Final Project Report on Computational Modeling and Analysis of Air-Based NSTF

Nuclear Engineering Division

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free via DOE's SciTech Connect (<http://www.osti.gov/scitech/>)

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):

U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd Alexandria,
VA 22312 **www.ntis.gov**
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

Final Project Report on Computational Modeling and Analysis of Air-Based NSTF

prepared by
R. Hu, A. Kraus, M. Bucknor, Q. Lv, and D. Lisowski
Nuclear Engineering Division, Argonne National Laboratory

June 2016

EXECUTIVE SUMMARY

The Reactor Cavity Cooling System (RCCS) is a passive safety concept under consideration for the overall safety strategy of advanced reactors such as the High Temperature Gas-Cooled Reactor (HTGR). The air-cooled RCCS uses natural convection to drive the flow of air from outside the power plant to remove decay heat during normal operation and accident scenarios. The Natural convection Shutdown heat removal Test Facility (NSTF) at Argonne National Laboratory (“Argonne”) is a ½-scale model of the primary features of one conceptual air-cooled RCCS design. The NSTF program performs ex-vessel, passive heat removal experiments in support of Department of Energy Office of Nuclear Energy’s Advanced Reactor Technology (ART) program.

This report documents the progress and achievements made in the analysis and computational modeling portion of the NSTF program at Argonne. The work performed has supported the development of modeling practices, design optimization, examination of heat removal performance, and understanding of physical behavior. Both system-level and high-fidelity computational fluid dynamics (CFD) codes were utilized to gain an understanding of the complex flow and heat transfer characteristics of these natural convection systems.

The objectives of the NSTF analyses further included assessment of limitations in typical modeling approaches of RCCS-type natural circulation systems, as well as evaluate the experimental procedures and generated data. Improvements in instrumentation, procedures, and facility design were identified through the analysis effort, including a number of new sensors that were added to facilitate code comparison and improved facility characterization. These measurements included thermal images and temperatures for calculating the facility heat loss, stratified building temperatures, local flow velocities, and distributed temperature measurements. The collaboration between experimentalists and computational analysts has been vital for improving the experimental program, driving model development, and obtaining an overall understanding of system behavior.

Throughout the testing program, strong influences of ambient conditions were observed in the experimental data when baseline tests were repeated with the same test procedures. Thus, significant analysis efforts were devoted to gaining a better understanding of these influences and the subsequent response of the NSTF. It was determined that air humidity had negligible impacts on NSTF system behavior and therefore did not warrant consideration in the models. However, temperature differences between the building exterior and interior air, along with the outside wind speed, were shown to be dominant factors. An analytical model was developed using experimental data to correlate zero-power system flow rates with ambient meteorological conditions. The analytical model was then implemented in the computational models of the NSTF, and strengthened the ability of analysis to accurately model the observed physical behavior.

To examine the specific transient and integral behavior of the NSTF, a RELAP5-3D model was developed. Several modifications were made in FY16 to accurately capture the effects of outdoor air temperature, wind, building pressure, and the addition of new chimney caps on the performance of the facility. Then, experimental data from a single test was used to calibrate the RELAP5-3D model. The model was ultimately used to simulate additional experimental tests conducted during different seasons of the year. These simulation results

demonstrated the ability of the code to capture the effects of ambient conditions on the performance of the facility over a wide range of conditions (outdoor temperatures ranging from -22.3°C to 33.4°C and wind speeds ranging from 0 m/s to 8.5 m/s). The simulation results compared well with experimental data for all of the studied test cases for the overall system natural circulation flow rate, with average absolute errors less than 5% in simulations where these ambient effects were included. The ability of the model to capture these effects and their influence on the integral behavior provided confidence that the natural circulation behavior of the NSTF could be well replicated by RELAP5-3D simulations, and its usage could provide insights into the performance of the RCCS under various system conditions.

The CFD analysis of the NSTF in STAR-CCM+ progressed significantly since the program's inception. Prior work has shown that reasonable estimates of the NSTF performance could be obtained with moderate computational cost. Building upon earlier work, the first in-depth studies of refined wall meshes and turbulence models were undertaken. These took full advantage of the distributed temperature sensing (DTS) fibers installed in the riser ducts in order to refine computational models, as well as forced flow test data for more controlled boundary conditions. Turbulence and meshing studies demonstrated that some of the models used in prior work were too diffusive (i.e. high numerical diffusion) and their results under-predicted duct wall temperatures. A comparison study of several turbulence and mesh models was performed, and a select number of models were included for further analysis given the level of accuracy desired and computational time available. The Wolfstein model with high- y^+ mesh in the primary flow path was found to be sufficient for predicting the bulk behavior of the system, while refinements in the cavity and ducts improved local temperature predictions, notably in the riser air temperature profiles.

The extension of modeling to the natural circulation cases more rigorously confirmed that the NSTF could be well simulated by CFD methods. This study was *a priori* test, given that only the system geometry, wall heat flux, and indoor and outdoor temperatures were needed. The predicted mass flow rates for the low-power test were lower than the experimental data by 9%, but could be reduced to only a 2% difference when accounting for the effects of ambient conditions. For the high-power test case, the difference in predicted mass flow rate was further reduced, and found to be within the uncertainty of experimental measurements. Finally, CFD simulation results corroborated physical observations that changes in the system flow rate have only minor impacts on the heated surface (RPV) temperatures.

In conclusion, computational modeling of the NSTF was performed at Argonne using both 1D system-level and 3D CFD simulations and compared against experimental data generated from the testing program. It was demonstrated that the natural circulation airflow and the coupled radiation-convection-conduction heat transfer could be accurately modeled using the RELAP5-3D and STAR-CCM+ codes when appropriate boundary conditions are applied. These boundary conditions included not only traditional operator driven parameters, e.g. supplied power and geometric configuration, but also meteorological and ambient conditions. The ability of both codes to match experimental data across a range of testing conditions suggests a high level of confidence in the codes capabilities, and their use is strongly encouraged for future endeavors of passive safety concepts. Ultimately, the collaboration between experimental and analytical portions of the NSTF program proved to be a mutually beneficial relationship that supported the DOE vision of examining the heat removal performance of the RCCS concept.

Table of Contents

EXECUTIVE SUMMARY	I
TABLE OF CONTENTS.....	III
LIST OF FIGURES	V
LIST OF TABLES	VII
1 INTRODUCTION	1
REFERENCES	4
2 EXPERIMENT PROGRAM SUPPORT	5
2.1 ANALYSIS SUPPORT FOR ADDITIONAL INSTRUMENTATIONS	5
2.2 FACILITY IMPROVEMENT	6
2.3 NSTF TEST SUPPORT	9
REFERENCES	13
3 SYSTEM CHARACTERIZATION AND LESSONS LEARNED FOR NSTF MODELING	14
3.1 AIR HUMIDITY EFFECTS.....	14
3.2 AMBIENT CONDITION EFFECTS	16
3.2.1 Stack- and Wind-Effect Pressures	19
3.2.2 An Empirical Model of Stack and Wind Effects in NSTF	20
3.3 BUILDING TEMPERATURE EFFECTS	24
3.4 HEAT LOSS CHARACTERIZATION OF NSTF EXPERIMENTS.....	25
REFERENCES	26
4 RELAP5-3D SYSTEM ANALYSIS OF NSTF	28
4.1 MODEL DESCRIPTION.....	28
4.2 MODEL DEVELOPMENT AND VERIFICATION.....	30
4.2.1 Outdoor Air Temperature Effects	30
4.2.2 Building Form Loss Effects	30
4.2.3 Wind Effects	31
4.2.4 The Addition of New Chimney Caps.....	31
4.2.5 Model Input Power	31
4.2.6 Flow Instabilities at Low Flow Conditions.....	32
4.3 MODEL INPUTS AND “TUNING” TO RUN022.....	33
4.4 BASE CASE TESTS PROCEDURES	38
4.5 ADDITIONAL BASE CASE TEST SIMULATIONS.....	38
4.5.1 Run011	39
4.5.2 Run020	40
4.5.3 Run024	40
4.5.4 Importance of Capturing Ambient Effects in Base Case Simulations	41
4.6 SUMMARY OF RELAP5-3D ANALYSES	48
REFERENCES	48
5 AIR-COOLED NSTF CFD ANALYSES.....	49
5.1 CFD MODEL DESCRIPTION	49
5.1.1 Boundary Conditions and General Details	49

5.1.2	Turbulence Modeling.....	51
5.1.3	Meshing.....	52
5.2	FORCED FLOW RESULTS AND DISCUSSION.....	53
5.2.1	Point Measurement Comparison.....	53
5.2.2	LUNA Fiber and 3D Comparison.....	63
5.2.3	Conclusions from Turbulence Modeling and Mesh Studies.....	70
5.3	FULL NATURAL CONVECTION SIMULATIONS.....	71
5.3.1	Base Case.....	71
5.3.2	Low Power Case.....	75
5.4	BUILDING PRESSURE AND WEATHER EFFECTS.....	79
5.5	SUMMARY OF FINDINGS AND ACCOMPLISHMENTS FROM CFD MODELING.....	80
	REFERENCES.....	80
6	CONCLUSIONS.....	82
	ACKNOWLEDGEMENTS.....	83

