Spring 2017

Design and analysis of a passive heat removal system for a small modular reactor using STAR CCM+

Raymond Michael Fanning

Department:

Recommended Citation

Fanning, Raymond Michael, "Design and analysis of a passive heat removal system for a small modular reactor using STAR CCM+" (2017). *Masters Theses*. 7640.
https://scholarsmine.mst.edu/masters_theses/7640

DESIGN AND ANALYSIS OF A PASSIVE HEAT REMOVAL SYSTEM FOR A SMALL MODULAR REACTOR USING STAR CCM+

by

RAYMOND MICHAEL FANNING

A THESIS

Presented to the Graduate Faculty of the

MISSOURI UNIVERSITY OF SCIENCE AND TECHNOLOGY

In Partial Fulfillment of the Requirements for the Degree

MASTER OF SCIENCE IN NUCLEAR ENGINEERING

2017

Approved by

Ayodeji B. Alajo, Advisor

Joshua P. Schlegel, Co-Advisor

Gary E. Mueller

PUBLICATION THESIS OPTION

This thesis is formatted to the Missouri University of Science and Technology formatting specifications and consists of the following article that has been submitted for publication:

DESIGN AND ANALYSIS OF A PASSIVE HEAT REMOVAL SYSTEM FOR A SMALL MODULAR REACTOR USING STAR CCM+, pages 1-29 have been submitted to the ASME.

ABSTRACT

Next generation nuclear power plants, specifically small modular reactor designs, are the best alternative to fossil fuels for power generation due to their power density and low carbon emissions and constant awareness of safety concerns. A promising safety feature of new designs is the removal of heat by passive systems in accident scenarios. The passive systems require no moving parts and no intervention by personnel. These systems must be accurately simulated for better understanding of the heat transport phenomena: natural convection cooling. Due to the fact that most work developing these passive heat removal systems are proprietary information, a passive heat removal system for a small modular reactor was designed and simulated in Star CCM+ to evaluate the capability of natural convective flows to remove decay heat in a shutdown scenario. The size and dimensions of the heat exchanger are based on the Westinghouse-SMR design. The design of the passive heat removal system was a hexagonal lattice heat exchanger. The final design was projected to dissipate the 56MW of decay heat at the rate simulated in Star CCM+.

ACKNOWLEDGEMENTS

I would foremost like to thank the one true God for His direction and grace. This journey I have been on has been a long one and without the help I received, I would not have made it this far.

I would also like to thank my loving family for their continued support from the time I started graduate school to the completion of this work. I want to thank my professors and academic advisers for their counsel and wisdom.

And lastly I would like to thank my campus minister, Andy Conklin, for being more than just an awesome source of encouragement and wisdom. He has been a dear friend to me and, with my small circle of close friends and family, helped maintain my sanity.

TABLE OF CONTENTS

APPENDICES

SECTION

LIST OF ILLUSTRATIONS

Page

LIST OF TABLES

NOMENCLATURE

Acronym	**Definition**
W-SMR	Westinghouse Small Modular Reactor
UHS	Ultimate Heat Sink
HE	Heat exchanger
Rx	Reactor
RANS	Reynolds Averaged Navier Stokes
CFS	Coupled Flow Solver
SFS	Segregated Flow Solver

PAPER

I.DESIGN AND ANALYSIS OF A PASSIVE HEAT REMOVAL SYSTEM FOR A SMALL MODULAR REACTOR USING STAR CCM+

by

RAYMOND MICHAEL FANNING

ABSTRACT

Next generation nuclear power plants, specifically small modular reactor designs, are the best alternative to fossil fuels for power generation due to their power density and low carbon emissions and constant awareness of safety concerns. A promising safety feature of new designs is the removal of heat by passive systems in accident scenarios. The passive systems require no moving parts and no intervention by personnel. These systems must be accurately simulated for better understanding of the heat transport phenomena: natural convection cooling. Due to the fact that most work developing these passive heat removal systems are proprietary information, a passive heat removal system for a small modular reactor was designed and simulated in Star CCM+ to evaluate the capability of natural convective flows to remove decay heat in a shutdown scenario. The size and dimensions of the heat exchanger are based on the Westinghouse-SMR design. The design of the passive heat removal system was a hexagonal lattice heat exchanger. The final design was projected to dissipate the 56MW of decay heat at the rate simulated in Star CCM+

1 INTRODUCTION

Passive safety is a growing trend in the power industry. The safety and public acceptance benefits that passive safety systems offer are unique and increasingly necessary for the effectiveness of nuclear energy acceptance and safety concerns. Public opinion has swayed nuclear-related issues in the past, such as geological repository at Yucca Mountain (Batt, 1992). Passive safety is a cornerstone of public outlook and natural convection is the foundation of passive safety systems in Generation III+ nuclear power plant design.

Accident probability and risk assessment are major contributors to the design of advanced reactors. The economical evaluation of new designs is constantly balanced with their effectiveness as well as safety limitations. Small modular reactors have been designed to improve the economic feasibility of nuclear energy and be more appealing to investors as well as governing safety regulators. (Weinberg, 1985) (Nayak and Sinha, 2007) The future of nuclear power safety was documented in 'The Safety of Nuclear Power' (IAEA, 1992), which stated that reducing complexity and the reduced dependence on operator action would be required of new systems to improve upon existing safety regulations. It was suggested this might be accomplished by implementing passive safety into the design process.

The IAEA defines passive components as components that do not require any external input to operate and define passive systems as being composed of entirely passive components or only using active components in a very limited way. Passive components and systems would drastically reduce the amount of human errors possible for a nuclear power plant when operating or when encountering an accident.

Thus, such systems make the plant resistant to the series of human errors that resulted in the incidents at Three Mile Island and Chernobyl. (Angelo and Andrade, 2012) Accidents have and will always continue to be a tremendous source of lessons learned. The more reliability that can be inherent in a safety system, the less risk there will be. The potential for passive safety reliability is a significant motivating factor for SMR designs to consider them in the design process. (IAEA, 2005)

The intent of safety systems is to ensure that accident scenario will not lead to meltdown or more serious consequences such as the exposure of the general public to radiation from the reactor system. Passive safety systems accomplish this with neither active personnel intervention nor significant use of powered system components.

Natural convective cooling allows a power plant to remove heat from an accident scenario without the aid of the personnel on site; this is classified as a type B passive safety system (IAEA, 2011). The ultimate heat sink (UHS) reservoir must provide a high enough transfer of heat so that a natural circulation flow will develop (Schultz, 2012). The density difference will develop a velocity that determines the dissipated heat from the reactor.

A small modular reactor like the Westinghouse-SMR would be one such application for natural convection passive safety systems. The heat exchanger would be required to dissipate the decay heat of the reactor after an accident scenario has induced shutdown of the reactor. This would mean that the natural circulation flow would have to achieve a specific flow rate to dissipate enough heat to keep the water in the reactor core from boiling; thus preventing fuel exposure. The current development of small modular reactor technology is primarily being achieved by

NuScale and Westinghouse. However, this means that the information concerning the design of the passive safety systems for those SMR designs is proprietary.

The goal of this research is to design a passive heat removal system that would be able to dissipate the 56 MW of decay heat in the event of a shutdown of the Westinghouse-SMR design. This passive safety system would be a benchmark for future design of heat exchanger systems for the scale of small modular reactors. The passive safety system would need to be small enough to be implemented within the W-SMR design but able to dissipate its decay heat effectively.

