. However, the magnitude of the outlet velocity varies for different internal structure temperatures. For higher vessel internal structure temperatures, the magnitude of the outlet velocity is greater for all axial positions of the hot duct. This shows why the accumulated air in the vessel increases less rapidly with respect to time for higher vessel internal structure temperatures.

Figure 4.18. Gas mixture compositions at the center of the hot duct at t = 5 s
4.4.2. Consequences of flow oscillation on density-driven air-ingress

As shown in Figure 4.14, the internal structure temperature gives a constant density difference even after air fills the reactor vessel which is the driving force for the next stage of the air-ingress accident. The Figure 4.18 (c) shows the ingress amount air does not change in spite of the internal structure temperature change. The increased temperature in the vessel structure increase the density difference that develop the density-driven air-ingress flow at the break, however, the CFD results show the flow maintains constant. Instead, increased gas mixture temperature make more air contains in the discharged gas mixture. Therefore, in the analysis of the depressurization oscillation effect to the subsequent air-ingress stage, isothermal simulations are performed with the internal structure temperature set to room temperature to leave the temperature effect.

The discharged helium during the depressurization decreases the gas mixture density in the containment, and the gas mixing at the end of the depressurization by flow oscillation increases the gas mixture density in the pressure vessel. These two effects may influence the next stage of the air-ingress accident since the density differences are the driving forces for density-driven flow. To investigate the flow oscillation phenomenon, more simulations are performed which are summarized in Table 4.6. In cases 1-3, the initial pressure vessel pressure is varied to see the pressure effect. In cases 4-10, the containment concentration is varied to see the effect of the containment mixture
density on the density-driven stratified flow. In this simulation, the initial temperature of air and helium are set at 300 K.

Table 4.6. Initial conditions of CFD simulations for various species concentration and pressure

<table>
<thead>
<tr>
<th>Case no.</th>
<th>Pressure (MPa)</th>
<th>Species concentration (Air mole fraction)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Vessel</td>
<td>Containment</td>
</tr>
<tr>
<td>1</td>
<td>1.1</td>
<td>0.1</td>
</tr>
<tr>
<td>2</td>
<td>0.6</td>
<td>0.1</td>
</tr>
<tr>
<td>3</td>
<td>0.2</td>
<td>0.1</td>
</tr>
<tr>
<td>4</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>5</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>6</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>7</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>8</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>9</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>10</td>
<td>0.1</td>
<td>0.1</td>
</tr>
</tbody>
</table>

As shown in Figure 4.11 case (b), the pressurized vessel represents three stages: depressurization, flow oscillations and density-driven air-ingress flow. Table 4.7 shows the air-helium composition at each stage. The termination of the depressurization is when the containment and pressure vessel pressure equalize the first time. The initiation of the density-driven flow is when a density-driven flow has progressed to the same point as the fourth figure of Figure 4.11 case (b). The progress of the density-driven flow is when the air-ingress plume flows through the vessel bottom to the back wall.
Table 4.7. Air mole fraction in the containment and the pressure vessel

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Termination of depressurization</td>
<td>0.461 / 0</td>
<td>0.695 / 0</td>
<td>0.941/0</td>
</tr>
<tr>
<td>Initiation of density driven flow</td>
<td>0.409 / 0.02</td>
<td>0.614 / 0.014</td>
<td>0.909 / 0.005</td>
</tr>
<tr>
<td>Progress of density driven flow</td>
<td>0.382 / 0.041</td>
<td>0.595 / 0.070</td>
<td>0.866 / 0.084</td>
</tr>
</tbody>
</table>

Figure 4.19 shows the accumulated amount of air in the pressure vessel and Table 4.8 summarizes the air-ingress mass flow rates for the all 10 cases. In cases 1-3, the vessel is initially pressurized above atmospheric pressure. The rapid air accumulation at the beginning is due to the flow oscillations near the end of the vessel depressurization. Therefore, the mass flow rates of the pressurized cases are taken after the flow oscillations are complete.

For the cases 4-10, the initial air mole fraction in the containment is decreased from 100 to 40 % in increments of 10%. Since the driving force of the density-driven flow is the density difference between the gases at the pipe break, the results show that the mass flow rate of higher air concentration has a faster air flow rate. The containment air mole fraction of cases 1, 2, and 3, when the density-driven flow initiate was initiated, are 0.409, 0.614, and 0.909, respectively. The air mass flow rates of the pressurized cases display similar tendencies of the equivalent air mole fractions in the containment vessel for the non-pressurized cases. The density-driven flow may depend on the concentration in the containment after the depressurization. The depressurization may
have a minor effect on the subsequent mode of air- ingress due to gas composition changes from the flow oscillation mixing in the vessel.

![Graph showing ingressed air into the vessel with various initial conditions.

Figure 4.19. Amount of air in the vessel with various initial conditions]

![Table showing air-ingress mass flow rate (g/s).

Table 4.8. Air-ingress mass flow rate (g/s)

<table>
<thead>
<tr>
<th>Case no.</th>
<th>Flow rate</th>
<th>Case no.</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.04</td>
<td>6</td>
<td>2.44</td>
</tr>
<tr>
<td>2</td>
<td>1.78</td>
<td>7</td>
<td>2.13</td>
</tr>
<tr>
<td>3</td>
<td>2.74</td>
<td>8</td>
<td>1.81</td>
</tr>
<tr>
<td>4</td>
<td>3.12</td>
<td>9</td>
<td>1.49</td>
</tr>
<tr>
<td>5</td>
<td>2.71</td>
<td>10</td>
<td>1.15</td>
</tr>
</tbody>
</table>

4.5. Local natural circulation

Two cases were run in the calculation: an isothermal case and a non-isothermal case. For the isothermal calculation, the temperatures of the helium in the vessel and the
air in the containment were both set as 300K. For the non-isothermal calculation, the
temperatures of helium in the vessel and air in the containment were set as 1,023 and
300K, respectively. The isothermal case is to see the mechanism of the flow induced by
the density difference of the two different gases and the non-isothermal case is to see the
mechanism change of air-ingress due to the heat transfer from the supporting column
inside the hot plenum.

In calculating a time scale for the density-driven air-ingress, a method was used
similar to that found in Oh and Kim [13]. Following that procedure, a time scale for the
stratified flow was calculated to be 16.07 s. Helium temperature in this case was 850°C
while the outside air temperature was 25°C. A similar calculation was done for a 1/8th
scaled-down facility using the same density ratio (same temperatures). A time scale for
this facility was found to be 5.68 s. It is three times slow than CFD calculation. As
shown in the Figure 4.20, air fills the hot plenum and changes the direction to top as air
plume hit the back vessel in the figure of 2.5 s. It’s time difference comes from the
assumption of scaling analysis: 1. the area of hot plenum path is half of the vessel, 2. the
speed of air plume is at the center of the vessel. In Figure 4.20, the incoming air plume
does not fill the half of the vessel. In addition, as air propagate at the hot plenum, the air
spreads out and it slows down the speed. Thus, its larger path area and slow speed make
the scaling analysis get 3 times slow time scale than CFD calculation.
Figure 4.20. Fluent calculation of air mass fraction for a 1/8th scaled-down facility
Figure 4.21. Accumulated air in the experimental facility

The amount of air in the facility at the cross vessel with respect to time are presented in Figure 4.21. The saturated amount of air difference between isothermal and
non-isothermal case is due to gas density difference in the hot plenum. While isothermal case maintains constant temperature, non-isothermal case heats the incoming gas mixture. For the non-isothermal case, the amount of air in the hot plenum is saturated at approximately 40 seconds. However, the incoming mixture velocity through the cross vessel is maintained.