LIST OF FIGURES

Figure 1-1: NSTF Layout. Left: solid model rendering; Right: primary segments, A. inlet downcomer, B. inlet plenum, C. heated cavity, D. riser ducts, E. outlet plenum, F. chimney ...	3
Figure 2-1: Temperature profile across 10" width of riser exit face as measured by the DTS fibers (left facing the heated wall), steady-state average at baseline testing conditions.....	6
Figure 2-2: Gap between butterfly valve and inner ductwork. Average gap across entire circumference of 0.125"	8
Figure 2-3: GA MHTGR Reactor Vessel and RCCS	10
Figure 2-4: Partial CFD Model of MHTGR Cavity.....	11
Figure 2-5: Schematic for Air-Cooled RCCS behind Cross Ducts.....	11
Figure 2-6: Riser ducts with high view factors in MHTGR Cavity.....	12
Figure 2-7: Riser ducts view factors from the heated surface in NSTF Cavity.....	13
Figure 3-1: The moist air density as a function of temperature with the relative humidity	15
Figure 3-2: The relative humidity as a function of temperature with inlet air at 80% RH and 20 °C	15
Figure 3-3: The effects of humidity to system flow rate, RELAP5 simulations of Run020	16
Figure 3-4: Repeatability of NSTF baseline tests	18
Figure 3-5: Stack effects on pressure and flow. A: (stack pressure head), B: (pressure difference), C: air infiltration mass flow.....	19
Figure 3-6: Wind effects to air infiltration in Building 308 and NSTF system flow.....	20
Figure 3-7: Measured flow rates and wind speed in SPEF005	22
Figure 3-8: Response surface of fitting correlation	23
Figure 3-9: Fitting correlation applied to SPEF005.....	23
Figure 3-10: Fitting correlation applied to SPEF007.....	24
Figure 3-11: Relationship between density and temperature for air.....	25
Figure 3-12: Thermal efficiency of the NSTF, determined by separate effects testing at varying electric powers and flow rates	26
Figure 4-1: RELAP5-3D nodalization diagram of the NSTF.....	29
Figure 4-2: Example comparison of model input power estimation methods	32
Figure 4-3: Example comparison of mass flow rate results using different input power methods.....	32
Figure 4-4: Run022: RELAP5-3D model inputs estimated from experimental data.....	36
Figure 4-5: Run022: Comparison of RELAP5-3D simulation results and experimental data	37
Figure 4-6: Installation locations of pitot tubes in the chimney ducts.....	38
Figure 4-7: Run011: RELAP5-3D model inputs estimated from experimental data.....	42
Figure 4-8: Run011: Comparison of RELAP5-3D simulation results and experimental data	43
Figure 4-9: Run020: RELAP5-3D model inputs estimated from experimental data.....	44
Figure 4-10: Run020: Comparison of RELAP5-3D simulation results and experimental data.....	45
Figure 4-11: Run024: RELAP5-3D model inputs estimated from experimental data.....	46
Figure 4-12: Run024: Comparison of RELAP5-3D simulation results and experimental data.....	47
Figure 5-1: Left: Locations of axial DTS fibers, at "GF" points vertically within the flow. Right: Example duct exit DTS fiber.	50
Figure 5-2: Computational domain (L) and zoomed view of the outlet plenum (R).	52
Figure 5-3: Mesh examples for the low-y+ cavity high-y+ main (L) and high-y+ cavity low-y+ main (R) cases	53

Figure 5-4: TC locations for the heated cavity wall	55
Figure 5-5: TC locations for the cavity side wall	56
Figure 5-6: Heated surface temperatures for the Two-Layer and SST models	57
Figure 5-7: Heated surface temperatures for the Standard and EB models	57
Figure 5-8: Cavity side wall temperatures for the Two-Layer and SST models	58
Figure 5-9: Cavity side wall temperatures for the Standard and EB models	58
Figure 5-10: Duct outlet TC temperatures for the Two-Layer and SST models	59
Figure 5-11: Duct outlet TC temperatures for the Standard and EB models	59
Figure 5-12: Duct 9 outer surface TC locations	60
Figure 5-13: Duct 9 outer surface temperatures for the Two-Layer and SST models	60
Figure 5-14: Duct 9 outer surface temperatures for the Standard and EB models	61
Figure 5-15: Duct pressure drops for the Two-Layer and SST models	62
Figure 5-16: Duct pressure drops for the Standard and EB models	62
Figure 5-17: Axial fiber temperature comparison for the high-y+ mesh models. Solid lines are for fiber GF_E; dashed lines are for fiber GF_C	64
Figure 5-18: Axial fiber temperature comparison for models with main low-y+ meshes. Solid lines are for fiber GF-E; dashed lines are for fiber GF-C	64
Figure 5-19: Axial fiber temperatures for models with cavity low-y+ meshes. Solid lines are for fiber GF-E, dashed lines are for fiber GF-C	65
Figure 5-20: Duct 9 exit fiber temperatures for the high-y+ mesh models	65
Figure 5-21: Duct 9 exit fiber temperatures for models with main low-y+ meshes	66
Figure 5-22: Duct 9 exit fiber temperatures for models with cavity low-y+ meshes	66
Figure 5-23: Comparison of mesh and model approaches for Duct 9 outlet temperatures	67
Figure 5-24: Duct 9 riser outlet velocity comparison.	68
Figure 5-25: Velocity distribution at the riser axial midplane for the MhighWolf_ChighXu model (left) and the MlowEB_ClowEB model (right)	69
Figure 5-26: Temperature distribution at the riser axial midplane for the MhighWolf_ChighXu model (left) and the MlowEB_ClowEB model (right)	69
Figure 5-27: Axial velocity and temperature distributions at the Duct 9 midplane for the MhighWolf_ChighXu model (left) and the MlowEB_ClowEB model (right). Plots are 1/5 axial scale.	70
Figure 5-28: Heated surface temperatures for the Run22 high power case	72
Figure 5-29: Cavity side wall temperatures for the Run22 high power case	72
Figure 5-30: Duct outlet TC temperatures for the Run22 high power case	73
Figure 5-31: Duct pressure drops for the Run22 high power case	73
Figure 5-32: Axial fiber temperatures for the Run22 high power case. Solid lines are for fiber GF-E, dashed lines are for fiber GF-C	74
Figure 5-33: Duct 9 exit fiber temperatures for the Run22 high power case	74
Figure 5-34: Heated surface temperatures for the Run22 low power case	75
Figure 5-35: Cavity side wall temperatures for the Run22 low power case	76
Figure 5-36: Duct outlet TC temperatures for the Run22 low power case	76
Figure 5-37: Duct 9 outer surface temperatures for the Run22 low power case	77
Figure 5-38: Duct pressure drops for the Run22 low power case	77
Figure 5-39: Axial fiber temperatures for the Run22 low power case. Solid lines are for fiber GF-E, dashed lines are for fiber GF-C	78
Figure 5-40: Duct 9 exit fiber temperatures for the Run22 low power case	78

LIST OF TABLES

Table 2-1: Summary of major improvements to test facility resulting from modeling observations	7
Table 2-2: Influence on system flow rate due to varying levels of air ingress past damper valve	9
Table 2-3: Measured (original) and corrected values sample baseline test case	9
Table 3-1: Overview of repeatability testing at baseline conditions	17
Table 3-2: Span of available data collected at zero power conditions	21
Table 4-1: Measurement uncertainties of primary sensors in test facility	34
Table 4-2: Approximate range of RELAP5-3D inputs values for each base case simulation ..	39
Table 5-1: Database of all turbulence model and mesh refinement combinations used in the forced-flow modeling studies	54
Table 5-2: Mass Flow rate comparisons for natural convection simulation cases.	79

1 Introduction

The Reactor Cavity Cooling System (RCCS) is one of the primary systems under consideration for passive safety of the Very High Temperature Reactor (VHTR)—a gas-cooled nuclear plant with a thermal-spectrum reactor. The RCCS forms a passive heat sink external to the reactor, and is located within the reactor cavity/silo surrounding the reactor vessel. During a postulated accident when all AC (alternating current) and DC (direct current) power is lost, natural convection of the RCCS coolant removes the heat radiated or naturally convected from the reactor vessel wall to the inner boundary of the RCCS facing the vessel. RCCS will be a first-of-a-kind (FOAK) passive safety system for core decay heat removal when the FOAK VHTR is built [1-1].

The Natural convection Shutdown heat removal Test Facility (NSTF) [1-1][1-2][1-3] is a large-scale thermal hydraulics test facility at Argonne National Laboratory (“Argonne”). The facility was constructed to carry out highly instrumented experiments to study the performance of the RCCS concept for reactor decay heat removal that rely on natural convection cooling with either air- or water-based systems. It is supported by the Department of Energy (DOE) as part of the Advanced Reactor Technologies (ART) program.

The general facility layout is provided in Figure 1-1 [1-3]. Air is first drawn through the downcomer, where sensors are installed to characterize this inlet boundary condition with measurement of mass flow rate, humidity, and gas temperature, among others. The air then enters the large volume inlet plenum, where it splits and enters twelve riser ducts. Within these channels, heat is transferred from the riser walls to the incoming air and due to density driven buoyancy differences, allows a natural circulation driven flow to be established. The heat source originates from a 1-inch thick steel plate, which mimics the walls of a reactor pressure vessel (RPV) in both surface texture and emissivity. Behind this plate, 220 electric heaters, positioned in an 10×4 array, allow precise shaping and control of integral power. The heated air within the risers continues to rise in elevation until discharging into the outlet plenum. Within this volume, the heated air from all twelve channels mixes and again splits among the two chimney ports, before finally continuing to rise in elevation and ultimately being rejected to the outside atmosphere.