2 SYSTEM DESCRIPTION

Systems providing passive safety are designed with the assumption that the system will not have pumping power to provide circulation for heat removal from the reactor core. Therefore natural convection is required for proper heat removal. The effectiveness of the natural convection heat exchanger is dependent on the surface area available for heat transfer between fluids. For this reason, a hexagonal lattice was chosen. The hexagonal lattice affords the high surface area for heat transfer without unnecessary complication of the geometry. Figure 2.1 is a conceptual layout of the Westinghouse-SMR. The Westinghouse-SMR was chosen as a benchmark design over the NuScale SMR design due to contact with Thomas Kindred, a Missouri University of Science and Technology alumnus who works for Westinghouse.

The passive heat removal system dimensions were chosen based on the Westinghouse small modular reactor design. This helps benchmark the size of a heat exchanger system that is representative of reactors that operate at 800 MW (t). It was determined that the heat exchanger could be no more than 2 meters in diameter. The pipes are spaced accordingly in the hexagonal lattice to fit an appropriate number of pipes into the specified diameter (4045 pipes of 1.25 cm outer diameter and a pitch of 2.5cm). As shown in the conceptual geometry of Figure 2.2, the inlet pipes come into the heat exchanger, bringing in the reactor water, flow downward and then out of the heat exchanger by outlet pipes that go back to the reactor to repeat the cycle. The feed water for the passive safety heat exchanger comes from the ultimate heat sink (UHS). In the conceptual design shown in Figures 2.1 and 2.2, there are only 37 pipes. The purpose of these figures is to show the hexagonal lattice structure of the pipes. The actual design contains 4045 tubes. Pipes with diameter of 1cm are common in heat exchanger design and will afford more heat transfer surface area (Ametek, 2016).

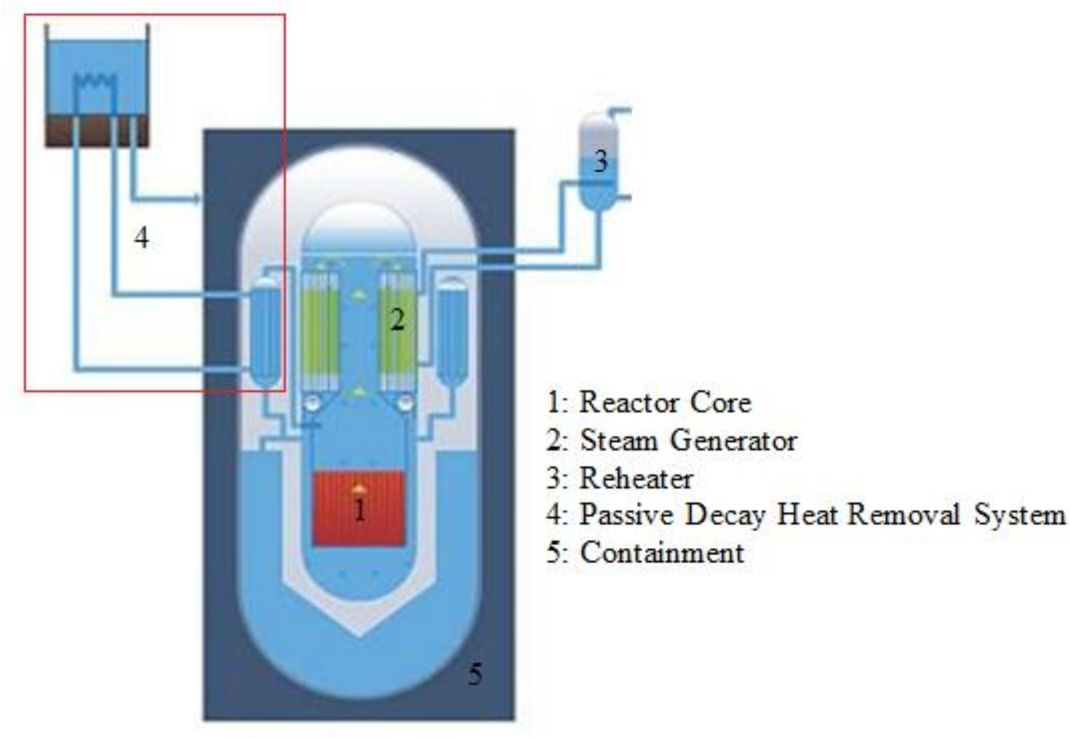

Figure 2.1:Westinghouse-SMR Concept Design (Smith and Wright, 2012)

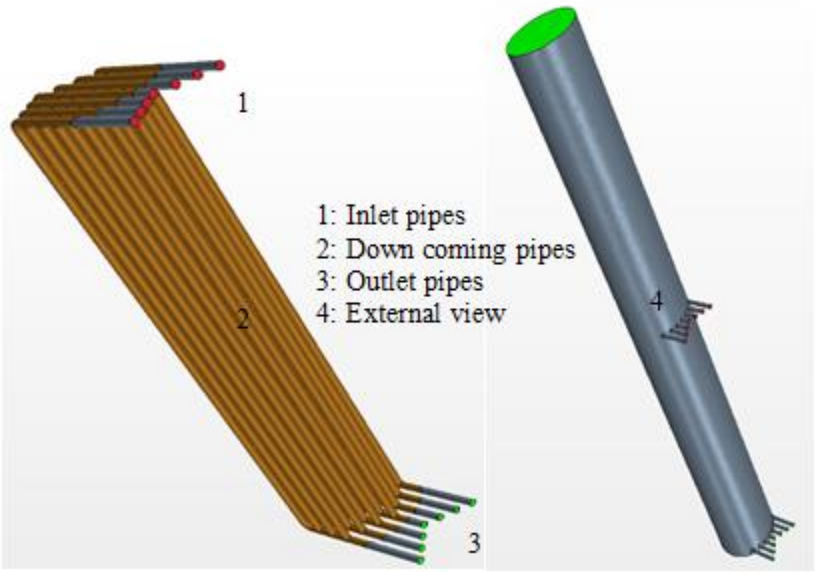

Figure 2.2:Geometry for Hexagonal Heat Exchanger Internal and External View

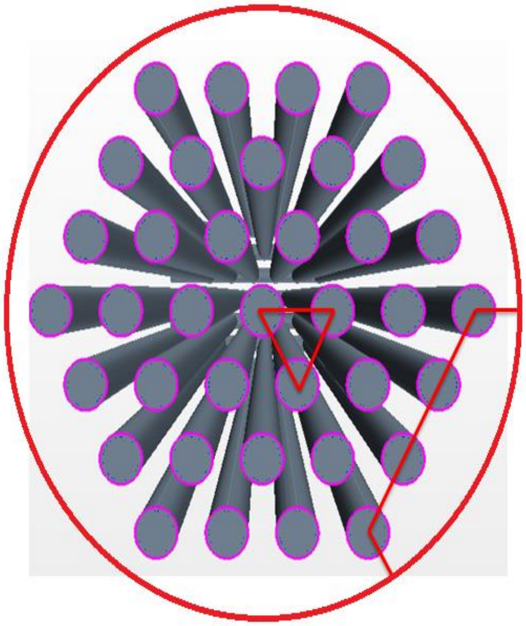

Figure 2.3:Simplification of Geometry

3 FEASIBILITY VERIFICATION

Research is being performed small modular reactor technology to determine and improve its effectiveness in the commercial industry of power production. The reliability of passive safety systems for small modular reactors was evaluated through a RELAP-5 simulation of the Multi Application Small Light Water Reactor (MASLWR) design (Butt &Ilyas, 2016). The passive safety systems of the URANUS (Shin & Choi, 2015), and SLIMM (Haskins & El-Genk, 2017), SMR designs both used natural circulation of liquid metals for passive heat removal. An analysis of the systems that make up the W-SMR passive residual heat removal system was performed for a LOCA using the WCOBRA/TRAC-TF2 systems computer code (Liao and Kucukboyaci, 2016). However, from that analysis, the dimensions of those systems are not discussed in detail due to the proprietary nature of that information. A design that shows the dimensions and simulation of the passive heat exchanger in Star CCM+ that can dissipate the decay heat of a small modular reactor has yet to be performed.