![Density contour of isothermal (top) and non-isothermal (bottom) at 40 seconds](image)

Figure 4.22. Density contour of isothermal (top) and non-isothermal (bottom) at 40 seconds

On the other hand, the intrusion velocity of the isothermal case decreases with respect to time. This may suggest that the air-ingress mechanism is changing from the density-driven air-ingress phenomenon to hot plenum natural circulation. As shown in Figure 4.22, the density difference is maintained between inside and outside facility for
the non-isothermal case even though the air mass in the hot plenum is saturated. Thus, continuous air-helium mixture flow can be measured for the non-isothermal case. While the flow of the isothermal case is decreased as air fills the inside experimental facility.

4.6. Discussion

The depressurization process of the air-ingress accident is investigated by using FLUENT. By performing a benchmark study, it is demonstrated that a FLUENT pressure-based solver could simulate the pressurized tank depressurization process with less than 3% uncertainty. For the analysis of the air-ingress mechanism, the OSU air-ingress test facility is selected to generate a CFD model. The mesh structural and transient time steps are selected with quantified uncertainty of the discretization error by performing a solution verification study suggested by the ASME standard. It is found that the depressurization of a large break such as a double-ended guillotine break of the cross-vessel follows an isentropic process. As Martineau and Berry [35] demonstrated, the flow oscillations are observed in the FLUENT simulations; however, the oscillations are too fast to measure experimentally. Therefore, the effect of the flow oscillation is investigated by measuring the accumulated effect of the total air concentration in the vessel. The discharged helium during the depressurization process decreases the containment air mole fraction and the flow oscillations at the end of the depressurization increase the air mole fraction in the vessel. Although the depressurization process decreases the density difference around the hot duct pipe break as shown in the third
figure in Figure 4.11 case (b), the diluted gas mixtures near the break by oscillation mixing recovers to the containment concentration as the fourth figure. Therefore, the air-ingress mass flow rate is dominated by the air concentration in the containment. Despite a limited number of case studies for the depressurization being performed, this study shows that the effects of the flow oscillations might be negligible in the overall air-ingress mechanism.
Chapter 5: CFD simulations of the OSU air-ingress test facility

To understand the air-ingress accident, a series of simulations are performed by changing single parameters.

5.1 Simulation matrix

In the test matrix of the experiments, the temperature and pressure of the helium, as well as the break sizes, locations, shapes, and orientations are varied to simulate the effect of different initial and boundary conditions. Test matrixes, which represent minimum sets of experiments for the air-ingress scenario, are summarized in Table 5.1-Table 5.2.
Table 5.1. DEG test matrix

<table>
<thead>
<tr>
<th>No</th>
<th>Vessel initial conditions</th>
<th>Containment initial conditions</th>
<th>Initial air mole fraction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Pressure (MPa)</td>
<td>Temperature (K)</td>
<td>Pressure (MPa)</td>
</tr>
<tr>
<td>1</td>
<td>0.308</td>
<td>973</td>
<td>0.155</td>
</tr>
<tr>
<td>2</td>
<td>0.308</td>
<td>973</td>
<td>0.101</td>
</tr>
<tr>
<td>3</td>
<td>0.155</td>
<td>973</td>
<td>0.155</td>
</tr>
<tr>
<td>4</td>
<td>0.101</td>
<td>973</td>
<td>0.101</td>
</tr>
<tr>
<td>5</td>
<td>0.101</td>
<td>973</td>
<td>0.101</td>
</tr>
<tr>
<td>6</td>
<td>0.101</td>
<td>973</td>
<td>0.101</td>
</tr>
<tr>
<td>7</td>
<td>0.101</td>
<td>973</td>
<td>0.101</td>
</tr>
<tr>
<td>8</td>
<td>0.101</td>
<td>973</td>
<td>0.101</td>
</tr>
<tr>
<td>9</td>
<td>0.101</td>
<td>673</td>
<td>0.101</td>
</tr>
<tr>
<td>10</td>
<td>0.101</td>
<td>673</td>
<td>0.101</td>
</tr>
<tr>
<td>11</td>
<td>0.101</td>
<td>673</td>
<td>0.101</td>
</tr>
<tr>
<td>12</td>
<td>0.101</td>
<td>673</td>
<td>0.101</td>
</tr>
<tr>
<td>13</td>
<td>0.101</td>
<td>373</td>
<td>0.101</td>
</tr>
<tr>
<td>14</td>
<td>0.101</td>
<td>373</td>
<td>0.101</td>
</tr>
<tr>
<td>15</td>
<td>0.101</td>
<td>373</td>
<td>0.101</td>
</tr>
<tr>
<td>16</td>
<td>0.101</td>
<td>373</td>
<td>0.101</td>
</tr>
<tr>
<td>17</td>
<td>0.308</td>
<td>300</td>
<td>0.101</td>
</tr>
<tr>
<td>18</td>
<td>0.308</td>
<td>373</td>
<td>0.101</td>
</tr>
</tbody>
</table>
Table 5.1. CONTINUED

<table>
<thead>
<tr>
<th></th>
<th>Vessel initial conditions</th>
<th>Containment initial conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Pressure (MPa)</td>
<td>Temperature (K)</td>
</tr>
<tr>
<td>19</td>
<td>DEG-30-673-0-300-100</td>
<td>0.308</td>
</tr>
<tr>
<td>20</td>
<td>DEG-30-973-0-300-100</td>
<td>0.308</td>
</tr>
<tr>
<td>21</td>
<td>DEG-30-300-0-300-100</td>
<td>0.308</td>
</tr>
<tr>
<td>22</td>
<td>DEG-14.5-300-0-300-100</td>
<td>0.201</td>
</tr>
<tr>
<td>23</td>
<td>DEG-0-300-0-300-100</td>
<td>0.101</td>
</tr>
<tr>
<td>24</td>
<td>DEG-0-300-0-300-90</td>
<td>0.101</td>
</tr>
<tr>
<td>25</td>
<td>DEG-0-300-0-300-80</td>
<td>0.101</td>
</tr>
<tr>
<td>26</td>
<td>DEG-0-300-0-300-70</td>
<td>0.101</td>
</tr>
<tr>
<td>27</td>
<td>DEG-0-300-0-300-60</td>
<td>0.101</td>
</tr>
<tr>
<td>28</td>
<td>DEG-0-300-0-300-50</td>
<td>0.101</td>
</tr>
<tr>
<td>29</td>
<td>DEG-0-300-0-300-40</td>
<td>0.101</td>
</tr>
<tr>
<td>30</td>
<td>DEG-0-300-0-300-30</td>
<td>0.101</td>
</tr>
<tr>
<td>31</td>
<td>DEG-0-300-0-300-20</td>
<td>0.101</td>
</tr>
<tr>
<td>32</td>
<td>DEG-0-300-0-300-10</td>
<td>0.101</td>
</tr>
<tr>
<td>33</td>
<td>DEG-0-300-0-373-100</td>
<td>0.101</td>
</tr>
<tr>
<td>34</td>
<td>DEG-0-300-0-500-100</td>
<td>0.101</td>
</tr>
<tr>
<td>35</td>
<td>DEG-0-300-0-700-100</td>
<td>0.101</td>
</tr>
<tr>
<td>36</td>
<td>DEG-0-300-0-900-100</td>
<td>0.101</td>
</tr>
</tbody>
</table>
Partial break