Construction of the test section was completed in September 2013. After completing the facility shakedown tests and checkout activities, the data collection started in early 2014, and a large number of tests have been performed. The interested reader may refer to earlier Argonne reports for documentation on the scaling studies and preparation tasks [1-1][1-2], the engineering specifications, as-built dimensions, and details of the data acquisition suite [1-3], as well as the preliminary results and findings from the NSTF tests [1-4].

Parallel modeling and simulation efforts were revitalized in FY15 [1-5] to support the design, operation, and analysis of the natural convection systems. Updates to prior models [1-2][1-6] were made to reflect changes in the final “as-tested” design. The primary objective of the NSTF analyses is to assess the limitations in typical approaches for modeling this type of natural circulation RCCS systems, and validate the analysis methods and computer codes which can be used in licensing support. Both system-level thermal-hydraulics (STH) and computational fluid dynamics (CFD) codes were utilized in this effort. RELAP5-3D [1-7] and STAR-CCM+ [1-8] code analyses are performed to gain a complete understanding of the complex flow and heat

transfer phenomena in natural convections systems. Additionally, the NSTF analyses aid in the RCCS design optimization, and supporting experiment activities, i.e. helping assure that the experimental procedures, setup, and measurements were performed as planned.

Throughout the course of model development and code benchmark, the computational effort evolved to strengthen the experimental program. This mutually beneficial relationship has become integral to the overall program objective of examining the heat removal performance of the RCCS concept.

This report documents the progress and achievements made in the computational analyses of NSTF tests. Both system-level and high-fidelity CFD analyses of NSTF tests are performed. The main project tasks include:

1. Analysis support for facility and instrumentation improvements;
2. Understanding the physics phenomena in the tests and investigating the impacts of ambient conditions;
3. Performing code benchmark simulations of NSTF tests using both RELAP5 and STAR-CCM+ codes.

A number of opportunities for fine-tuning the test facility design and operation were identified through the NSTF analyses support, and these improvements benefited both the experimental program and computational modeling efforts. A large number of new measurements were added to the facility for code comparison and better facility characterization, including measurements for calculating facility heat loss, stratified building temperatures, turbulence intensities of key flow regions, and additional distributed temperature measurements. Several improvements were also made to the physical construction of the test facility, based on the result of comparisons between experimental and simulation results. This work is detailed in Chapter 2.

Significant efforts were devoted to better understand the effects of weather on NSTF system flow behavior. Strong influences of ambient conditions have been observed in the experimental results, when the baseline test was repeated several times with the same test conditions, yet the response of the system varied among the tests. The effects of air humidity, the building outside and inside temperature, and the wind speed were investigated. Based on theoretical derivations and engineering judgments, an analytical model was developed to correlate the zero-power system flow rates and the ambient conditions. Some constant coefficients were obtained empirically by best fitting the experimental data. This work is detailed in Chapter 3.

The primary focus of the RELAP5-3D modeling was to analyze the integral system performance of the facility. The transient behavior of such a large system would be difficult to analyze with CFD due to the computational time required to perform such an analysis, but is readily investigated utilizing a system-level code. In FY16, several modifications were made to the RELAP5-3D model of the NSTF to properly capture the effects of outdoor air temperature, wind, building pressure, and chimney caps on the performance of the facility. Experimental data from a single test was used to calibrate the RELAP5-3D model. The model was then used to simulate three similar experimental tests, conducted during different seasons of the year, to confirm model performance over a wide range of outdoor air temperatures and wind speeds. The overall behavior of the facility was captured well by the model in all of the scenarios simulated. This work is detailed in Chapter 4.

Steady-state Reynolds-Averaged Navier-Stokes (RANS)-based CFD simulations of natural convection and radiation heat transport were performed using the STAR-CCM+ code to support the NSTF design and the instrumentation layout, as well as to analyze the detailed multi-dimensional flow and heat transfer in the NSTF, such as the radiation heat transfer and natural convection in the cavity, non-uniform flow split among all the risers, and large-scale flow structures in mixing plena and chimney structures. Significant advancements have also been made regarding the CFD analysis of the NSTF. The first in-depth studies of refined wall meshes and turbulence models have been undertaken. These take full advantage of the distributed temperature sensing (DTS) fibers installed in the riser ducts in order to aid in the validation of computational models, as well as forced flow experimental data for more reliably controlled boundary conditions. The turbulence and meshing studies showed that some of the models used in prior work were actually too diffusive, and under-predicted the duct wall temperatures. Lessons from the turbulence and meshing studies were then applied to simulating two full natural convection cases, one at high power (the same as that of the modeling studies) and one at low power. The effects of ambient conditions were also modeled. This work is detailed in Chapter 5.

Figure 1-1: NSTF Layout. Left: solid model rendering; Right: primary segments, A. inlet downcomer, B. inlet plenum, C. heated cavity, D. riser ducts, E. outlet plenum, F. chimney

References

- [1-1] S. Lomperski, W. D. Pointer, C. P. Tzanos, et al., "RCCS Studies and NSTF Preparation Air-Cooled Option," ANL-GenIV-142, Argonne National Laboratory, September 2010.
- [1-2] S. Lomperski, W. D. Pointer, C. P. Tzanos, et al., "Generation IV Nuclear Energy System Initiative: Air-Cooled Option RCCS Studies and NSTF Preparation", ANL-GenIV-179, Argonne National Laboratory, September 2011.
- [1-3] D. Lisowski, M. Farmer, et al., "Design Report for the 1/2 Scale Air-Cooled RCCS Tests in the Natural convection Shutdown Heat Removal Test Facility (NSTF)", ANL-SMR-8, Argonne National Laboratory, June 2014.
- [1-4] D. Lisowski, C. Gerardi, D. Kilsdonk, et al., Argonne National Laboratory, unpublished information, September 2015.
- [1-5] R. Hu, A. Kraus, M. Bucknor, et al., Argonne National Laboratory, unpublished information, September 2015.
- [1-6] R. Hu and W. D. Pointer, "CFD Analyses of Natural Circulation in the Air-Cooled Reactor Cavity Cooling System," *Proceedings of M&C 2013*, Sun Valley, ID, May 5-9, 2013.
- [1-7] The RELAP5-3D Code Development Team, "RELAP5-3D Code Manual," Idaho National Laboratory, Idaho Falls, ID, INEEL-EXT-98-00834, Rev. 4, 2012.
- [1-8] CD-Adapco, "STAR-CCM+ 9.04 Manual," Melville, NY 2014.

2 Experiment Program Support

To support the experimental program, computational modeling was performed using both 1D system-level and 3D CFD codes. Throughout the course of computational model developments, the computational effort has evolved to also play a major role in strengthening the experimental program, including improvements on the instrumentation, test procedures, and the physical construction of the test facility. This mutually beneficial relationship has become integral to the overall program objective of examining the heat removal performance of a RCCS concept.

2.1 Analysis Support for Additional Instrumentations

A large number of new measurements were added to the facility for code comparison and better facility characterization, including measurements for characterizing facility heat loss, stratified building temperatures, turbulence intensities of key flow regions, and distributed temperature measurements in key locations. The inclusion of high fidelity instruments was intended to not only supplement existing sensors, e.g. thermocouples, flow meters, but also to provide new information on the local details of thermal-hydraulic phenomena. One such example was the installation of the DTS fibers across the 10" exit faces of the riser ducts. Previous measurements at this location had relied only on a 1/16" Type-K thermocouple positioned at the centerline of the duct, and were the basis for early thermal power calculations. However, with the installation of the fiber, a temperature gradient was observed which contradicted the use of a single point measurement for comparison with localized CFD models. The average temperature obtained using the DTS data was used in the RELAP5-3D model as well. The average temperature, as determined by the fibers, was found to average 106 – 112% of the thermocouple measured at the riser outlet centerline. The identification of the significantly non-uniform duct temperature seen in Figure 2-1 was confirmed with the CFD results. The fact that the experimental results match very well with the fiber results (Section 5.2) gives additional evidence that the calibration methods and implementation of the fibers are correct.

Computational analyses can also aid in uncovering potentially defective or “underperforming” measuring instruments. As an example, the shiny heat flux measurements (HFMs) in the cavity are intended to reflect most radiation and only measure the convective portion of the heat flux. With the help of computational analyses, it was found that the shiny HFMs were likely severely overestimating the convective heat flux, whether due to physical degradation, tarnishing over time, factory defects, or other reasons. This prompted rigorous inspections of the related data acquisition channels and sensor installations at NSTF, and inquiries with the manufacturer on the measurement techniques and the calibration. The current disposition will be to re-calibrate these sensors upon removal of the riser ducts, both in-house and through blind calibration by the original vendor. Another example is on the differential pressure measurement, in which impulse plumbing is used to connect the ‘high’ and ‘low’ ports to different physical segments on a pipe. It was found that, to obtain a common reference for modeling comparisons, experimental measurements should be adjusted to include the gravitational head of the impulse plumbing. Thermocouples were also added along the differential pressure impulse tubing to accurately measure the added gravitational head.

Figure 2-1: Temperature profile across 10" width of riser exit face as measured by the DTS fibers (left facing the heated wall), steady-state average at baseline testing conditions

2.2 Facility Improvement

Over the course of the experimental testing program, a number of improvements were made to the physical construction of the test facility. Several of these were the result of observations made by the analytical team during review of previous data, and ultimately proved valuable in strengthening the overall impact of the experimental program. A summary of these improvements is provided below in Table 2-1.