The first analysis is a one dimensional, first principle approach to the natural convection flow development. The first constraint that is accounted for is the critical heat flux (CHF). CHF is evaluated in the reactor core itself because it is the heat flux at which the departure from nucleate boiling is observed. This can lead to boiling off of the reactor water and eventually the melting of fuel, which is why it is a design constraint. The flow rate in the reactor core would be dependent on the natural convection flow that develops in the passive heat exchanger. The flow rate of the reactor water in the passive heat exchanger can be approximated iteratively by assuming values in (1)

$$v_{ref}^{3} = \frac{2g\beta\Delta H}{\Sigma\left(\frac{f(Re)_i L_i}{D_i}+\Sigma k_{ij}\right)\left(\frac{A_{ref}^{2}}{A_i^{2}}\right)}\frac{Q}{\rho_{ref}c_p} \tag{1}$$

where ref designates the passive heat exchanger,v_{ref} is the reference velocity, g is gravity, β is the coefficient of thermal expansion, $\Delta H$is the height change between the heat exchanger and the UHS, $f(Re)_i$ is the friction factor of the ith component, L_i is the length of the ith component, D_i is the diameter of the ith component, k_{ij} is the inlet and exit pressure losses of the ith component, A_{ref} is the reference area, A_i is the area of the ith component, Q is the heat flux from the reactor, ρ_{ref} is the reference density, and c_p is the specific heat. The value of the velocity in the passive heat exchanger, v_{ref}, can be obtained by approximating the reference velocity with an initial guess then calculating the new value and repeating until convergence. A derivation of Eq. (1) is shown Appendix II. This equation calculates the steady state velocity from the head loss and driving head due to buoyancy to be approximately 0. 15 m/s if the flow area of the HE is 0.37m^2 (4045 pipes of 1 cm diameter) and the pipes that come from the reactor have an effective cross sectional area of 0.19625m^2. This area was chosen as a rough estimate of the piping that would exist between the two components for transport of the water from the reactor to the heat exchanger and from the heat exchanger back to the reactor core after heat transfer.

The feed water velocity that would be required can also be approximated by assuming a change in temperature that will arise from the heat transfer. The equation for the total heat transferred to the feed water is:

$$Q = mc_{p_f}\Delta T_f + L_e m x \qquad (2)$$

where Q is the total heat to be transferred (7% of the SMR operational power), m is the mass flow rate, c_{p_f} is the specific heat of the liquid feed water, L_e is the latent heat of evaporation, x is the exit quality of the mixture (~12%vapor), and ΔT_f is the change in temperature from the inlet to the boiling temperature. The quality of 0.12 is typical of steam generator design. The properties of water for this equation can be found in the appendix. Approximately 90 ft of water was assumed to be distance between the heat exchanger and the UHS. This means that there would be a pressure of 47.02 psi at the heat exchanger. By assuming a steady state temperature increase of 109.34K ($\Delta T_f = 109.34$)the mass flow rate can be calculated by solving the equation for m. Using the Westinghouse-SMR upper limit of operational power, 56 MW (t) of decay heat, this equation gives a steady state feed water velocity of 0.1 m/s for each sixth of a flow channel which will be discussed in the geometry section. .

This approximation describes the velocity reached at steady state in this system, which can be used to determine this systems capability of removing the decay heat. But this approximation does not describe the time that will be required before this flow velocity is developed. This transient time necessary for the development of flow on the reactor and feed water side of the heat exchanger is paramount to the classification of a heat removal system as passively safe, however, analysis of the transient is beyond the scope of this work.

4 COMPUTATIONAL CODE

Star CCM+ is a computational engineering package first developed for CFD purposes which has grown to include heat transfer and solid stress models. Star CCM+ is updated to provide the best practices and features for engineers to use in simulation. It is capable of simulating the heat transfer in a simplified natural convection system, provided that the user knows the limits and constraints of the models available in the Star CCM+ package. The Star CCM+ and Star-CD codes were designed to solve both fluid flow and heat transfer problems simultaneously, unlike other codes that try to do this separately and justify their results post solving. CD-Adapco developed the first commercially available polyhedral meshing algorithm for use in Star CCM+, which includes automatic surface repair for computational domains and many advantages over tetrahedral and hexahedral meshing methods. Using the Star CCM+ polyhedral meshing algorithm allows the same degree of accuracy in simulation while using considerably fewer cells and less computational resources such as RAM, and processor runtime. (Peric and Ferguson) However, a limitation of Star CCM+ is that it requires the user to control all the inputs with precision for accurate results. This can be quite extensive and requires a learning period before reasonable results are obtained. In the same manner as all computational fluid dynamics codes, Star CCM+ follows the rule: garbage in, garbage out. Star CCM+ requires input of geometry, mesh, models, boundary conditions and inlet conditions to set up the scope of the simulation. The geometry can be made in Star CCM+ or imported from an accepted CAD program. The mesh function is unique, as previously discussed but can be imported from select CFD packages. The models are chosen within the continua node of Star CCM+ and must accurately represent the physical phenomena being simulated. The same can be said of the boundary conditions and the initial conditions. If they do not, the simulation will have instability that can lead it to divergence or incorrect results.

4.1 COMPUTATIONAL DOMAIN

The conceptual geometry is shown in Figure 2.2; however, to model a system 6.1 m in length and 1.5 m in diameter would require a copious amount of computational power to obtain the desired detail of the natural convection development. The computational constraints lie with the mesh characteristics. For the mesh to account for the small thickness of the pipes and the length it would need to be very fine in the radial direction and coarse in the axial direction. This would make the cell count for the file incredibly large and it would hinder the apprehension of results.

4.1.1 Simplifications. It is important to consider computational power as a finite resource that must be managed efficiently. To simplify the heat exchanger system for computational practicality, the heat exchanger can be cut into a single triangular flow channel. Figure 2.3 contains the cuts conducted in order to simplify the model into two sections. This will allow the model to use a finer mesh and still give accurate information. Then, by using lines of symmetry, the triangular flow channel can be reduced again to a sixth of that triangle. Figure 4.1 shows the two finished geometries for this simulation. Figure 4.2 shows the final geometry to be drawn into Star CCM+ using its computer aided design (CAD) functions.