Table 5.2. Partial break test matrix

<table>
<thead>
<tr>
<th></th>
<th>Orientation</th>
<th>Vessel initial conditions</th>
<th>Containment initial conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Pressure (MPa)</td>
<td>Temperature (K)</td>
</tr>
<tr>
<td>1</td>
<td>SB0-30-973-0-300-100 Top</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>2</td>
<td>SB90-30-973-0-300-100 Side</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>3</td>
<td>SB180-30-973-0-300-100 Bottom</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>4</td>
<td>CB0-30-973-0-300-100 Top</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>5</td>
<td>CB90-30-973-0-300-100 Side</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>6</td>
<td>CB180-30-973-0-300-100 Bottom</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>7</td>
<td>SIB0-30-973-0-300-100 Vertical(0)</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>8</td>
<td>SIB45-30-973-0-300-100 Diagonal(45)</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>9</td>
<td>SIB90-30-973-0-300-100 Horizontal(180)</td>
<td>0.308</td>
<td>973</td>
</tr>
<tr>
<td>10</td>
<td>SB90-0-373-0-300-100 Side</td>
<td>0.101</td>
<td>373</td>
</tr>
<tr>
<td>11</td>
<td>SB90-0-673-0-300-100 Side</td>
<td>0.101</td>
<td>673</td>
</tr>
<tr>
<td>12</td>
<td>SB90-0-973-0-300-100 Side</td>
<td>0.101</td>
<td>973</td>
</tr>
<tr>
<td>13</td>
<td>CB90-0-373-0-300-100 Side</td>
<td>0.101</td>
<td>373</td>
</tr>
<tr>
<td>14</td>
<td>CB90-0-673-0-300-100 Side</td>
<td>0.101</td>
<td>673</td>
</tr>
<tr>
<td>15</td>
<td>CB90-0-973-0-300-100 Side</td>
<td>0.101</td>
<td>973</td>
</tr>
<tr>
<td>16</td>
<td>SIB90-0-373-0-300-100 Vertical(0)</td>
<td>0.101</td>
<td>373</td>
</tr>
<tr>
<td>17</td>
<td>SIB90-0-673-0-300-100 Vertical(0)</td>
<td>0.101</td>
<td>673</td>
</tr>
<tr>
<td>18</td>
<td>SIB90-0-973-0-300-100 Vertical(0)</td>
<td>0.101</td>
<td>973</td>
</tr>
</tbody>
</table>

Semicircle break (SB), Circumferential break (CB), Slit break (SIB)
The first letter of the test matrix represents the type of the break. DEG, SB, CB, and SIB represent double-ended guillotine breaks, semicircle breaks, circumferential breaks, and slit breaks, respectively. The first and the second number represent pressure and the temperature in the pressure vessel, respectively. The concentration of the vessel is always 100% helium in the simulations, the concentration of the vessel does not need to be represented in this naming. The third, fourth, and fifth number represents the containment pressure, temperature, and the air mole fraction, respectively. Therefore, the first simulation for the double ended case in Table 5.1 DEG-30-973-8-373-80 represents the initial condition of the simulation as the double ended break with the vessel pressure and temperature as 30 psig and 973K. The containment pressure, temperature, and air mole fraction are 8 psig, 373 K and 0.8, respectively.

5.2. Temperature decrease due to the adiabatic expansion during the depressurization process

The depressurization study in section 3.5, which considers the reactor building effects, shows that the air mole fractions and temperatures in the containment are slightly different depend on the vent location as summarized in Table 5.3. The values in Table 5.3 shows at two different vessel pressures when the pressure reaches the OSU experimental facility design pressure 30 psig, and the equalized pressure between the reactor vessel and the containment. During the depressurization process, there is no air
contained in the reactor vessel, therefore, the air mole fraction in the vessel is assumed as zero.

Table 5.3. Pressure and temperature after depressurization

<table>
<thead>
<tr>
<th>Vent location</th>
<th>Vessel</th>
<th>Containment</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Pressure (Psig)</td>
<td>Temperature (K)</td>
</tr>
<tr>
<td>Bottom Right</td>
<td>30.5</td>
<td>295</td>
</tr>
<tr>
<td></td>
<td>7.5</td>
<td>227</td>
</tr>
<tr>
<td>Top Right</td>
<td>30</td>
<td>295</td>
</tr>
<tr>
<td></td>
<td>7.6</td>
<td>240</td>
</tr>
<tr>
<td>Top Left</td>
<td>29.3</td>
<td>292</td>
</tr>
<tr>
<td></td>
<td>7.7</td>
<td>217</td>
</tr>
<tr>
<td>Average</td>
<td>29.93</td>
<td>294</td>
</tr>
<tr>
<td></td>
<td>7.7</td>
<td>224</td>
</tr>
</tbody>
</table>

The helium gas will have adiabatic expansion during depressurization, the gas temperature in the vessel falls below room temperature when the vessel pressure reaches 0.3 MPa. However, the OSU experimental facility cannot initiate the experiment with this condition as the initial condition. Since the temperature decrease instantaneously occurs, the helium temperature will be heated by heat transfer from the internal structure. The temperature of the helium gas in the vessel becomes the same as the internal structure temperature in a few seconds. Therefore, to investigate how the decreased temperature recovers after depressurization process, CFD simulations DEG 18-20 are performed.
The simulations are performed by setting the internal structure and the helium temperature the same at three different temperatures, 373, 673, and 973K, and the pressure as 0.3 MPa in the pressure vessel. The temperature and pressure in the containment are 300 K and 0.1 MPa, respectively.

Figure 5.1. Temperature decrease due to depressurization and temperature changes by the heat transfer from internal structures over time.

Figure 5.1 shows the average temperature in the pressure vessel for simulation DEG 18-20. At the beginning of the simulation, it shows the temperature decrease during the depressurization process. Since the initial pressure is 0.3 MPa, the temperature drop is not as big as the prototype design simulation from 7.0 MPa. The heat flux from the internal structure reaches maximum 16 kW which is about 70% energy that requires
increasing 300 K of the helium temperature in the vessel. The heat flux from the internal structure reaches a maximum 16 kW which is about 70% of the energy required in increasing 300 K of the helium temperature in the vessel. Arcilesi et al. showed the prototypic design temperature recovery would be 2 seconds [36]. This would be 10% of the time that density-driven flow and the heat transfer from the internal structure are faster when the temperature differences are large. Therefore the decreased temperature by adiabatic expansion during depressurization would not have a big effect on the density-driven flow analysis. Therefore, setting the temperature of the gas the same as the internal structure in the experiment would not make a significant different from the prototypic design accident.