Table 2-1: Summary of major improvements to test facility resulting from modeling observations

Improvement	Driving Observation	Description	Date Implemented
Matte stripes painted along outside of insulation panels	Characterization of the facility heat losses would benefit from exterior wall temperatures, however the shiny aluminum provided false IR readings	A high temperature paint was used to mark areas of the insulation panels to allow accurate thermal imaging	04/25/2014
New mounting of LUNA fibers within riser gas	The early iteration of SuperDuct, containing five fibers within the gas space, utilized positioning studs that unintendedly conducted heat causing temperature spikes	The 2 nd iteration of SuperDuct featured spring-loaded mounting studs that only support the fiber at the inlet and exit of the riser	09/15/2014
Thermocouples installed along chimney above roofline	Quantification of the heat loss along the chimney network and outside of the building was needed to improve boundary conditions for the predictions	New thermocouples were installed along the duct centerline and wall of the chimney above the roof	02/27/2015
Additional outlet plenum insulation	A selected number of wall thermocouples (TC) within the outlet plenum were reading abnormally cool temperatures. These were determined to reside near the joint of the outlet plenum cavity and false wall partition	Gas-sealing insulation strips were added to the outlet plenum wall joints to reduce hot-air exchange between false wall partition and outlet cavity	07/21/2015
Differential pressure impulse tubing temperature measurement	To obtain a common reference for modeling comparisons, experimental measurements should include the gravitational head of the impulse plumbing	Thermocouples were added along the differential pressure impulse tubing to accurately measure the added gravitational head	10/07/2015
Bldg. 308 ambient temperature measurement	Need for improved accuracy in calculating parasitic heat loss to environment	Additional thermocouples were added at multiple elevations within the Bldg. 308 high bay to measure ambient temperatures	10/07/2015
Anti-down draft chimney caps	Prevent performance degrading, wind-induced, flow reversals during heated operation	New chimney caps were installed to mitigate weather-induced down-drafts	11/24/2015
Sealed loft damper valves	Comparisons against RELAP5-3D suggested temperature loss in measurements abnormal	Fan loft ductwork was fully sealed to prevent air by-pass	01/11/2016
Split of LUNA riser outlet	Measurement points near end points of fiber were excessively noisy due to long fiber length	LUNA fiber was split into two separate sensors to improve data clarity	04/05/2016

Of these improvements, the sealed loft damper valves, which caused leakage of air past the fan loft damper valves, was a significant finding and addressed within the programs Quality Assurance Plan (QAP), ANL-NSTF-000000-DAR-005-R0. The chimney damper (butterfly) valves used to close the flow paths along the horizontal loft segments allow a portion of fresh air ingress due to a 0.125-in gap between the valve disk and inside duct walls, as shown in Figure 2-2. Across the 23.75-in diameter of the duct, this gap areas sums to 9.275-in² per chimney, or an area equivalent to a circle 3.437-in in diameter.

Figure 2-2: Gap between butterfly valve and inner ductwork. Average gap across entire circumference of 0.125"

Heat loss along the length of the chimney stack was identified as being higher than expected by CFD and RELAP5 models. Experimental losses averaged 10 – 15 °C from the upper plenum outlet to the chimney exit, while modeling results suggested a temperature drop closer to 5°C. During a separate effects study, the horizontal fan lofts were physically sealed from the outside and a baseline test was performed, resulting in a chimney temperature drop that agreed with the models, ~ 5°C.

A study was performed to quantify the amount of air leakage/ingress at the fan loft damper, which consisted of returning the duct loft to the original configuration and performing volumetric flow measurements of the leakages. This was supplemented by RELAP5 simulations with a parametric sensitivity study on the air-ingress flow rate, which allow high confidence in quantifying the impact of the leak on previous data.

The effect of air ingress on system flow rate can be determined by RELAP5 simulations of the NSTF. This model allowed varying the amount of fresh air ingress at the chimney ducts. The influences on the system flow rate at varying leakage rates are shown below in Table 2-2.

Table 2-2: Influence on system flow rate due to varying levels of air ingress past damper valve

Normalized Air-Ingress Flow	System Flow (kg/s)	Deviation
0%	0.627	0.00%
5%	0.621	0.96%
10%	0.615	1.91%
15%	0.610	2.71%

Based on these two simple analyses above, along with the modeled system flow rate influence, the air ingress flow rate can be calculated and to correct measured values from the data sets with influence of the chimney leakage. A sample calculation is provided below in Table 2-3 and is representative of a typical cold weather baseline test case ($Q_{full} = 1.5 \text{ MW}_t$, $Q_{NSTF} = 56 \text{ kW}_t$).

Table 2-3: Measured (original) and corrected values sample baseline test case

	As-measured	Corrected
System Flow Rate, kg/s	0.574	0.583
Chimney ΔT , °C	-12.21	-5.70
Leakage Flow Rate, kg/s	0.042	0.00

2.3 NSTF Test Support

Computational modeling and simulation is also very valuable in defining the test configurations, boundary conditions, and procedures in the air-based NSTF experiments. An example is given here to determine the azimuthal power distribution in one NSTF test to mimic the power skew in the full-scale RCCS, in which the non-uniformity in the temperature and power distributions of the full-scale reactor vessel wall is expected.

The full-scale design for the NSTF is based off the General Atomics (GA) Modular High Temperature Gas Reactor (MHTGR). Within the cavity, Figure 2-3a shows an isometric view of the reactor vessel surrounded by the hot riser ducts [2-1]. The upper cold plenum can be seen feeding the cold down-comer all the way to the bottom cold plenum. The cold air then rises upward through the hot risers removing the reactor vessel wall heat to the upper hot plenum. The upper hot plenum can be seen surrounding the vessel upper head. A plane cross-section view of the reactor cavity is shown in Figure 2-3b. The reactor vessel is shown surrounded by the distributed hot risers. Three reactor supports are displayed and the primary cross-vessel is also shown. The down-comer channels form the outer cavity wall.

Figure 2-3: GA MHTGR Reactor Vessel and RCCS

Analyses of the radiation heat transfer view factors in both MHTGR and NSTF cavities have been performed with the CFD software STAR-CCM+, which includes a ray-tracing model to calculate the surface-to-surface view factors. As the objective is to find the peak view factor per riser duct, all the surfaces (front, side, and back surfaces) are combined for each riser. Based on STAR-CCM+ results, the view factors from the vessel wall to the riser ducts are very non-uniform. The normalized peak view factor is ~ 1.84 for the peak riser duct, which is highlighted in Figure 2-4. However, it is found that one sidewall of this peak riser duct has a very large view factor facing the vessel wall, and large gaps exist in the array of riser ducts due to the vessel support structures and piping. As seen in Figure 2-3a, banks of red riser ducts extending the length of the heated cavity, and gaps are visible around the white support beams. In consultation with industry experts [2-2], it is suggested that the MHTGR and NGNP designs were only pursued to conceptual level, and therefore detailed design around cross ducts have not been addressed. In order to minimize azimuthal variation on vessel temperatures, it is necessary to have RCCS cooling panels above the structures and would have been addressed during the preliminary and final phases of the designs. An example approach is illustrated in Figure 2-5, which uses an octagonal plenum around the structures to connect the riser ducts from the bottom plenum to top plenum.

Figure 2-4: Partial CFD Model of MHTGR Cavity

Figure 2-5: Schematic for Air-Cooled RCCS behind Cross Ducts

Assuming that RCCS cooling panels will be more evenly distributed in the later phases of the design, the focus was then shifted towards the riser ducts with high view factors but not adjacent to those cross-duct structures. The top ten riser ducts with high view factors (but away from the cross-duct structures) are shown in Figure 2-6, with the normalized view factors ranging between 1.29 and 1.31. Note that this peaking factor would be reduced if additional risers were placed in the current gap space.

The view factors in NSTF between the heated surface and the riser surfaces are also calculated using the STAR-CCM+ ray-tracing model. It is found the peak normalized view factors are ~ 1.13 for the two central riser ducts (#6 and #7). However, the cavity sidewalls (with emissivity at 0.2 and not existing in the full-scale MHTGR cavity) in NSTF may have significant impacts on the actual power distribution among all the risers. The power distributions among all the risers were calculated in the CFD simulation of one NSTF baseline test, and the edge riser duct has a peaking factor ~ 1.06 . The distributions of the view factors (from the heated surface) and the power peaking are compared in Figure 2-7 for all the risers. Note that Riser #2 and #11 may also be getting non-negligible contribution from the sidewalls.

NSTF only has two azimuthal zones to control the power distribution. To mimic the azimuthal power skew of the MHTGR cavity in NSTF, the needed power peaking would be $\frac{1.31}{1.06} = 1.24$. A power peaking of 125% vs. 75% was used for the two zones of heaters in NSTF.

Figure 2-6: Riser ducts with high view factors in MHTGR Cavity

Figure 2-7: Riser ducts view factors from the heated surface in NSTF Cavity

References

- [2-1] S. Lomperski et al., RCCS Studies and NSTF Preparation Air-Cooled Option, ANL-GenIV-142, Argonne National Laboratory, September 2010.
- [2-2] Arkal Shenoy, General Atomics (retired), Personal Communications, February 2016.

3 System Characterization and Lessons Learned for NSTF Modeling

Significant efforts were devoted to better understand the effects of weather on NSTF system flow behavior. Strong influences of ambient conditions have been observed in the experimental results, when the baseline test was repeated several times with the same test conditions, yet the response of the system varied among the tests. The effects of air humidity, the building outside and inside temperature, and the wind speed were investigated. An analytical model was developed using the experimental data to correlate the zero-power system flow rates and the ambient conditions. Additionally, the heat loss characteristics of the facility were further investigated.