It is possible to reduce this geometry by symmetry because Star CCM+ has the ability to interpret surfaces as symmetry boundaries, which act as if the same geometry on one side of the face is projected to the opposite side of the face. Reducing the problem by symmetry will allow the mesh to be of the appropriate size, approximately 4-6 cells in the smallest region. If the geometry were not to be reduced, the mesh would have many more cells and require more computational time. The single flow channel analysis is very common in systems that have lattice structures and can be repeated to look at an entire system that has symmetry. However, the symmetry reduction of the geometry must be applicable for both heat transfer and fluid flow in the system. Therefore, an interface must be created in Star CCM+ that has a periodic rotational topology. This means that Star CCM+ will simulate the fluid flow and heat transfer as if the geometry from Figure 4.2 were to be rotated symmetrically and simulate a hot water pipe with feed water surrounding it. By using this interface, the accuracy of the simulation can be kept from diminishing while conserving computational resources.

4.1.2 Assumptions. The reactor water was assumed to be a constant temperature over time because the amount of heat removed over time would be similar in comparison to the decay heat. The reactor side water is assumed to be 615^{o}F (598K) from the W-SMR core outlet temperature. It is beyond the scope of this research to model how the water is diverted to the passive safety system heat exchanger during accident initiation. Therefore, the system will be modeled as if the heat exchanger has already been filled immediately following shutdown initiation. The inlet temperature of the feed water side is assumed to be 300K.

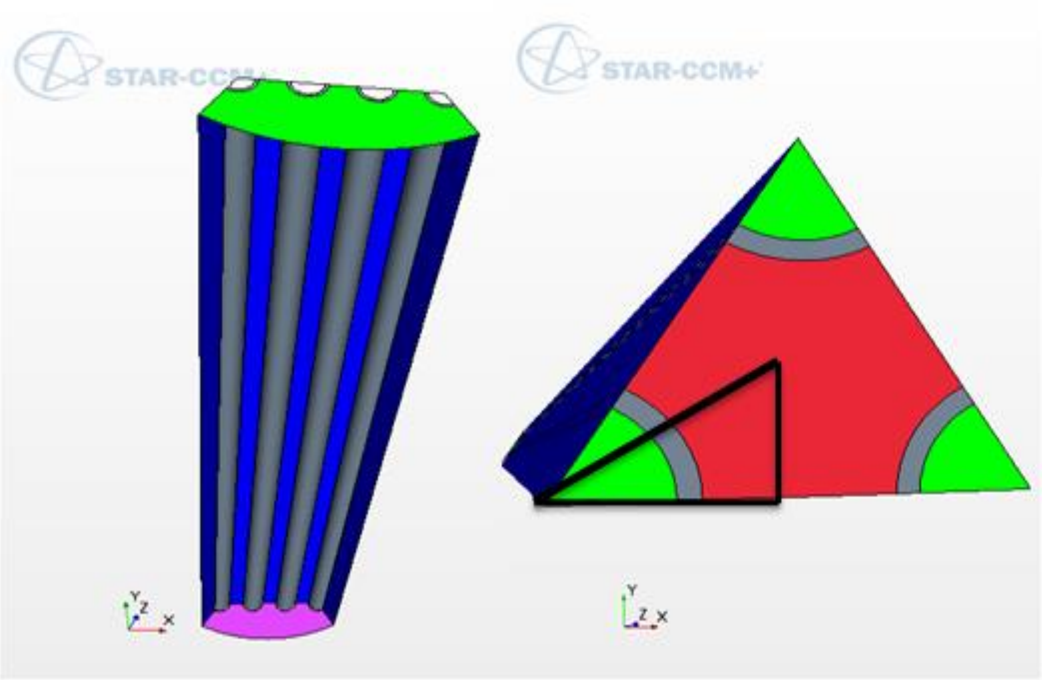

Figure 4.1:Two Simplified Geometries

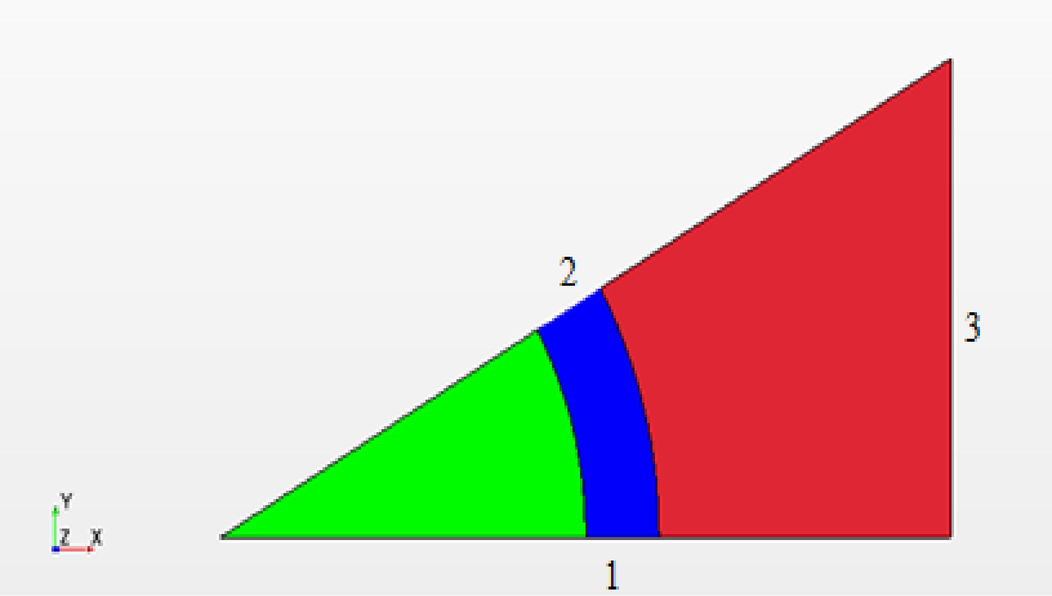

Figure 4.2:Final Geometry Reduction

4.2 COMPUTATIONAL MESH

Several different meshing models were considered for this geometry. Directed meshing was applied because it affords the best runtime with accuracy of the simulation. Directed meshing algorithms allow a 2D mesh to be extended into an axial direction and layered in a structured manner that can be chosen by the user. But the directed mesh was ultimately abandoned due to error output from the AMG solver. The AMG solver uses linearization of the solution to approximate the next guess for each of the solvers for the pressure, energy, turbulence, momentum, etc. It is used to reduce the number iterations before convergence. The polyhedral mesher is a very stable and reliable mesh for different geometries and models. This is because of the advantages of the number of neighboring cells that an individual cell will have. The polyhedral cells will have more neighboring cells compared to the tetrahedral or hexahedral cells, which makes the mesh more robust when encountering very large gradients in temperature or velocity or mass transfer. The axial cell size is larger because it does not require as many cells to show the bulk movement of the natural circulation. This also lets the mesh function create fewer cells. It is important to note that the number of cells in the mesh will be proportional to the speed of simulation completion. If there are more cells, the time it takes for a single iteration to complete increases and subsequently the entire runtime.

Figure 4.3 shows the mesh used for the final results. The smallest dimension of the geometry is the thickness of the reactor water pipes. Thus, to achieve the necessary number of cells within the smallest dimension, the directed mesher allows the user to set the curvature of the cells to fit the smallest geometry in that region. This is to improve the accuracy of the simulation.