Figure 5.2 shows the streamline of simulation DEG 20. The color represents the temperature or the gas. As the cold gas ingresses into the pressure vessel, it flows through the bottom of the vessel and heated up by the internal structure. Since the ingress air flows back to the vessel through the bottom as shown in Figure 5.2, the transient temperature build up are observed at the beginning Figure 5.1.
Figure 5.2. Variation of streamlines according to time

---

5.3. Local hot plenum natural circulation

Figure 5.3 shows the results of DEG 6 and 23. In the isothermal case, with the internal structure case of 300 K, as the air in the containment fills the pressure vessel, there would be no more flow established. As the DEG 6 and 23 uses the internal structure temperature 979 and 300 K, respectively, the difference between the density-driven stratified flow and the hot plenum natural circulation can be understood. The air concentration in the pressure vessel reaches the saturation value in both cases as the air fills the pressure vessel. It takes 54 and 23 sec to reach 95% of the saturation value for the isothermal and non-isothermal cases, respectively. Therefore, about a 10% time scale
was influenced for the temperature recovery during the density driven flow process that was shown in section 5.2.

![Graph showing air concentration in the vessel over time](image)

**Figure 5.3.** Air accumulation in the vessel over time

Figure 5.4 and Figure 5.5 show the air mole fraction and density contour of DEG 6 and 23 at 48 sec. DEG 6 shows that the air is evenly distributed while DEG 23 has a stagnant region at the top of the vessel higher than the hot duct break. Since there is no flow forced in the upward direction for DEG 23, while the vessel's internal structure is at room temperature, the air mole fraction continues to increase until the higher concentration reaches the duct height. Figure 5.7 shows that DEG 6 still has a faster flow than the isothermal case, even though the air accumulation reaches the saturation level. At this time DEG 6 shows the plenum natural circulation, developed by the density difference established vessel temperature.
Figure 5.4. Air mole fraction of DEG 6 (left) and DEG 23 (right) at 48 sec

Figure 5.5. Density contour of DEG 6 (left) and DEG 9 (right) at 48 sec
Figure 5.6. Temperature contour of DEG 6 at 48 sec

Figure 5.7. Flow velocity of DEG 6 (left) and DEG 23 (right) at 48 sec
5.4. Density difference of gas species between the containment and the vessel

The density driven flow is established by the density difference between the gases around the hot duct break. Simulation results of DEG 23-32 are presented in this section to show, according to the air concentration changes, the amount that air ingresses into the vessel.

Figure 5.8. The OSU air-ingress test facility thermo couple locations (red lines)

There are five thermo couples (TC) located in front of the supporting rods located vessel center. To take advantage of these thermocouples in measuring the air-ingress accident progress, air mole fractions, at the TC 3 location, are plotted in Figure 5.9 for the simulation of DEG 21, 23, 25, 27, and 29. Since the data for the simulation are saved discretely every 0.25 sec, the arrival time of AMF 1.0 and 0.8 cases are attached together. The tendencies of the higher air mole fraction cases have faster flow through the reactor vessel.
Arcilesi et al. [18] performed a scaling analysis for a GT-MHR and found that the density driven flow has relations with several pi terms. In this section, the air mole fraction and the viscosity ratio of the initial gas mixture in the containment to the initial helium in the vessel is used to find the connections. Since the air plume reaches TC 3 at different times, the x axis moves the time the plume reaches TC3 as the starting point. Then the air mole fractions measured at TC 3 are divided by the initial mole fraction in the containment. Figure 5.10 shows the results and the values are divided by the viscosity ratio. Figure 5.11 shows that the differences are narrowed by the multiplication of the air mole fraction and viscosity ratio. Further investigations will be performed to find the full relationship with the pi terms in the air-ingress accident propagation.
Figure 5.10. Air mole fraction measurement from the initial time the air plume reaches the thermocouple, divided by the initial air mole fraction in the containment.

Figure 5.11. Viscosity ratio division from the result of Figure 5.10.
5.5. Effects of Break location and size

During the actual accident scenario, the hot duct break could potentially occur in various sizes, shapes, and orientations. The flow behavior of air-ingress would be different for different scenarios. Therefore, it is required to investigate the effects of different break types. Some examples of break plates are shown in Figure 5.12. The last three breaks can be oriented in different ways such as a break at the top (T), side (S), or bottom (B) of the duct. These breaks will be run for a set of pressurized and non-pressurized conditions, all of which will maintain the proper air-to-helium mole ratio.

Figure 5.12. Partial break size, shape, and orientation

Figure 5.13 shows the partial break simulation 1-9. The break cross sectional area of the slit break and circumferential break are a quarter of the area of a semicircular break;
therefore, the accumulated air in the vessel is approximately one quarter of the semicircle. In addition, the density driven flow velocity is related to the height of the break as explained in section 3.2. The side break, which has a higher height among the same size break, has a faster air-ingress. Since the vertical height of the horizontal slit is the lowest, the air concentration in the vessel is the smallest for the horizontal slit.

![Graphs showing air concentration in different shapes and orientations](image)

Figure 5.13. Partial break simulation of different shapes and orientations a) semicircle, b) slit, c) circumferential
5.6. Generalize the density-driven flow

In this section, detailed analyses were performed to identify the density-driven flow. Section 5-4 showed that the density-driven flow has a relationship with the density ratio and the viscosity ratio. The simulation constrained the property changes, except the air concentration in the containment, by changing the air mole fraction gradually. The multiplication of the initial density ratio and the viscosity ratio collapsed the data for the DEG 23-32 cases. However, when the analysis was applied to other data sets, it did not work to collapse the simulations. Therefore, different powers to the ratio, including density ratio, viscosity ratio, and the air concentration ratio, were found to collapse the data. The air concentration was not only changed by the air mole fraction, but also by the temperature changes since the air mole change is inversely proportional to the temperature. The equation to collapse the accumulated air concentration, \( A \), in the vessel is expressed as,

\[
A = \left( \frac{\rho_C - \rho_V}{\rho_C + \rho_V} \right)^{1.1} \times \left( \frac{\mu_C}{\mu_V} \right)^{0.7} \times AMF_C^{0.9} \times \left( \frac{T_V}{300} \right)^{0.9}
\]  