3.1 Air Humidity Effects

In NSTF tests, the outdoor relative humidity (RH) was observed to span between 25 and 98%. The impact of the humidity on NSTF system behavior was evaluated. Figure 3-1 [3-1] shows the moist air density as a function of temperature for an increasing relative humidity from the minimum value of 0%, corresponding to dry air (top curve), up to the maximum value of RH = 100% (lower curve), corresponding to saturated conditions, in 10% steps. The increase of relative humidity leads to a decrease of humid air density, especially at the range of higher temperatures. While a relative humidity increase between 0 and 100% leads to an almost negligible density decrease of humid air at near freezing temperatures, it is responsible for a significant reduction of humid air density of about 37.5%, at temperatures close to 100 °C.

Similar findings were found for other properties of the humid air, i.e. when the temperature is low, the increase of relative humidity has negligible impacts on the air properties. This is expected as the low temperature air can only absorb very limited content of water vapor in the air. Figure 3-2 shows the change of relative humidity with the increase of temperature. If NSTF air has 80% RH and 20 °C air at the downcomer inlet, by the time it heats up and leaves the risers at ~100 °C its RH is less than 2%. The effects of air humidity were also investigated using RELAP5 simulations of Run020. Figure 3-3 compared the calculated mass flow rates of the system for Run020 with dry air and with saturated air. The maximum difference between the two cases in mass flow rates is approximately 2%. Note that Run020 has highest inlet temperature for all baseline tests, yet the differences between saturated air and dry air are very small. It therefore can be concluded that the air humidity would only have negligible impacts to NSTF system behavior, and does not need to be considered in the computational models.

Figure 3-1: The moist air density as a function of temperature with the relative humidity

Figure 3-2: The relative humidity as a function of temperature with inlet air at 80% RH and 20 °C

Figure 3-3: The effects of humidity to system flow rate, RELAP5 simulations of Run020

3.2 Ambient Condition Effects

NSTF is housed inside Building 308 at Argonne, with only a portion of the exhausting chimney ducts directly exposed to the outside environment. The presence of the building influences the ambient condition effects on the performance of the facility, which has been observed in the experimental data.

An example of these effects can be observed when comparing the experimental results of baseline tests, which are shown in Figure 3-4. The baseline test case, as performed in the experimental testing program, was conducted on a regular basis to allow continuous monitoring of nominal facility behavior. Each test was performed in an identical facility configuration (uniform power profile, full elevation discharge via vertical chimney stacks) and in such a manner that maintained an equal time-power history across the test procedures. However, the response of the system varied between the tests.

Across the operational testing window, a total of eight baseline runs were conducted, with dates and outdoor temperature conditions summarized below in Table 3-1. Across the eight runs, the observed outdoor temperature spanned -18.1°C to 23.7°C, or a total range of 41.8°C.

Table 3-1: Overview of repeatability testing at baseline conditions

Test No.	Test Quality	Dates Performed	Outdoor Temperature, °C		
			<i>Average</i>	<i>Minimum</i>	<i>Maximum</i>
Run003	Trending	03/10–12/2014	n/a	n/a	n/a
Run004	Accepted	04/09–11/2014	13.9	10.2	17.1
Run011	Accepted	01/28–30/2015	2.09	1.40	2.80
Run020	Accepted	09/05–07/2015	23.3	22.5	23.7
SPEF003	Separate Effects	10/12–19/2015	11.5	11.1	12.2
Run022	Accepted	01/17–22/2016	-17.6	-18.1	-16.8
Run024	Accepted	04/08–14/2016	5.3	-4.90	17.3
Run026	Accepted	05/16–20/2016	13.6	5.80	21.6

A number of examinations were made to the multiple baseline test cases in an attempt to quantify and ascertain facility repeatability. A number of metrics were used for comparison, e.g. thermal power within the test section determined by $\dot{m}C_p\Delta T$, where $\dot{m}$ is the total system flow rate, C_p is the specific heat determined at the average gas temperature, and ΔT is the difference between the gas at the outlet and inlet of the heated test section. Other metrics include as-measured temperatures, pressure drop, etc. At first glance of the measured thermal powers, Figure 3-4a, all test cases exhibit similar behavior and measured values fall within an acceptable variance (0.37 kW) from the sample mean (49.45 kW). However, further examination of other parameters, such as the heated section temperature rise (Figure 3-4b) and system mass flow rate (Figure 3-4c), indicate large differences across the different runs. Isothermal forced flow testing was performed on a regular basis and indicated that frictional losses, e.g. due to geometric changes, had not changed within the bounds of experimental uncertainty. The ambient conditions (outside temperature and wind) and the building interior temperature are likely the dominant factors.

(a) Thermal energy removed by air in risers

(b) Air temperature increase in risers

(c) Mass flow rates

Figure 3-4: Repeatability of NSTF baseline tests

The natural draft of air was continuously observed before power was turned on in NSTF experiments. As discussed in Ref. [3-2], the natural ventilation by air inflow and outflow depends on the size and location of all air leakage sites on the building envelope and the indoor-outdoor pressure difference across each of these sites. These pressure differences are the result of a non-linear interaction between wind pressures on the exterior of the building and stack effect pressures caused by the density difference between indoor and outdoor air. The independent wind and stack effects would naturally interact to set the building indoor pressure that maintains a balance between the overall inflow and outflow mass flow rates across the building envelope.

Considering the uncertainty in estimating wind shelter effects and in determining the distribution of leakage sites, an analytical study of the combined wind and stack effect in Building 308 (which houses NSTF) would be very challenging. The location of leakage on the building envelope determines the wind and stack pressures that occur across each leakage site.

Therefore, it was proposed to measure the airflow rates in the NSTF system under zero-power and a wide range of wind and outside temperature conditions. By relying on a large database of airflow rates, an empirical relationship was developed between the NSTF airflow rates and the ambient conditions.

3.2.1 Stack- and Wind-Effect Pressures

Figure 3-5A illustrates the linear change of stack pressure with height (assuming linear change of air density due to temperature change for illustration purpose), h , given by

$$P_s(h) = \rho_{out} g h \frac{T_{in} - T_{out}}{T_{in}} \quad (3-1)$$

where the maximum pressure is P_{stack} (at $h = H$). This figure is for the case when $T_{in} > T_{out}$. The sign of the pressure is reversed for $T_{in} < T_{out}$. In order to balance the flows in and out of the building there is a pressure shift inside the building ($P_{s,i}$) that results in the pressure variation shown in Figure 3-5B. This would result in outflow through the upper part of the building (chimney outlet) and inflow in the lower part. At the neutral level height $h_{o,s}$, dependent upon the leakage distribution, there is no pressure difference across the wall. The pressure difference across the structure then varies linearly with height, which results in flow distributions illustrated in Figure 3-5C.

Figure 3-5: Stack effects on pressure and flow. A: (stack pressure head), B: (pressure difference), C: air infiltration mass flow.

The wind would also affect the air infiltration dependent on leakage distribution, pressure coefficients, and inflow and outflow balance. In general, wind flowing over the top of a chimney can increase draft by producing a driving low pressure region that assists in pulling air from the chimney. However, under different geometric configurations and wind directions, the wind can be adverse to the chimney upward flow by creating positive pressure at the top of the chimney. The wind blowing around a building produces a positive pressure zone on the windward side and a negative pressure zone on the leeward side. These pressures act on the leaks in the building envelope, causing airflow through them and changing the pressures within the building. Assuming the largest leakage path of the Building 308 is through the NSTF chimney outlet, the wind would always enhance the flow in NSTF tests. The wind influence on the building envelope pressure and NSTF air flow rate are illustrated in Figure 3-6.

Figure 3-6: Wind effects to air infiltration in Building 308 and NSTF system flow

The wind effect on the driving pressure (pressure difference between inside and outside of the building) for NSTF flow can be modeled as:

$$\Delta P_w = C_w P_{wind} = \frac{C_w \rho_{out} V_w^2}{2} \quad (3-2)$$

In which C_w is the wind effect factor, and V_w is the wind speed.

3.2.2 An Empirical Model of Stack and Wind Effects in NSTF

Combining the stack and wind effects together, the total additional driving pressure can be modeled as:

$$\Delta P_{total} = P_s + \Delta P_w = \beta \rho (T_{in} - T_{out}) g H + \frac{C_w \rho_{out} V_w^2}{2} \quad (3-3)$$

In which β is the air expansion coefficient, and H is height between NSTF chimney outlet and the ground. This approach models the full air flow path from the outside of the building, infiltrating to the inside of the building at the ground level, entering into the NSTF downcomer, and passing through NSTF piping, and leaving the NSTF duct system through the chimney

outlets. The total driving force would be balanced by the pressure losses in all parts of the system. The total pressure loss can be correlated with the system flow rate:

$$\Delta P_{loss} = C_l \dot{m}^n = \sum \left(K_i \frac{1}{2} \rho v_i^2 + f_j \frac{L}{D} \frac{1}{2} \rho v_j^2 \right) \quad (3-4)$$

In which K_i is the form loss coefficient, and f_j is the friction coefficient. Combining Equation (3-3) and (3-4), we have:

$$C_l \dot{m}^n = \beta \rho (T_{in} - T_{out}) g H + \frac{C_w \rho_{out} V_w^2}{2} \quad \text{or} \quad (3-5)$$

$$\dot{m}^n = C'_s (T_{in} - T_{out}) + C'_w V_w^2 \quad (3-6)$$

in which

$$C'_s = \frac{\beta \rho g H}{C_l} \quad (3-7)$$

$$C'_w = \frac{C_w \rho_{out}}{2 C_l} \quad (3-8)$$

The values of the empirical coefficients C'_s , C'_w , and the exponent n ($1 < n < 2$) can be determined from the NSTF zero-power test results. For orifice-type resistance such as holes, we expect $n = 2$ (constant form loss coefficient). For long channels the lower limit on n is the fully-developed laminar flow condition $n = 1$, as $f = \frac{C}{Re}$. For the flow rates we observed in the NSTF riser ducts, the flow is always turbulent. For the Reynolds number range observed, the exponent n should be 1.75 if the Blasius friction factor correlation [3-3] is considered,

$$f = 0.316 Re^{-0.25}. \quad (3-9)$$

Combining the effects from both the form losses ($n = 2$) and friction losses ($n = 1.75$ for turbulent flow and $n = 1$ for laminar flow), and considering the dominant pressure loss in the whole system is the turbulent friction loss in the riser ducts ($n = 1.75$), we chose $n = 1.8$, and used the experiment data to determine the coefficients C'_s and C'_w .

$$\dot{m} = (C'_s \Delta T + C'_w V_w^2)^{5/9}. \quad (3-10)$$

The result will provide a physical basis for predicting the induced flow rate as a function of ambient temperature and wind conditions.