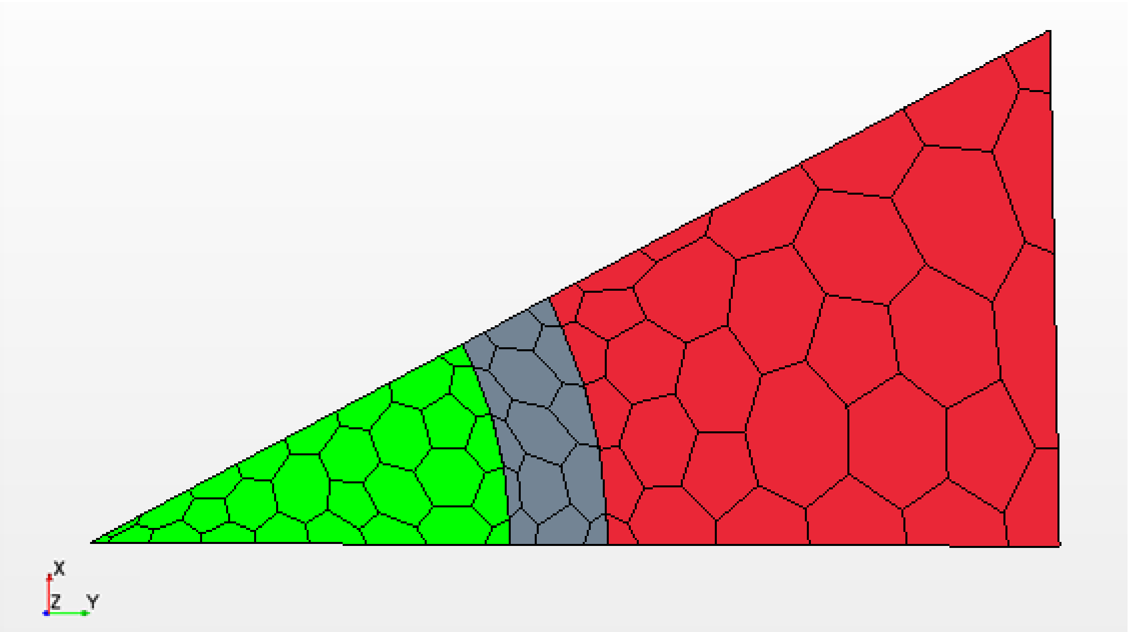

Figure 4.3:Mesh

4.3 BOUNDARY CONDITIONS

The boundary conditions used in the Star CCM+ simulation is described here for the purpose of explaining why they were chosen compared to other models. The boundary conditions and models for the simulation are detailed later in the methodology section.

4.3.1Symmetry. An interface is created from the hypotenuse (side 2) and the base (side 1) sides of the geometry from Figure 4.3. The interface is then chosen as a periodic rotational topology and the third side is kept as a symmetry boundary. Since the entire pipe lattice is hexagonal and the section tested is a piece of the entire problem this boundary condition is necessary for simulating the contribution of fluid flow and heat transfer from the surrounding pipes more accurately.

4.3.2Velocity Inlet. The inlet boundary for the feed water and the reactor water in the simulation is set to be velocity inlet with the velocity specifications mentioned previously in the feasibility analysis. The velocity inlet is set as constant unless specified by a field function to be a function written in the Star CCM+ language.

4.3.3Pressure and Flow Split Outlet. The pressure outlet boundary allows a pressure to be set for expected change in the pressure along the length of the pipe; the static head due to water above will decrease as the water travels upward toward the top of the heat exchanger. It was determined that the approximate pressure difference between the bottom and top of the flow channel is approximately 8 psi. Therefore the pressure outlet can be set to 8psi on the feed water side. The flow split outlet does not require any pressure setting and was used in some simulations for increased stability.

4.3.4Wall. All of the boundaries not associated with the inlets and outlets or symmetry planes are considered wall boundaries by default. The wall boundary, whether in a fluid or solid region, prevents the escape of mass through that boundary. These boundaries will transfer heat and can have specified thermal resistances. The material properties were set appropriately by the physics models of each material.

4.4 MODEL SELECTION

The physical models chosen for the simulation of the design shown in the next sections are described here and listed in later sections for reference.

4.4.1Three-Dimensional, Gravity and Density. Adequately understanding the physics of the simulation is necessary for the implementation of the correct models. This geometry is not symmetrical in more than two directions, so a three-dimensional problem is necessary to accurately predict the desired results. The density of stainless steel 316 will not vary considerably because the melting point of this material is at 2500 °F. This system also requires the gravity model to appropriately describe the density driven buoyancy that will develop the natural circulation flows in the feed water. The constant density model is chosen for the setting of the thermal properties for each of the fluids. This model requires that the density, viscosity, latent heat of vaporization, thermal conductivity, and the specific heat be set accordingly for each fluid at the specified temperature and pressure that is set in the simulation. These values are described in more detail in the results section.

4.4.2Gradients. Gradients are required because they reconstruct field values at the cell faces, and provide secondary gradients for diffusion terms. In addition, pressure gradients are used for pressure-velocity coupling in the segregated flow model and for strain-rate and rotation-rate calculations for turbulence models.

4.4.3Implicit Unsteady and Steady State Models. The simulation preformed in this investigation used the steady state model for time. This model is different from the implicit unsteady model where time is kept as a constant time marching technique. Although the inlet conditions for a non-forced flow should be time dependent, the steady state model is being used to set up the foundation for later implicit unsteady modeling of the flow development.

4.4.4Liquid. The state of the reactor water in the simulation will be liquid due to the operating pressure of the reactor (2250psi) being diverted into in the passive heat exchanger system.

4.4.5Segregated v. Coupled Flow and Energy. The segregated flow solver determines each of the momentum transport equations, x, y, and z. Then, the momentum and continuity equations are linked with a predictor-corrector approach. This model uses a co-located variable arrangement and a Rhie-and-Chow-type pressure-velocity coupling combined with a SIMPLE-type (Semi-Implicit Method for Pressure Linked Equations) algorithm. The coupled flow model simultaneously solves mass, momentum, and energy using a pseudo-time-marching approach (Eqns. (3)(4)(5)).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho u \tag{3}$$

$$\frac{\partial u}{\partial t} + (u \cdot \nabla) u = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon \nabla^2 u \tag{4}$$

$$\rho \frac{De}{Dt} = \dot{q}''' + k\nabla^2 T + \phi \tag{5}$$

The number of iterations that the coupled solver algorithm requires to solve a given flow problem is independent of mesh size, whereas the number of iterations that the segregated algorithm requires increases with the mesh size. Rayleigh number and Reynolds number play a significant role in choosing the type of solver to be used. The equations for the Rayleigh number and the Reynolds number are given by Eqns. (6) and (7). The Rayleigh number is better indicator of the natural forces that are happening as opposed to the forces from a pump that would be better described by the Reynolds number.

$$Ra_x = \frac{g\beta}{\upsilon\alpha}(T_s - T_\infty)x^3 \tag{6}$$

$$Re = \frac{\upsilon D}{\upsilon} \tag{7}$$

However, because the Star CCM+ input will be a constant velocity, the phenomena will be better described as a very low velocity forced flow scenario. For this reason, the Reynolds number will be more appropriate.