(4.9)

where, \( \rho \), \( \mu \), AMF, and T are density, viscosity, air mole fraction in the containment, and the temperature in the vessel, respectively. The subscript V and C represent the vessel and the containment, respectively. The accumulated air concentration, for selected cases, and the collapsed data are summarized in Figure 5.14 to Figure 5.18, and the overall data are collapsed in Figure 5.19. The standard deviation of the data in Figure 5.19 is 2.12, which is less than 10% deviation.
Figure 5.14. Accumulated air in the vessel for the change of air mole fraction in the containment (top) and the collapsed data by density, viscosity, and air concentration ratio (bottom)
Figure 5.15. Accumulated air in the vessel for the change of temperature in the containment (top) and the collapsed data by density, viscosity, and air concentration ratio
Figure 5.16. Accumulated air in the vessel for the change of concentration in the containment with vessel temperature 973K (top) and the collapsed data by density, viscosity, and air concentration ratio.
Figure 5.17. Accumulated air in the vessel for the change of concentration in the containment with vessel temperature 673K (top) and the collapsed data by density, viscosity, and air concentration ratio.
Figure 5.18. Accumulated air in the vessel for the change of concentration in the containment with vessel temperature 373K (top) and the collapsed data by density, viscosity, and air concentration ratio.
The results in Figure 5.14 and Figure 5.18 were obtained by changing one parameter for each data set. This method was applied to simulation DEG 1-6 which changed the temperature and concentration and temperature in the containment and the vessel simultaneously during the depressurization stage. The properties for the calculation of the density, viscosity, and air concentration for these simulations were taken around the end of the depressurization stage, which is the initial stage of the density-driven flow. The collapsed data in Figure 5.20 shows agreement with the previous analysis and the standard deviation of these simulations is 1.41.
Figure 5.20. Accumulated air in the vessel DEG 1-6 and the collapsed data by density, viscosity, and air concentration ratio
5.7. Discussion

This section shows various simulation results for the air-ingress accident by changing the air concentration, the initial vessel pressure, and the temperature in the vessel and the containment. During the depressurization stage, the vessel's temperature decreased; however, heat transfer from the internal structure recovers the helium temperature to the internal structure temperature in a few seconds. Because the density-driven flow is maintained for about 20 seconds before the hot plenum natural circulation is established, the temperature decrease affected less than 10% of the density-driven stage. So the temperature decrease by depressurization would have limited influence on the density-driven flow. The overall data collapse results show that the density ratio, viscosity ratio, air concentration and temperature ratio in the containment have influence on the air-ingress phenomena.
Chapter 6: Mitigation measures

The air-ingress accident may lead to degradation of graphite structure by oxidation. In extreme cases a loss of structural integrity may occur and lead to the release of radioactive materials. The study about the reactor building compartmentalization in section 3.5.2 showed that 92% of the air vanished from cavity 1. However, since the reactor building is located underground and the density of the gas mixture in the reactor cavity is lower than the air, air would flow back to cavity 1 and to the reactor vessel eventually. Therefore, mitigation strategies to mitigate the air-ingress accident are discussed in this chapter.

6.1 Previous studies

Previous studies introduced pressure build up in the reactor vessel by injecting helium gas in the reactor vessel to prevent intrusion of other gas into the reactor vessel. Yan et al. [37] injected inert helium gas at the top head of a reactor vessel, and Oh et al. [38] injected helium gas at the bottom of the lower plenum. The injection location was decided based on the protection priority. However, helium gas injection in to the reactor vessel requires continuous injections until the internal temperature cools down enough to prevent graphite oxidation. Yan et al. [37] and Oh et al. [38] indicated that it would take
3 months and 6 days, respectively. However, to inject helium gas directly into the reactor vessel, additional penetration is required on the reactor vessel, which makes additional break locations possible for air-ingress.

Oh et al. [38] also suggested additional confinement surrounding the reactor vessel and has an opening at the bottom so that the lighter helium enclosed in the confinement could prevent possible air intrusion into the reactor vessel; however, this design would degrade the passive cooling of VHTR by RCCS. The passive safety design of the VHTR is achieved by heat removal of the reactor vessel to RCCS. The additional enclosure might block the natural convection in the reactor cavity and the radiation by blocking the free space between the reactor vessel and the RCCS.

### 6.2. Reactor building design effect to air-ingress accident

As shown in Figure 3.28, when multi compartment design of the reactor building is considered in the air-ingress analysis, the available amount of air would be limited since the air concentration in the reactor cavity is 92% atmospheric. However, the vessel is located at a relatively lower level as compared to the rest of the compartments, so the heavier gas mixture could flow to the compartment located on lower levels. Figure 6.1 shows the CFD simulation result of how the air can flow back to the reactor cavity.
Figure 6.1. Air mass fraction change over time – air flow back to cavity 1

The simulation is initiated when the reactor vessel and cavity 1 are filled with 100% helium, and cavity 2 is filled with 100% air at 300K and atmospheric pressure. This initial condition represents the state when depressurization is terminated. Even though the concentration in cavity 1 and cavity 2 would not be 100% helium and 100% air,
respectively, after the depressurization process. It is simply the initial condition to identify how heavier air flows into the lower level. The simulation results show that the heavier air located in the higher compartment could easily flow to the lower compartment through the free open surface. Therefore, the multiple compartment design of the reactor building might interrupt the air-ingress accident; however, eventually air will flow back to the reactor cavity which is located lower in the ground in the building and also could flow to the reactor vessel.

6.3. Filling inert gas in the reactor cavity

The concept of the previous studies introduced in section 6.1 is filling the reactor vessel with helium gas. The injected helium gas is chemically inert and the gas will slightly pressurize the reactor vessel to prevent the air-ingress. However, additional penetration to the reactor vessel to inject a gas in to the reactor vessel is not desirable. Therefore, other measures are here.

6.3.1. Filling the reactor cavity with inert gas during normal operation

Air is the heaviest gas in the reactor building during the air-ingress accident; air could flow back to the reactor cavity as shown in Figure 6.1. Therefore, if the reactor cavity is filled with heavier gas than air, it would be hard to establish the air flow back to the reactor cavity. Among the inert gas listed in Table 6.1, Argon is heavier than air and
it is more affordable than krypton and xenon, therefore, it would be the best material to prevent air ingresses by filling the reactor cavity.

Table 6.1. Inert gas density and price

<table>
<thead>
<tr>
<th>Atomic number</th>
<th>Density (kg/m$^3$)</th>
<th>Price ($/m^3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helium</td>
<td>2</td>
<td>0.16</td>
</tr>
<tr>
<td>Nitrogen</td>
<td>7</td>
<td>1.12</td>
</tr>
<tr>
<td>Neon</td>
<td>10</td>
<td>0.81</td>
</tr>
<tr>
<td>Argon</td>
<td>18</td>
<td>1.60</td>
</tr>
<tr>
<td>Krypton</td>
<td>36</td>
<td>3.37</td>
</tr>
<tr>
<td>Xenon</td>
<td>54</td>
<td>5.89</td>
</tr>
</tbody>
</table>

However, the gas placed in the reactor cavity during normal operation would be replaced with depressurized helium during the depressurization stage. As shown in Figure 3.28, most of the gas filled in the reactor cavity would be replaced. Therefore, filling the reactor cavity with argon gas would not be effective.

6.3.2. Injecting argon gas near the end of depressurization stage

During the depressurization stage, the reactor cavity will be mixed with the depressurized helium; therefore, initiating argon gas at the beginning of the depressurization process would waste it. For that reason, the argon injection needs to be initiated near the end of the depressurization. To make this injecting process become passively activated, the injection port could be blocked by utilizing the pressure of the reactor system. The normal operating pressure of GT-MHR is 7.0 MPa. If there is a
break on the pressure boundary the pressure of the reactor will decrease. Figure 3.28 shows how the depressurization progresses during the double-ended guillotine break of the cross vessel. The injecting port could be activated passively when the reactor pressure is lower than set pressure.