Relevant data generated at zero power conditions were compiled and prepared into a format that would allow fitting of these two coefficients. These data sets included pre-testing zero flow values, along with an additional separate effects test SPEF005 conducted without electric heating. The span of available data is summarized below in Table 3-2. The measured flow rate and ambient wind speed from the separate effect test SPEF005 are shown in Figure 3-7. Even without active heating, the driving pressure differences, caused by density differences (stack effects) and augmented by wind, create a natural draft through the NSTF.

Table 3-2: Span of available data collected at zero power conditions

	$T_{inlets}, ^\circ\text{C}$	$T_{outdoor}, ^\circ\text{C}$	$\Delta T, ^\circ\text{C}$	Wind Speed, m/s	Flow Rate, kg/min
Minimum	15.85	-16.7	0.44	0.04	0.12
Maximum	25.77	27.97	32.93	11.20	37.19

Figure 3-7: Measured flow rates and wind speed in SPEF005

A fitting program was then scripted in MATLAB [3-4] that iterated over values of constants C'_s and C'_w , compared the resulting mass flow rate against actual values, and identified best fit values based on a minimum root-mean-squared-error (RMSE). Both constants were given a range from 0 to 20 at 0.01 increments. The results of determined best-fit values for C'_s and C'_w are 5.53 and 3.75, respectively.

$$\dot{m} = (5.53 \Delta T + 3.75 V_w^2)^{5/9}. \quad (3-11)$$

A response surface was generated showing the calculated flow rate as a function of temperature and wind speeds, as shown in Figure 3-8.

The correlation was then compared first against the test data from SPEF005, shown in Figure 3-9. Given that this data was included in determining these constants, the high level of agreement is expected. To ensure validity of this predicative capability, a new test (SPEF007) was performed in the test facility that generated data not used in the fitting correlation. The results are shown in Figure 3-10, further confirming validity and confidence in the findings. The goodness of the fit between the correlation and measured data were further confirmed by the Root Mean Squared Errors (RMSE) of 0.79 kg/min and 0.66 kg/min for SPEF005 and SPEF007, respectively. This is deemed very good considering that the measurement uncertainty of mass flow rate is 0.72 kg/min. The R^2 values of the comparisons between the correlation and measurements are 0.87 for SPEF005 and 0.93 for SPEF007.

Figure 3-8: Response surface of fitting correlation

Figure 3-9: Fitting correlation applied to SPEF005

Figure 3-10: Fitting correlation applied to SPEF007

3.3 Building Temperature Effects

The differences in the baseline tests can also be partially attributed to changes in the building temperature. This effect can be described as follows: the driving pressure head of a natural circulation system is highly dependent on the density difference between the cold and hot segments. Given the non-linear relationship between density and temperature for an ideal air gas, Figure 3-11 [3-5], the absolute inlet temperature plays a role, even for two systems with identical temperature differences. With higher absolute inlet temperatures, the $\Delta\rho/\Delta T$ is reduced, which lowered the driving pressure head and reduced the overall efficiency of the stack effect.

Thus, with elevated downcomer inlet temperatures causing a reduction in the driving head, the mass flow rate is reduced, and to maintain equal thermal powers, all of the system parameters shift. Specifically, the air temperature rise increases, absolute gas temperature increases, frictional pressure drop decreases, etc. The inlet temperature effect can be easily captured by applying realistic air property models in the computational models.

Figure 3-11: Relationship between density and temperature for air

3.4 Heat loss characterization of NSTF Experiments

The power removed by the NSTF riser ducts is a key parameter for assessment of the facility performance. Since an adiabatic wall was not achieved experimentally, thermal losses were unavoidable. While these do not detract from the overall quality of the testing, it is important to accurately quantify these to obtain knowledge of the heat input into the test section adjusted for losses.

To more confidently assess the conversion of electric heater power to thermal energy within the test section, or “efficiency”, under a wide range of conditions, a separate effects forced-flow test was performed. This test obtained trends for the efficiency for a number of combinations of mass flow rate and heater power, which covered the range of NSTF test conditions while allowing for greater control over inlet conditions than in typical natural convection cases. The power supplied to the electric heaters is controlled from a set of industrial power controllers [3-6]. The ability to control voltage and current levels to the resistive heaters is one basis for determining thermal losses. The heat transfer path for electric power from the heaters to the working fluid within the riser ducts includes a complex route of conduction, convection, and radiation. During this process a portion of the originating power is removed as parasitic losses across the insulated walls, back cavity, and other paths of leakage around the heated enclosure. The efficiency of this thermal network was determined through the separate effects study that examined a range of electric powers and system flow rates, with comparisons to the measured thermal power removed by the risers.

The calculated power has incorporated the average temperature of the outlet gas via DTS fiber measurements. Figure 3-12 shows that the NSTF operates at a nominal 65% thermal

efficiency from the original electric power to the heaters, with only small flow-rate dependence. With this knowledge, testing procedures can be refined to include best estimates of the required electric power to achieve a desired test section heat removal rate, along with more accurate inputs to computational modeling. Results for the heated surface temperature are similar to those from the natural convection experiments. The efficiency is a very useful parameter for assessing integral behavior of the system, but more detailed thermal boundary conditions should be applied for better comparisons on the heated surface and cavity wall temperatures between experiments and simulations.

Figure 3-12: Thermal efficiency of the NSTF, determined by separate effects testing at varying electric powers and flow rates

Previous heat loss calculations [3-7] through cavity side walls were estimated based on the convective heat transfer at the outer surfaces of the cavity walls and the temperature measurements from thermal imaging data. Less than 10 kW heat loss was calculated during the steady-state periods of the baseline tests (nominally at 80 kW_e) from the cavity walls, significantly less than the heat loss observed in the above experiments. It was found that there is an air gap between the primary heated plates and heater panels [3-6], which allows airflow and significant heat loss through natural convection. Although multiple factors may affect the heat loss in NSTF cavity, the free convection, through the air gap between the heated plated and heater panels, is likely the dominant factor.

References

- [3-1] P.T. Tsilingiris, "Thermophysical and transport properties of humid air at temperature range between 0 and 100 °C," *Energy Conversion and Management*, Vol. 49, pp. 1098–1110, 2008.

- [3-2] I.S. Walker and D.J. Wilson, “Evaluating Models for Superposition of Wind and Stack Effect in Air Infiltration,” *Building and Environment*, Vol. 28, No. 2, pp. 201-210, 1993.
- [3-3] F. P. Incropera and D. P. DeWitt., *Fundamentals of Heat and Mass Transfer*. New York: J. Wiley, 2002.
- [3-4] The MathWorks, Inc., MATLAB and Statistics Toolbox Release 2013b, Natick, Massachusetts, United States, 2013.
- [3-5] D.D. Lisowski, A.R. Kraus, M.D. Bucknor, R. Hu, M.T. Farmer, “Experimental observations of natural circulation flow in the NSTF,” *Nuclear Engineering and Design*, 2016. <http://dx.doi.org/10.1016/j.nucengdes.2016.01.014>
- [3-6] D.D. Lisowski, M.T. Farmer, et al., “Design Report for the 1/2 Scale Air-Cooled RCCS Tests in the Natural convection Shutdown heat removal Test Facility (NSTF),” ANL-SMR-8, Argonne National Laboratory, June 2014.
- [3-7] R. Hu, A. Kraus, M. Bucknor, et al., Argonne National Laboratory, unpublished information, September 2015.

4 RELAP5-3D System Analysis of NSTF

Systems codes, such as RELAP5-3D [4-1], provide the ability to analyze overall system behavior during transients with minimal computational resources. RELAP5-3D was developed by Idaho National Laboratory to simulate transients in light water reactor systems, such as loss of coolant accidents and anticipated transients without scram. However, the code is flexible enough to handle a wide variety of hydraulic and thermal transients in both nuclear and non-nuclear systems. For these reasons, a RELAP5-3D model of the NSTF was developed and used to investigate the overall system behavior during transients, which includes simulation of the effects of ambient conditions, such as outdoor air temperature and wind, on the performance of the facility. To analyze the ability of the model to simulate important performance metrics of the facility (such as the air mass flow rate), comparisons were made between experimental test data and code simulations results.