It is suggested to use coupled solver for more complicated natural convection flows to increase accuracy but this is unnecessary as the simplifications will reduce flow complexity. The modeling of the stainless steel thickness uses the same energy model but does not need a flow model as it is a solid for the entirety of the simulation. The only other model required for the feed water pertaining to the energy is the Segregated Fluid Enthalpy. This model adds the required relations for the heat transfer within the feed water and later the boiling models.

4.4.6Turbulence. Laminar flow in pipes normally occurs when the Reynolds number is 2200 or less, however, the size of the pipes and flow channels allow for the flow to be turbulent or in the transition range even though the velocity developed from natural convection is relatively small. The Reynolds number during the accident initiation would be in the transitional period between laminar and fully turbulent flow. But in the steady state analysis the flow is turbulent. The standard k-epsilon turbulence model was used for this simulation for its robustness and recommendation for natural convection flows. The Reynolds number calculation is shown for both the feed water and the reactor water in the Table 4.1below.

Table 4.1:Reynolds Number Calculations

	Reynolds #	Density (kg/m^3)	Velocity (m/s)	Hydraulic Diameter (m)	Viscosity(Pa s)
Fw	20,985	996.66	0.1	0.042632	0.00020248
Rw	12,752	666.98	0.15	0.01	0.00007845

5 METHODOLOGY

With the purpose of finding out how much heat is transferred by the passive heat removal system, a Star CCM+ simulation was used to evaluate the thermal hydraulics of natural circulation in the heat exchanger. The closed loop approach emulates the design seen in Figure 2.1. The feed water side of the heat exchanger is a closed loop that uses the ultimate heat sink to dump the heat gained from the passive heat removal system. The passive heat removal system is then modeled in Star CCM+ to evaluate how much heat might be transferred to the feed water. The models used for this simulation are listed in Table 5.1.

Table 5.1:Closed Loop Simulation Physics Models

Closed Loop Simulation Physics Models	
Reactor Side and Feed water Side	
Flow	Segregated Flow
Energy	Segregated Energy
State	Liquid
Viscous Regime	Turbulent
Optional	Gravity, Segregated Fluid Enthalpy

The boundary conditions and initial conditions are specified in the next table and are chosen based on the feasibility analysis.

Table 5.2:Closed Loop Simulation Boundary Conditions

Closed Loop Simulation Boundary Conditions		
Boundary		Boundary Condition
Reactor water (Rw) Side		
	Rw Inlet	Velocity Inlet
	Rw outlet	Flow split Outlet
	Rw side 1	Wall
	Rw side 3	Wall
	Note: an interface is created from Rw side 1 and 3.	
Feed water (Fw) side		
	Fw inlet	Velocity Inlet
	Fw outlet	Flow split Outlet
	Fw side 1	Symmetry
	Fw side 2	Symmetry
	Fw side 3	Symmetry
	Note: the stainless steel 316 boundaries and others not listed are all wall boundaries.	

Note: the Stainless Steel 316 boundaries and others not listed in this table are wall boundaries.

Table 5.3:Closed Loop Simulation Initial Conditions

Closed Loop Simulation Initial Conditions		
Boundary		Initial Condition
Reactor (Rw) Side		
	Rw Inlet	Velocity----------0.15 m/s
	Ref. Pressure	2250 psi
Feed water (Fw) side		
	Fw inlet	Velocity ---------0.1 m/s
	Ref. Pressure	47.02 psi

The closed loop design requires a single phase simulation because the steady state temperature does not reach the boiling temperature, 409.34 K. Figure 4.2 shows the geometry that is simulated in Star CCM+. The geometry, after the reductions by symmetry, is effectively one twelfth of a single pipe and the corresponding feed water channel that removes the heat from the reactor water. This means heat transferred in this simulation can be scaled to the number of pipes that share the same geometry. The properties used for the feed water and reactor water models are listed in the appendix.

6 RESULTS

6.1 CLOSED LOOP SIMULATION

The closed loop simulation yielded a heat transfer plot shown in Figure 6.1.

Figure 6.1:Heat Transfer to the Feed Water

The heat transferred in this simulation is converged at 1165W and the temperature converges at a feed water temperature of 385 K as seen in Figure 6.2. This temperature is below the boiling temperature of 409.34 K which keeps the simulation single phase. The 1165 W of heat is the simulated heat transfer for only one twelfth of the pipe, which means that a single reactor water pipe would transfer approximately 13980 W. In a design with 1 cm diameter pipes there would need to be 4045 pipes at a pitch of 0.025m. The total heat transfer from this design is then 56.55 MW.

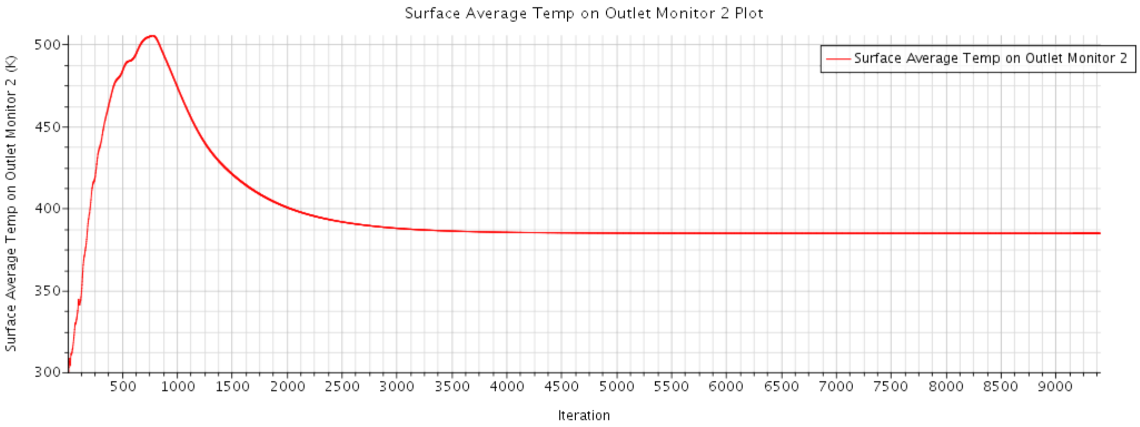

Figure 6.2:Surface Average Temp. at the Outlet of the Feed Water

The heat transfer result from this simulation allows for scaling of the passive heat exchanger to the needs of different small modular reactors. Designs that operate at lower thermal power can be scaled appropriately using this per-pipe heat transfer using standard temperature feed water that is easily accessible.

6.2 FINAL PASSIVE SAFETY SYSTEM DESIGN

The final design of the passive safety heat exchanger is just under the 2 meter constraint set in the system description section. However, this system could easily be split into divisions that would make smaller diameter modules that would have the same effective heat transfer. This is possible because the system was analyzed based on the individual pipe and flow area around the pipe in Star CCM+, which will stay the same regardless of the number of pipes in the system. Figure 6.4 is a depiction of the final passive heat removal systems’ hexagonal lattice. The actual number of pipes and dimensions of the design are described in Table 6.1.