To investigate the argon injecting process, CFD model used in section 3.5.2 is modified to have the injection port at the bottom left of cavity 1 as shown in Figure 6.2.

![Figure 6.2. Compartmentalized CFD model with argon injection port at the bottom of cavity 1.](image)

During the depressurization process, the gas species in cavity 1 would be moved to the neighboring compartment. Therefore, if the injection into cavity 1 starts in the early stage of the depressurization process, it would make the density of the gas outside
cavity 1 increase, and the driving forces to cavity 1 from the other compartment would contain more air and argon. Therefore, the argon injection should initiate near the end of the depressurization stage.

Figure 6.3 shows the results cavity 1 and reactor vessel are initially filled with 100% helium, and cavity 2 is filled with 100% air at room temperature and atmospheric pressure. Since during the depressurization process, 92% air moves to the neighbor compartment and fills with depressurized helium. Therefore, filling 100% helium would not make big differences. Initially 1 kg/s of argon injected into the cavity 1. However, as shown in the Figure 6.3, air flow back in to the cavity 1 is observed.
Table 6.2. Air concentration changes with injection and without injection (mol)

<table>
<thead>
<tr>
<th></th>
<th>Initial</th>
<th>10 s</th>
<th>15 s</th>
<th>20 s</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>No injection</td>
<td>Injection</td>
<td>No injection</td>
<td>Injection</td>
</tr>
<tr>
<td>Cavity 1</td>
<td>0</td>
<td>325</td>
<td>235</td>
<td>546</td>
</tr>
<tr>
<td>Vessel</td>
<td>0</td>
<td>2</td>
<td>0.35</td>
<td>5</td>
</tr>
</tbody>
</table>

At the initial stage of argon injection, there are large density differences between Cavity 1 and Cavity 2. As a result, it would take quite a bit of time to increase the density of the gas in cavity 1 after the initiation of argon gas injection. Therefore, air flow back to cavity 1 is observed in this simulation.

![Density contour at 20 sec](image)

Figure 6.4. Density contour at 20 sec
To decrease the time it takes to increase the density of gas mixture in cavity 1, the argon injection rate is increased to 10 kg/s. Even though the injection rate increased, the density of gas mixture in cavity 1 is still lower than cavity 2 at 20 seconds after the argon injection is initiated as shown in Figure 6.4, right. However, the air flow back to cavity 1 is not observed in Figure 6.5, since the injected helium pressurizes cavity 1 so that the air in the cavity 2 cannot flow into cavity 1.

Figure 6.5. Air mass fraction contour at 20 sec (Argon injection rate 10 kg/s)

The argon injection model simulation gave insight on how to inject argon gas into the reactor cavity. The argon injection needs to be initiated near the end of the depressurization process to minimize argon gas loss by the depressurized helium from the
reactor vessel. In addition, the argon injection rate needs to make cavity 1 pressurized to prevent air flow from cavity 2 when the density in the cavity 1 is lower than cavity 2.

To generate future validation data, the CFD model was developed for the OSU test facility as shown in Figure 6.6. Since the containment volume of the OSU test facility is 1 m\(^3\), the required argon gas injection amount would be around 1.5 kg. For the simulation, 1 kg of argon was injected for 2 sec which is slightly more to compensate the argon leakage to top of the containment and to the pressure vessel during the argon injection in to the containment.

![Injection model for the OSU air-ingress test facility](image)

Figure 6.6. Injection model for the OSU air-ingress test facility

Initially the vessel and the containment are filled with 100% helium and top of the containment is filled with 100% air at room temperature and atmospheric pressure. Figure 6.7Figure 6.9 show how argon, air, and helium gas, respectively, flow while
injecting argon gas into the containment. Even though argon gas was injected into the containment to make the density higher than air, there was leakage to the reactor vessel and top of the containment. Therefore, the amount of gas injected was more than expected. When the argon gas injection was terminated at 2 sec, the contours shows that there is no air in the containment and the containment is filled with argon gas which is heavier than the air. Figure 6.10 shows how argon gas injection changes the density in the containment. The 2 kg argon gas injection in to the containment would make slightly higher density than air. However, as the helium in the pressure vessel flow out the vessel and accumulated at the top of the containment, the density at the top of the containment would become lower than the air density so that there is little amount air flow into the containment as shown in Figure 6.11. However, this results show the argon gas injection successfully limits the air-ingress accident. The density decrease in the containment by the helium flow from the vessel would be controllable by the amount argon gas injection.

After injecting the argon gas into the contiennent, The density in the containment is not high enough to make the density higher than air as shown in Figure 6.10. So there is air flow into the containment cab as observed in Figure 6.11. However, the argon injection into the space next to the reactor is effective to mitigate air-ingress.
Figure 6.7. Argon mass fraction contour with 1 kg/s injecting rate
Figure 6.8. Air mass fraction contour with 1 kg/s injecting rate
Figure 6.9. Helium mass fraction contour with 1 kg/s injecting rate
Figure 6.10. Density contour— without injection (left) and with injection (right) at 20 sec

Figure 6.11. Air mass fraction – without injection (left) and with injection (right) at 20 sec
6.4 Physical barrier to air flow

During the depressurization process, fluid in the reactor cavity flows out to neighboring compartments. Since the reactor building is not air-tight, the fluid movements are not disturbed and are established through free surface area. However, if no action is taken, the air could flow back to the reactor cavity, which is located in the lowest place in the reactor building. Therefore, using physical barriers to block the path of the fluid flow in the reactor building would be effective to mitigate the air-ingress accident. Since the physical block not only disturbs incoming flow to the reactor cavity, but also blocks the escaped flow from cavity 1, it would over pressurize the reactor cavity. Therefore, a hinge type gate closed by gravity can be used as a possible solution. This gate could be set to open by pressure build-up in the reactor cavity and closed by gravity when the depressurization is completed. Even though the gate closed by gravity is not air-tight, the gate would disturb the air flow and slow down the flow of air back into the reactor cavity.