4.1 Model Description

The computational code used for the system-level modeling of the NSTF was RELAP5-3D, version 4.0.3 [4-1]. Because the goal of the modeling effort was to capture the integral natural circulation system behavior of the facility, not all components of the NSTF were represented in the model. For example, the cavity of the NSTF was not modeled. The heat transfer in the cavity of the NSTF is a combination of thermal radiation and natural convection. While RELAP5-3D can model thermal radiation, the radiation model has limitations. Also, the geometry of the cavity is complex, which makes it very difficult to accurately model the complex three-dimensional natural convective flow using RELAP5-3D. Since the cavity was not modeled, heat flux boundary conditions were modeled directly at the outer surfaces of riser ducts which greatly simplifies the analysis and assures that limitations of the code to model the radiative heat transfer and multi-dimensional flow, do not limit the ability of the model to accurately simulate the overall performance of the facility.

A nodalization diagram of the RELAP5-3D model of the NSTF is shown in Figure 4-1. In the model, air enters the system from a time dependent volume (TD900 in Figure 4-1) that represents the atmosphere outside of Building 308 at Argonne. The air flows through three virtual (discussed in Section 4.2.1) volumes (P902-P904-B905), a downcomer (P910), and into the lower plenum (B920) before entering the twelve hot riser ducts represented by a single flow path (P930). The air temperature increases as the air passes through the hot riser ducts and into the outlet plenum (B941). The air then flows from the outlet plenum into two chimney ducts represented by a single flow path (P950-B960-B970-P980-P985) before being exhausted to a time dependent volume (TD990) that represent the atmosphere outside Building 308. The model also includes two fan loft ducts represented by a single pathway that are utilized during forced flow testing at the facility. During these forced flow tests, air is directed from B970 to TD975, and the flow path from B970 to P980 is blocked.

Heat structures were modeled on the upper plenum and on the main chimney ductwork in an attempt to treat the heat losses that occur from those components. These heat structures represent the metal walls of the components as well as any insulation material applied to the outside of the facility to minimize heat losses. An additional heat structure is modeled on virtual component P904 and its purpose is discussed in Section 4.2.1.

Figure 4-1: RELAP5-3D nodalization diagram of the NSTF

4.2 Model Development and Verification

Previous model development and verification focused on characterizing the effects of certain components of the NSTF such as the pressure drop from the presence of the flow conditioner on the inlet downcomer and verifying simulation results for the pressure drop across the riser ducts [4-2]. Modeling efforts in FY16 were primarily focused on capturing the effects of outside air temperature, building pressure, and wind speed on the performance of the NSTF. Several major RELAP5-3D model modifications were made which include:

- 1) Virtual components were added to account for the effects of outdoor (of building 308 at Argonne) air temperature on the facility.
- 2) The form loss of the air entering the building was adjusted to “tune” the air mass flow rate of the simulation to the measured air mass flow rate of the facility.
- 3) Pressure boundary conditions were added to the inlet and outlet of the model to account for the effects of wind on the facility.
- 4) The elevation of the chimney ducts was extended in some simulations to account for the addition of new chimney caps added to the facility to reduce the likelihood of flow instabilities.
- 5) The model input power was calculated from experimental data of the mass flow rate of the system and the difference in air temperature of the air entering and existing the riser ducts. Previously, the model input power was scaled from the power level of the electric heaters.
- 6) The two chimney ducts were combined into a single flow path in the model to reduce the likelihood of flow reversals during low flow conditions.

4.2.1 Outdoor Air Temperature Effects

As discussed in Section 3.2, the effects of outside air temperature and wind on the mass flow rate of air through the facility during natural circulation experimental tests are substantial. In FY15, the effect of outdoor air temperature was investigated, but only to characterize its effect on heat losses through the chimney duct walls and insulation. In FY16, the effect of outdoor air temperature on the mass flow rate of the system was modeled directly. To simulate this effect on the performance of the facility, several virtual volumes were added to the model (see Figure 4-1). The first volume, P902, was added to account for the hydrostatic head of a column of air outside of building 308. The height of this column was set to the elevation of the exit of the chimney caps above ground level. A second virtual volume, P904, was added to the model to condition the air (using a temperature boundary condition on heat structure HS6000) from outdoor air temperatures (measured via a weather station) to NSTF inlet air temperatures (measured by thermal couples at the riser duct inlets). Virtual volume P905 was added to account for the hydrostatic head of air between ground level and the inlet of the downcomer (located in the pit of building 308).

4.2.2 Building Form Loss Effects

To account for the resistance of air flowing from outside of building 308 to inside, form losses were modeled in the junction (J903) between P902 and P904. This form loss was also used to “tune” the mass flow rate of a single simulation to experimental data, which will be

discussed in more detail in Section 4.3. Once the model had been calibrated to a single simulation, additional simulations of other experimental tests were conducted to access the ability of the model to predict the overall behavior of the facility.

4.2.3 Wind Effects

The effect of wind on the performance of the facility was captured by inducing an artificial pressure difference between the model inlet (TD900) and outlet (TD990). This pressure difference was determined from Equation (3-2) considering only the wind effect as the effect of outdoor air temperature on the performance of the facility was modeled separately. For RELAP5-3D modeling,

$$\Delta p(t) = c_w \frac{\rho u(t)^2}{2} \quad (4-1)$$

in which Δp is the time-dependent pressure difference between TD900 and TD990 in Pa, c_w is the wind effect constant, ρ is the outside air density in kg/m^3 , and $u(t)$ is the time-dependent wind speed in m/s. Using experimental data and combining the empirical correlation Equation (3-11) with Equations (3-7) and (3-8), c_w is calculated to be 0.738.

4.2.4 The Addition of New Chimney Caps

In an attempt to reduce the likelihood of flow instabilities during startup, new anti-down draft chimney caps were designed and installed on the facility in November 2015. The new caps increase the elevation of the outlet of the chimney ducts by 30 inches. This additional height was added to the RELAP5-3D model of the NSTF (on P985) when the model was utilized to simulate tests performed subsequent to the addition of the new caps.

4.2.5 Model Input Power

In FY15, the model input power was calculated by scaling the electric power history of the heaters of the NSTF and applying that power history as a boundary condition on the riser duct outer surface. This approach did not successfully capture the time required for heat to transfer from the heaters to the riser ducts, which indicated that the heat capacity of all structures in the NSTF cavity and the heat losses were not correctly captured. As the cavity was no longer modeled in the updated model, the input power was re-calculated using experimental data of the mass flow rate of air and the temperature difference of air entering and exiting the riser ducts (along with the specific heat of air). This method led to much better agreement between the mass flow rate of air in the facility and the simulation results. A comparison of the model input power methods used in FY15 and FY16 along with corresponding simulation results are shown in Figure 4-2 and Figure 4-3 respectively.

Figure 4-2: Example comparison of model input power estimation methods

Figure 4-3: Example comparison of mass flow rate results using different input power methods

4.2.6 Flow Instabilities at Low Flow Conditions

In FY15, the RELAP5-3D model of the NSTF included two flow paths that represented the two chimney ducts of the facility. During simulations, at low flow conditions (such as during startup), RELAP5-3D would often predict flow reversals in one (or both) of the chimneys. Once a flow reversal occurred in the simulation, it would persist until the end of the simulation. These reversals drastically altered the simulation results and made comparisons of simulation results to experimental data difficult. To combat the flow reversal issue, a small artificial pressure difference (less than 10 Pa) was added between the inlet and outlets of the facility to ensure the flow reversals would not occur. Once the mass flow rate of the system increased enough such that the flow reversal would not occur, the artificial pressure difference between the inlet and

outlets of the facility was removed. However, the time at which this pressure difference could be removed was not consistent between simulations because the boundary conditions of the simulation changed depending on which experimental test was being investigated. Also, the pressure difference artificially increased the mass flow rate of the simulation results, which also made it difficult to compare the simulation results to experimental data.

Flow reversals are common during startup of the NSTF and actions (adjusting chimney dampers) are sometimes taken by the NSTF staff to ensure they do not occur. It is unclear if the flow reversals in the simulations are physical, due to the boundary conditions of the model, or are the result of an issue in RELAP5-3D to simulate flow through symmetric flow paths at low flow conditions and further investigations are needed to provide an answer. To temporarily resolve the issue until further investigations are conducted, in FY16, the two symmetric flow paths representing the chimney ducts were combined into a single flow path. Following this modification, no flow reversals have been observed in the simulation results. The assumption of a single flow path impacts the comparison of the model results with experimental data since the flow rates of each chimney are not identical in the experiments. To compare experimental data with model results, the data are averaged over both chimneys.

4.3 Model Inputs and “Tuning” to Run022

After the model modifications discussed in the previous section were made, four time-dependent inputs are required to perform a simulation using the RELAP5-3D model of the NSTF. Those inputs include:

- Power
- Wind speed
- Inlet air temperature
- Outdoor air temperature

The time-dependent inputs are all derived from experimental data. However, due to limitations in the number of entries allowed in tables in RELAP5-3D (due to the card numbering system), post-processing of the experimental data is required before it can be utilized in the simulation. Two methods were used to post-process the experimental data for a base case test (Run022). The first method calculated the moving average of the experimental data at points that were preselected by members of the analysis team. The number of points required for each input varied depending on the fluctuations of the input parameter and the magnitude of the fluctuations. The second method also used a moving average at preselected points, but allowed the moving average to deviate in order to capture maximum values of certain fluctuations. This method was only utilized with the wind data, which exhibited more frequent and higher magnitude fluctuations than the other input data. Figure 4-4 provides a comparison of the experimental data to the input values used in the simulation of Run022.