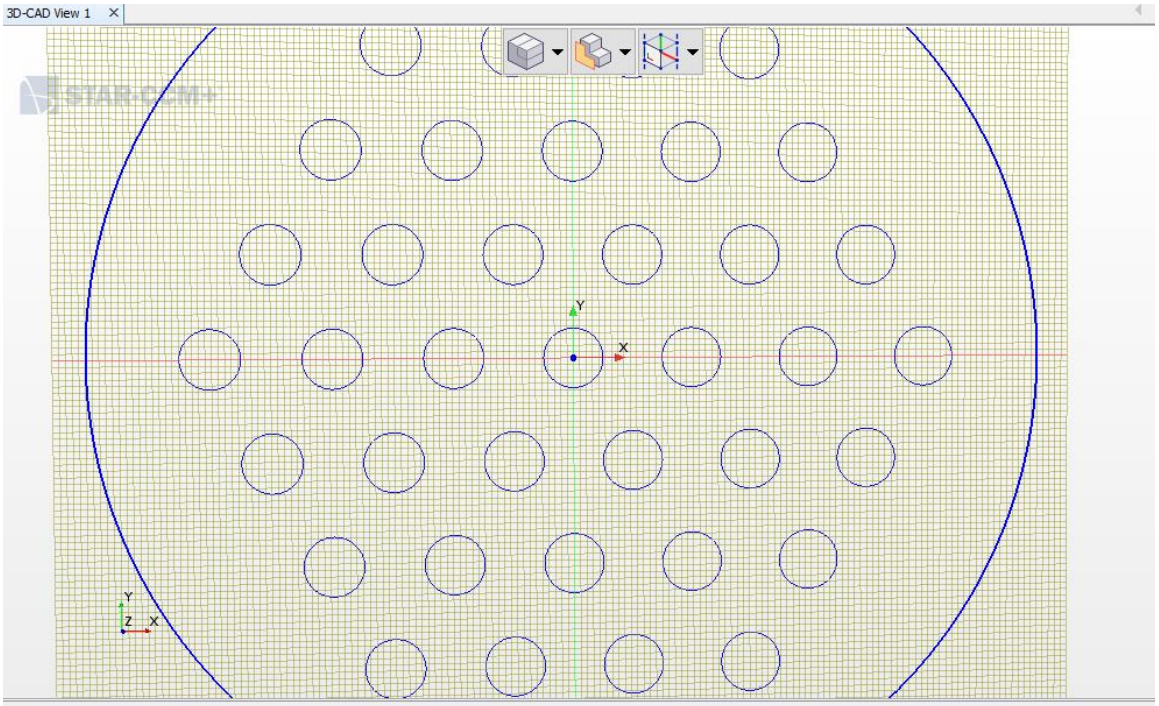

Figure 6.3:Hex Lattice Geometry Concept

Table 6.1:Final Design Parameters

Final Design Parameters	
Outer Diameter	1.93 m
Height	6.1 m
# of Pipes	4045
Inner Pipe Diameter	0.01 m
Outer Pipe Diameter	0.0125 m
Hex Lattice Pitch	0.025 m

7 CONCLUSIONS AND FUTURE WORK

7.1 CONCLUSIONS

It was shown through the first principle analysis that it is feasible to design a passive safety system that would remove the decay heat from a small modular reactor design using the Westinghouse-SMR as a benchmark for dimensional limitations.

The closed loop design analysis showed that a passive safety heat removal system with a closed feed water loop dissipated approximately 34.07MW which is only 60% of the intended 56MW. However, this design would still be appropriate for a small modular reactor that operates at 485MW (t).

The only way for this particular design to remove the total 56MW of decay heat was to increase the effectiveness of the heat exchanger. The final design of the passive heat removal system for a small modular reactor dissipates 13.98 kW per pipe and has 4045 pipes inside of a 1.93 m diameter heat exchanger that is 6.1 m in length. This design dissipates the entirety of the 56 MW of decay in a single unit but could easily be broken into equivalent divisions that make up the same number of pipes for heat transfer.

7.2 FUTURE WORK

The transition from shutdown to the steady state of the passive system should be investigated to evaluate the benefit or drawback of the operational mass flow rate being diverted to the heat exchanger. The outer channels that were not included in the Star CCM+ simulation should be evaluated to determine how much heat transfer they contribute to the passive safety system.

APPENDIX A

FLUID PROPERTIES

Reactor water at 2250 psi and 598K

ρ=666.98 kg/m^3, μ=7.8451E-05 Pa s, c_p=6.3979 kJ/kg K, h=1483.4kJ/kg,

k = 0.51027W/mK

Feed water at 47.02 psi and 300K

ρ=996.66 kg/m^3, μ=8.5379E-04 Pa s, c_p=4.18 kJ/kg K, h=112.86 kJ/kg,

k = 0.61042W/mK

APPENDIX B

NATURAL CONVECTION FLOW RATE DERIVATION

Derivation of the first principle analysis equation:

The pressure drop equation is calculated using the continuity and momentum equations for an incompressible ($\rho = constant$) fluid, in a pipe section,

Continuity: $\frac{\partial}{\partial t}\rho \quad + \nabla \rho v$ = 0, (8)

From incompressible: $\rho \frac{\partial v}{\partial z} = 0, \frac{\partial v^2}{\partial z}$

Momentum: $\frac{\partial}{\partial t}\rho v + \frac{\partial}{\partial z}\rho v^2 = -\frac{\partial P}{\partial z} - \frac{f_i}{D_i}\frac{\rho v^2}{2} + \rho g_z - \rho_o \beta \Delta T \;\; g_z$

From continuity, the second term of the momentum equation is equal to zero, then:

$$\rho \frac{\partial}{\partial t} v = -\frac{\partial P}{\partial z} - \frac{f_i}{D_i}\frac{\rho v^2}{2} + \rho g_z - \rho_o \beta \Delta T g_z$$

The pressure gradient $\frac{\partial P}{\partial z}$ is made up of the major and minor losses

$$\frac{\partial P}{\partial z} = \frac{\partial P}{\partial z_{major}} + \frac{\partial P}{\partial z_{minor}} = \frac{\rho v^2}{2}\left(\frac{f}{D} + \Sigma K_j\right)$$

Then,

$$\rho \frac{\partial v}{\partial t} = \frac{-\rho v^2}{2}\left(\frac{f}{D} + \Sigma K_j\right) - \frac{f}{D}\frac{\rho v^2}{2} + \rho g_z - \rho_o \beta \Delta T g_z$$

Integrate this equation along z, the ith terms are added for the individual components.

For a closed loop the integration of the pressure gradient becomes

$$\int \frac{\partial P}{\partial z} \partial z = \Delta P_{pump}, \text{ then } \int \frac{f_i \rho v^2}{2D_i} dz = \rho {v_r}^2 \sum_{i=1}^{n} \left(\frac{f_i L_i}{D_i} + \Sigma K_{ij}\right)\left(\frac{A_r}{A_i}\right)^2$$

Note: The mass flow rate continuity can be solved for a reference velocity,

$$\rho_i v_i A_i = \rho_r v_r A_r, \text{ then } v_r = \frac{\rho_i v_i A_i}{\rho_{r A_r}} \quad ,$$

The gravity and thermal expansion terms after integration become,

$$\int [\rho_i g_{zi} + \rho_o \beta \Delta T g_z] dz = \Sigma\left[\rho_r g_z \Delta H_i - \rho_r g_z \beta \Delta T_g \Delta H_i\right] = \rho_r g \beta \Delta T \Delta H_h$$

Modifying for natural convection means that the pump becomes a loss

$$\Delta P_{pump} = \frac{\rho {v_r}^2}{2} K_{pump}$$

which becomes part of the summation of the losses.