6.5. Discussion

This chapter proposed air-ingress mitigation measures. After depressurization, there is almost no air in the reactor cavity; however, the air could flow back to the reactor cavity since the reactor cavity is placed in the lowest place in the reactor building. The heavier air could flow to the reactor cavity through free surface areas in the reactor building. Therefore, Argon gas injection in the reactor cavity is introduced. The
injected argon would prevent the flow by pressurizing the reactor cavity initially, and eventually it prevents the flow by making the gas a heavier density than air in the reactor cavity. In addition, a hinge type physical block closed by gravity is introduced. The gate opens when the reactor cavity is pressurized during the depressurization and it closes by gravity when the depressurization is terminated so that it can slow down the air flow to the reactor cavity.
Chapter 7: Conclusion and future work

7.1 Conclusion

The current work has focused on CFD simulations for various air-ingress accident scenarios in order to understand the air-ingress mechanisms. Since the air-ingress accident is a time-dependent problem, the experimental measurement will be time and location dependent. Therefore, the CFD results of the accident progression have assisted in the design of the experimental facility and the placement of forty-one thermocouples, six O₂ sensors and five pressure transducers to yield optimal experimental results. The following outlines the salient conclusions of this dissertation.

a. To obtain a better understanding of the air-ingress progression that could occur during a cross vessel break and to investigate a CFD simulation performance on the analysis of the air-ingress accident, a 1/30\textsuperscript{th} reduced-length scale-down model of the GT-MHR was constructed and its CFD model was developed. Vegetable oil aerosol particles were seeded in the helium gas located in the vessel, allowing the flow of the helium gas to be visually distinguished from the air flow. Due to the density difference between the air and helium gas, density-driven stratified flow was observed at the duct. The agreement of the velocity measurement and
the CFD results, gives confidence in applying CFD to the air-ingress accident analysis.

d. The OSU 1/8th scaled-down test facility can hold pressure up to 0.343 MPa, at temperature up to 538°C. Since the operating temperature and pressure of the prototypic design is 7 MPa and 850 °C, depressurization analysis is required to provide accurate initial and boundary condition for the scaled-down test facility. To reduce the computational burden of a CFD depressurization analysis, 1-D analytical model was developed. The analytical solution assumed the depressurization of the guillotine break is an isentropic process. This is a reasonable assumption since the depressurization is a fast process and, therefore, the heat transfer from structures can be ignored. The analytical solutions are within the uncertainty range of the CFD solutions (i.e. maximum 3.8 %), so that the analytical solution can provide the depressurization process without sacrificing the accuracy while saving the computational resources. Generally when the pressure of the prototypic design reaches 0.308 MPa (30 psig) and equilibrium to the containment pressure during the depressurization process, the concentration in the vessel were 0.825 and 0.823, respectively.

c. Benchmark studies were performed to select the correct parameters for the CFD model for the air-ingress analysis. Mesh and time refinement studies were performed to reduce the uncertainty of the discretization error. In applying the
Richardson extrapolation method and the GCI index to quantify the uncertainty of the spatial and the temporal discretization error, the refinement ratios of 1.3 and 2 were selected, respectively, to balance the computational time and accuracy of the uncertainty quantification of the grid space and time refinement. For the OSU air ingress test facility, about 1% uncertainty was achieved from the discretization.

d. The CFD simulations for a double-ended guillotine break show that the air-ingress rate is directly proportional to the initial density ratio between the gas mixture in the vessel and the containment. The viscosity ratio between the containment and the vessel, and the temperature ratio of the containment to the reference temperature 300 K were secondary effects. A correlation has been obtained which quantifies these effects. All the double guillotine break simulations are within 10% standard deviation of the correlation results.

e. Although no direct correlation was obtained for all the partial breaks, the CFD simulations for partial breaks show the air-ingress is somewhat a function of the size and the orientation of the break. Using the parameters obtained from the double-ended break, the amount of air accumulated in the vessel for the set of same break size and orientation correlates to within 1% standard deviation.

f. Mitigation measures were presented which utilized the reactor building design and an understanding of the air-ingress mechanism. Argon (a heavy and
relatively affordable) gas can be injected into the reactor cavity near the end of the depressurization process. With the reactor cavity density higher than the air density, the amount air ingressed into the vessel was reduced to 2% of the same accident with no argon injection.

7.2 Future work

Frist, the CFD solutions in this research have followed verification work by performing mesh and time refinement for the time dependent transient problems. However, since the experimental data for validation is not available at this time, the validation work will be performed in the future, once the experimental data is generated.

Second, graphite oxidation model is not implemented in this study which requires for the evaluation the air-ingress accident. The graphite oxidation model will be generated and perform the analysis.

Third, during accident condition the passive safety systems of the VHTR could maintain the vessel wall temperature to 500°C and the RCCS could maintain the reactor building concrete up to 49°C. Therefore, the injected argon would become lighter than air and the temperature differences between two walls make natural convection in the reactor cavity. Therefore, further improvement would be required for the mitigation method.
References


Appendix A: 1-D Depressurization analytical solution

Figure A.1 shows the simplified drawing of vessel and containment. $V, P,$ and $T$ represent the volume, pressure, and temperature, respectively. The subscript $V$ and $C$ represent the RPV and the containment, respectively.

![Figure A.1. Simplified vessel and containment](image)

Isentropic depressurization

Isentropic flow involves constant entropy, adiabatic and frictionless flow.

\[
\left( \frac{T_2}{T_1} \right)^{k/(k-1)} = \left( \frac{\rho_2}{\rho_1} \right)^k = \left( \frac{P_2}{P_1} \right)
\]  \hspace{1cm} (A.1)
\[
\frac{P}{P_0} = \left[ \frac{1}{1 + \left(\frac{k-1}{2}\right)Ma^2} \right]^{1/(k-1)}
\]  \tag{A.2}

\[
\frac{T}{T_0} = \frac{1}{1 + \left(\frac{k-1}{2}\right)Ma^2}
\]  \tag{A.3}

\[
\frac{\rho}{\rho_0} = \left[ \frac{1}{1 + \left(\frac{k-1}{2}\right)Ma^2} \right]^{1/(k-1)}
\]  \tag{A.4}

The mass change in the RPV can be express as,

\[
\frac{dm_v}{dt} = -\dot{m}_v = V \frac{d\rho_v}{dt} = V \frac{d\left(P_v / RT_v\right)}{dt}
\]  \tag{A.5}

\[
\dot{m}_v = -V \rho_{v,t=0} \frac{d\left(P_v / P_{v,t=0}\right)^{1/k}}{dt}
\]  \tag{A.6}

\[
\dot{m}_v = -\frac{V \rho_{v,t=0}}{K} \left(\frac{P_v}{P_{v,t=0}}\right)^{1-k} \frac{d\left(P_v / P_{v,t=0}\right)}{dt}
\]  \tag{A.7}
If the flow is choked, the choking occurs at the location of pipe break $A$. When the pressure inside the vessel, $P_V$ is higher than critical pressure, $P^*_V$, the choking occurs. The Eq. (B.2-4) can be expressed as,

$$P_A = P_V \times \left( \frac{2}{1 + k} \right)^{k/(k-1)} \tag{A.8}$$

$$T_A = T_V \times \left( \frac{2}{1 + k} \right) \tag{A.9}$$

$$\rho_A = \rho_V \times \left( \frac{2}{1 + k} \right)^{1/(k-1)} \tag{A.10}$$

The mass flow rate at the break when choked flow

$$\dot{m}_A = A \rho_A v_A = A \rho_V \left( \frac{2}{k + 1} \right)^{1/(k-1)} \sqrt{kRT_V \left( \frac{2}{k + 1} \right)}$$

$$= A P_V \sqrt{\frac{k}{RT_V} \left( \frac{2}{k + 1} \right)^{(k+1)/(k-1)}} \tag{A.11}$$
The scaled-down test facility initial pressure is higher than the critical pressure; therefore, the analysis can be solved by choked flow assumption. The mass flow through the break, Eq. (A.11), can be express as,

\[ \dot{m}_V = \dot{m}_A \]  \hspace{1cm} (A.12)

\[ \frac{-V P_{V,t=0}}{K} \left( \frac{P_V}{P_{V,t=0}} \right)^{1-k} \frac{d}{dt} \left( \frac{P_V}{P_{V,t=0}} \right) = A P_T \sqrt{\frac{k}{RT_V}} \left( 2 \right)^{(k+1)/(k-1)} \]  \hspace{1cm} (A.13)

\[ \frac{d}{dt} \left( \frac{P_V}{P_{V,t=0}} \right) = -\frac{A}{V} \sqrt{k^3 RT_{V,t=0}} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} \left( \frac{P_V}{P_{V,t=0}} \right)^{\frac{3k-1}{2k}} \]  \hspace{1cm} (A.14)

\[ \left( \frac{P_V}{P_{V,t=0}} \right)^{1-3k} \frac{d}{dt} \left( \frac{P_V}{P_{V,t=0}} \right) = -\frac{A}{V} \sqrt{k^3 RT_{V,t=0}} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} \]  \hspace{1cm} (A.15)

\[ \left( \frac{P_V}{P_{V,t=0}} \right)^{1-k} = -\frac{A}{V} \sqrt{k^3 RT_{V,t=0}} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} \left( 1 - \frac{k}{2k} \right) t + C \]  \hspace{1cm} (A.16)

\[ \left( \frac{P_V}{P_{V,t=0}} \right)^{1-k} \] is one when the time, t, is zero. Therefore the pressure change over time is
\[
(P_v / P_{v,t=0}) = \left[ 1 - \frac{A}{V} \sqrt{k^3 RT_{v,t=0} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} \left( \frac{1-k}{2k} \right)^{1-k}} \right]^{2k} 
\]  
(A.17)

where, \( \rho, v, A \) are the density of gas, velocity, and area of the break. Subscript \( A \) represents the pipe break location.

**Isothermal depressurization**

When the vessel wall temperature is constant and the heat flux from the vessel wall is enough to maintain the gas temperature inside the vessel, the mass change in the RPV can be express as,

\[
\frac{dm_v}{dt} = -m_v = V \frac{d\rho_v}{dt} = V \frac{d(P_v)}{RT_v dt} 
\]  
(A.18)

The discharge flow from RPV is instantaneous, the mass flow rate at the break when choked flow is Eq. (A.11).

The mass change inside the vessel, Eq. (A.18), and the mass flow through the break, Eq. (A.11), can be express as,
\[ \dot{m}_v = \dot{m}_a \]  
(A.19) 

\[ -\frac{V}{RT_v} \frac{d(P_v)}{dt} = AP_v \sqrt{\frac{k}{RT_v}} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} \]  
(A.20) 

\[ -\frac{1}{P_v} \frac{d(P_v)}{dt} = \frac{A}{V} \sqrt{kRT_v} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} \]  
(A.21) 

\[ \left( \frac{P_v}{P_{v,t=0}} \right) = \exp \left\{ -\frac{A}{V} \sqrt{kRT_v} \left( \frac{2}{k+1} \right)^{(k+1)/(k-1)} t \right\} \]  
(A.22)
Appendix B: Mass fraction contour of density driven flow benchmark study using different viscous model for the experiment of Grobelbauer et al. [30]

1. Laminar, 2. RSM, 3. Standard k-ε, 4. k-ε realizable

(a). CO₂-Argon at 14 sec
(b). Argon-Air at 6 sec

(c). R22 (Chlorodifluoromethane)- Argon at 4 sec
(d). R22 and Air at 3 sec

(e). He-Air at 2.2 sec
(f). Argon- Helium at 2 sec

(g). R22-He at 1.4 sec
Appendix C: side view of Air mass fraction contour for selected CFD simulations

Double Ended Guillotine (DEG) break

Semicircle break (SB)

Circumferential break (CB)

Slit break (SIB)
a) 0.5 sec                             b) 1.0 sec                             c) 1.5 sec

d) 2.0 sec                              e) 2.5 sec                             f) 3.0 sec

g) 3.5 sec                             h) 4.0 sec                             i) 5.0 sec

j) 6.0 sec                             k) 7.0 sec                             l) 8.0 sec
a) 0.5 sec                             b) 1.0 sec                             c) 1.5 sec
d) 2.0 sec                              e) 2.5 sec                             f) 3.0 sec
g) 3.5 sec                             h) 4.0 sec                             i) 5.0 sec
j) 6.0 sec                             k) 7.0 sec                             l) 8.0 sec

DEGB-0-300-0-300-100
a) 0.2 sec                             b) 0.6 sec                             c) 1.1 sec

d) 1.6 sec                             e) 2.1 sec                             f) 2.6 sec

g) 3.1 sec                             h) 4.1 sec                             i) 5.1 sec

j) 6.1 sec                             k) 7.1 sec                             l) 8.1 sec

SB90-30-973-0-300-100
a) 0.2 sec                             b) 0.6 sec                             c) 1.1 sec
d) 1.6 sec                              e) 2.1 sec                             f) 2.6 sec
g) 3.1 sec                             h) 4.1 sec                             i) 5.1 sec
j) 6.1 sec                             k) 7.1 sec                             l) 8.1 sec
SB180-30-973-0-300-100
a) 0.2 sec                             b) 0.6 sec                             c) 1.1 sec
d) 1.6 sec                              e) 2.1 sec                             f) 2.6 sec
g) 3.1 sec                             h) 4.1 sec                             i) 5.1 sec
j) 6.1 sec                             k) 7.1 sec                             l) 8.1 sec

CB0-30-973-0-300-100
a) 0.2 sec                             b) 0.6 sec                             c) 1.1 sec

d) 1.6 sec                              e) 2.1 sec                             f) 2.6 sec

g) 3.1 sec                             h) 4.1 sec                             i) 5.1 sec

j) 6.1 sec                             k) 7.1 sec                             l) 8.1 sec

CB90-30-973-0-300-100
a) 0.2 sec                             b) 0.6 sec                             c) 1.1 sec

d) 1.6 sec                              e) 2.1 sec                             f) 2.6 sec

g) 3.1 sec                             h) 4.1 sec                             i) 5.1 sec

j) 6.1 sec                             k) 7.1 sec                             l) 8.1 sec

SIB0-30-973-0-300-100
a) 0.2 sec                             b) 0.6 sec                             c) 1.1 sec

d) 1.6 sec                              e) 2.1 sec                             f) 2.6 sec

g) 3.1 sec                              h) 4.1 sec                             i) 5.1 sec

j) 6.1 sec                             k) 7.1 sec                             l) 8.1 sec

SIB45-30-973-0-300-100
a) 0.2 sec  
b) 0.6 sec  
c) 1.1 sec  
d) 1.6 sec  
e) 2.1 sec  
f) 2.6 sec  
g) 3.1 sec  
h) 4.1 sec  
i) 5.1 sec  
j) 6.1 sec  
k) 7.1 sec  
l) 8.1 sec  
SIB90-30-973-0-300-100