Then,

$$\rho_r \sum_{i=1}^{n} \frac{A_r}{A_i} L_i \frac{\partial v_r}{\partial t} \quad g\beta\Delta T\Delta H_h - \frac{\rho_r {v_r}^2}{2} \sum^{n} \left(\frac{f_i L_i}{D_i} + \Sigma K_{ij}\right)\left(\frac{A_r}{A_i}\right)^2$$

3 4

Eq. 1 2

Quasi-steady flow means that $\frac{\partial v_r}{\partial t}$=0, therefore

$$g\beta\Delta T_h \Delta H_h = \frac{\rho_r {v_r}^2}{2} \sum_{i=1}^{n} \left(\frac{f_i L_i}{D_i} + \Sigma K_{ij}\right)\left(\frac{A_r}{A_i}\right)^2$$

The temperature difference is a function of the heat being transferred to the system,

$$Q = mc_p \Delta T_h,\ \Delta T_h = \frac{Q}{mc_p} = \frac{Q}{\rho_r v_r A_r c_p}$$

Simplifying,

$$\frac{g\beta Q \Delta H_h}{v_r A_r c_p} = \frac{{v_r}^2}{2} \sum_{i=1}^{n} \left(\frac{f_i L_i}{D_i} + \Sigma K_{ij}\right)\left(\frac{A_r}{A_i}\right)^2$$

Solve for v_r,

$$v_{ref}^3 = \frac{2g\beta\Delta H}{\Sigma\left(\frac{f(Re)_i L_i}{D_i} + \Sigma k_{ij}\right)\left(\frac{A_r^2}{A_i^2}\right)} \frac{Q}{\rho_r c_p A_r} \tag{1}$$

REFERENCES

Ametek, 2016, "Heat Exchanger Tubes." *Shell and tube heat exchanger tubes for the cooling, heating or re-heating of fluids, gases, or air | Fine Tubes*. Ametek, n.d. Web. 4 Nov. 2016.

T. Batt, 1992, Nevada claims victory in Yucca deal, Las Vegas Rev. J. (1992), pp. 1A-3A.

Nayak and Sinha, 2007, A.K. Nayak, R.K. Sinha, Role of passive systems in advanced reactors, Progress in Nuclear Energy, Volume 49, Issue 6, August 2007, Pages 486-498, ISSN 0149-1970, http://dx.doi.org/10.1016/j.pnucene.2007.07.007.

IAEA, 1992, International Nuclear safety Advisory Group, The Safety of Nuclear Power, Safety Series No. 75-INSAG-5, (IAEA, Vienna 1992).

G. Angelo, D.A. Andrade, E. Angelo, W.M. Torres, G. Sabundjian, L.A. Macedo, A.F. Silva, A numerical and three-dimensional analysis of steady state rectangular natural circulation loop, Nuclear Engineering and Design, Volume 244, March 2012, Pages 61-72, ISSN 0029-5493, http://dx.doi.org/10.1016/j.nucengdes.2011.12.020.

IAEA, 2005, Natural Circulation in water cooled nuclear power plants, IAEA TECDOC-1474 (November, 2005).

IEAE, 2011 "Passive Safety and Advanced Nuclear Energy Systems." Risk and Safety Analysis of Nuclear Systems Lee/Risk (2011): 349-400. Passive Safety Systems and Natural Circulation in Water Cooled Nuclear Power Plants. IAEA. Web. 10 May 2016. <http://www-pub.iaea.org/MTCD/publications/PDF/te_1624_web.pdf>.

Butt and Ilyas, 2016, Hassan Nawaz Butt, Muhammad Ilyas, Masroor Ahmad, Fatih Aydogan, Assessment of passive safety system of a Small Modular Reactor (SMR), Annals of Nuclear Energy, Volume 98, December 2016, Pages 191-199, ISSN 0306-4549, http://dx.doi.org/10.1016/j.anucene.2016.07.018. (http://www.sciencedirect.com/science/article/pii/S0306454916305485).

Cd-Adapco, 2016, "STAR-CCM ® | CFD Simulation Software." STAR-CCM ® | CFD Simulation Software. Sharron L. MacDonald. Web. 10 May 2016. <http://www.cd-adapco.com/products/star-ccm®>.

Haskins and El-Genk, 2017, Denise A. Haskins, Mohamed S. El-Genk, Natural circulation thermal-hydraulics model and analyses of “SLIMM” – A small modular reactor, Annals of Nuclear Energy, Volume 101, March 2017, Pages 516-527, ISSN 0306-4549, http://dx.doi.org/10.1016/j.anucene.2016.11.036. (http://www.sciencedirect.com/science/article/pii/S0306454916308088).

Jun Liao, Vefa N. Kucukboyaci, Richard F. Wright, Development of a LOCA safety analysis evaluation model for the Westinghouse Small Modular Reactor, Annals of Nuclear Energy, Volume 98, December 2016, Pages 61-73, ISSN 0306-4549, http://dx.doi.org/10.1016/j.anucene.2016.07.023. (http://www.sciencedirect.com/science/article/pii/S0306454916302390).

Schultz, Ian. "Passive Nuclear Safety Technology." Passive Nuclear Safety Technology. Stanford, 3 Mar. 2012. Web. 10 May 2016. <http://large.stanford.edu/courses/2012/ph241/schultz1/>.

Shin and Choi, 2015, Yong-Hoon Shin, Sungyeol Choi, Jaehyun Cho, Ji Hyun Kim, Il Soon Hwang, Advanced passive design of small modular reactor cooled by heavy liquid metal natural circulation, Progress in Nuclear Energy, Volume 83, August 2015, Pages 433-442, ISSN 0149-1970, http://dx.doi.org/10.1016/j.pnucene.2015.01.002. (http://www.sciencedirect.com/science/article/pii/S0149197015000050).

Smith & Wright,(2012), M. C. Smith, R. F. Wright. *Westinghouse Small Modular Reactor passive safety system response to postulated events*. United States: American Nuclear Society - ANS.

Peric and Ferguson 2016, Peric, Milovan, and Stephen Ferguson. "The Advantage of Polyhedral Meshes - PLM MarketPlace." Accessed October 26, 2016. http://www.plmmarketplace.com/upload/Temp/The_Advantage_of_polyhedral_1.pdf.

Weinberg, 1985, Alvin M. Weinberg, Irving Spiewak, Doan L. Phung, Robert S. Livingston, The second nuclear ERA: A nuclear renaissance, Energy, Volume 10, Issue 5, 1985, Pages 661-680, ISSN 0360-5442, http://dx.doi.org/10.1016/0360-5442(85)90098-2.

SECTION

VITA

Raymond M. Fanning was born in Independence, Missouri. In May 2014, he received his B. S. in Nuclear Engineering from the Missouri University of Science and Technology, Rolla, Missouri. In May 2017, he received his M.S. in Nuclear Engineering from the Missouri University of Science and Technology, Rolla, Missouri.