Once the simulation input parameters were derived, the model was “tuned” (calibrated) until the mass flow rate of the simulation approximated the average of the mass flow rate of the experimental data. To calibrate the model, the form loss representing the resistance of airflow from outside of building 308 to the interior of the building was adjusted in J903 (See Figure 4-1). Run022 data was selected to “tune” the simulation results for the following reasons:

- Long periods (several hours) of low wind conditions (less than 1 m/s)
- Facility improvements were made just prior to Run022 to reduce leaks in the chimney ducts (addressed in Section 2.2).
- New chimney caps were added prior to Run022 in an attempt to reduce asymmetric air flow in the chimneys due to wind.

The measurement uncertainties of major parameters are summarized below in Table 4-1. The detailed descriptions of the instrumentation and uncertainties in experimental data can be found in ANL-SMR-8 [4-3].

Table 4-1: Measurement uncertainties of primary sensors in test facility

Sensor Type	Measurement Uncertainty
Temperature - air gas and heated plate	± 1.1 °C or 0.4%
Temperature - air gas temperature rise	± 1.55 °C
Temperature - all other walls	± 2.2 °C or 0.75%
Mass flow rate	$\pm$ (1% of reading + 0.3 kg/min)
Riser Differential Pressure	± 1.24 Pa
DTS Fiber (riser region - axial and riser outlet)	± 2.2 °C
DTS Fiber (plenum gas)	± 4 °C
Wind Speed	± 0.1 m/s

Simulation results for Run022 are provided in Figure 4-5. Simulation results for the “RELAP5 – No Wind/Temperature” are provided to demonstrate the importance in capturing the effects of outside air temperature and wind of the facility. If these effects were neglected, the simulation results significantly under-predicted the mass flow rate (the average absolute error is approximately 18%) of the facility which led to an over-prediction in average temperature rise of the air entering and exiting the riser ducts and an under-prediction of both the average pressure drop across the riser ducts and the air velocity in the chimneys.

For the additional simulations in which the effects of outdoor air temperature on the performance of the facility are modeled directly, the results provide better agreement in mass flow rate to the experimental data. For example, the average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 2%, which is about the same as the measurement uncertainty (shown in Table 4-1). Since the mass flow rate at low wind conditions was utilized to “tune” the model, the agreement between the simulation and the experimental data is very good (the average absolute error between the model simulations and the experimental data is less than 1.7%) from 40 hours to 120 hours. Prior to 40 hours, significant wind effects were observed in the experimental data. In this region, the simulation results with no wind effects modeled under-approximate the mass flow rate of the system (the average absolute error is less than 5%). The simulation results where several peak wind speed were also included to demonstrate that the simulation can capture the peak mass flow rates in the experiments if the detailed variation of wind speeds were modeled in the simulation. It should be noted that the average wind speed plus peak simulation results do not correspond to the highest experimental data points because the data collected by the weather station was actually being averaged during data acquisition. The weather station sampled wind speed several times a

minute, but only records a 1-minute average and a peak value during that 1-minute time period. Therefore, the peaks seen in the experimental wind data are lower than the true peak wind speed during the same timeframe. For example, the highest 1-minute-averaged wind speed measured by the weather station during Run022 was 8.5 m/s, however, the actual peak wind speed was measured to be approximately 12 m/s.

As expected, since the mass flow rate of the simulations closely match the experimental data, the average temperature rise of the air entering and exiting the riser ducts predicted by the simulation also agree well with the experimental data. The average absolute error of the air temperature rise between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 2%. The time-dependent behavior of the pressure drop across the riser ducts was captured well in the simulation results; however, the simulation results consistently under-predicted the absolute value of the experimental data. These results need further investigation, but are currently believed to be the result of post-processing the experimental data. Impulse plumbing is used to obtain these pressure data and a consequence of using impulse plumbing is that the signal represents a measurement at the device itself. Thus, when using impulse plumbing to measure the losses across a vertical pipe, the measured value at the device includes the effects of the gravitational head in the impulse plumbing, which is not necessarily the same as the gravitational head in the riser duct. To accurately quantify the hydrostatic head within the impulse tubing, thermocouples were added along the bundle of 1/4” nylon tubing. However, due to the nearly 10 m total length of the lines, local variations may exist and not necessarily be captured by the thermocouples. The estimated hydrostatic head (calculated from thermocouple measurements) is added to the data after each experimental test. If the thermocouple measurements under-predict the temperature of the air inside the impulse tubing, the hydrostatic head will be over-predicted, which is likely the reason here for the difference in the experimental data and the simulation results. This issue is present in all of the average riser pressure drop comparisons between RELAP5-3D simulation results and experimental data in this report. It should be noted that comparisons between simulation results and experimental data for the pressure drop across the riser ducts due to friction were investigated previously [4-2] and were found to agree well (less than 1% differences).

The simulation predicted averaged (over the two chimney ducts) air velocity in the chimney ducts was significantly less than the measured air velocity even though the simulation accurately predicts the mass flow rate of the facility. This is most likely due to the location of the pitot tubes in the chimney ducts, shown in Figure 4-6, which are located just downstream of bends in the ducts. In fully-developed turbulent flow, the velocity profile should be relatively flat (except near the walls) in the chimney ducts, however, due to the presence of the bends in the ducts, the flow is most likely not fully-developed and therefore the velocity profile is not flat. The velocity measurements are single-point measurements taken at the center of the ducts. These measurements are most likely near the maximum of the velocity profile and are therefore recording a higher-than-average velocity. The velocities predicted by the simulations are average velocities and are therefore lower than the measured data. This issue is present in all of the averaged chimney air velocity comparisons between RELAP5-3D simulation results and experimental data in this report.

Figure 4-4: Run022: RELAP5-3D model inputs estimated from experimental data

Figure 4-5: Run022: Comparison of RELAP5-3D simulation results and experimental data

Figure 4-6: Installation locations of pitot tubes in the chimney ducts

4.4 Base Case Tests Procedures

Base case tests are conducted with the facility in the natural circulation configuration, meaning that the chimney dampers are positioned to direct flow through the north and south chimneys instead of through the horizontal ductwork associated with forced flow tests. As the test begins, the electrical power (used to heat the reactor vessel analogue) is increased from 0 kW to a low-power steady-state level over a two-hour period. The power level remains at this level for some duration of time (until steady-state conditions are reached) before increasing to a high-power steady-state level (again over a two-hour period). The power then remains at the high-power level for some time duration before being reduced back to 0 kW. An example of a base case test power profile is shown in Figure 4-2.

4.5 Additional Base Case Test Simulations

Once the modifications had been made to the RELAP5-3D model of the NSTF and the model had been “tuned” to Run022, additional base case test simulations were performed and the results were compared to experimental data. The additional simulations performed were for base case tests Run011, Run020, and Run024. By performing the comparison of the simulation results from these tests to the experimental data, the performance of the model can be ascertained over a wide range of conditions. The range of each input parameter for each test is provided in Table 4-2 along with an overall simulation range, which summarizes the conditions over which the performance of the model has been evaluated.

Table 4-2: Approximate range of RELAP5-3D inputs values for each base case simulation

Simulation	Power Range [kW]	Wind Speed Range [m/s]	Avg. Riser Inlet Air Temp. Range [°C]	Outdoor Air Temp. Range [°C]
Run011	0.0 – 48.9	0.6 – 7.0	18.6 – 20.1	-5.9 – 2.4
Run020	0.0 – 44.6	0.4 – 3.9	24.4 – 31.1	22.4 – 33.4
Run022	0.0 – 49.5	0.0 – 8.5	15.3 – 21.1	-20.3 – 0.3
Run024	0.0 – 50.7	0.3 – 7.2	17.4 – 25.4	-4.9 – 10.3
Overall	0.0 – 50.7	0.0 – 8.5	15.3 – 31.1	-20.3 – 33.4

The four time-dependent model input parameters of each test case were derived from experimental data using the methodology described in Section 4.3. The model input parameters and the experimental data from which they were derived are provided for Run011, Run020, and Run 024 in Figure 4-7, Figure 4-9, and Figure 4-11 respectively. Comparisons of the simulation results to experimental data for each test are provided in Figure 4-8, Figure 4-10, and Figure 4-12. The remainder of this section will provide a discussion of the conditions of each test and of the comparisons of the simulation results to the experimental data. The discussion for Run022 is provided in Section 4.3.

4.5.1 Run011

Run011 was conducted from 01/28/15 to 01/30/15. The results presented in this section focus solely on the timeframe (approximately 50 hours) during which the heaters were operating. It should be noted that the chimney caps on the facility during the time period that Run011 was conducted were the older style chimney caps, which were not intended to minimize the effects of wind on the facility. Due to the time of year the test was conducted, the outdoor air temperature was relatively cold (approximate range: -5.9°C to 2.4°C). The average riser inlet air temperature was significantly warmer (approximate range: 18.6°C to 20.5°C) due to the building heating system. The averaged wind speed range, as measured by a weather station located on the roof of Building 308, (see discussion in Section 4.3 which provides details on how the weather station records data) was 0 m/s to 10 m/s.

The RELAP5-3D simulation results (provided in Figure 4-8) highlight the effect of wind on the performance of the facility. Considering the comparison of the simulation results and experimental data for the mass flow rate of the system, the no-wind simulation results agreed very well with the average experimental data while the average wind simulation results slightly over-approximated the average experimental data. These results were expected for Run011 due to the mass flow rate of the system being slightly lower than normal (as much as approximately 2%) due to outdoor air ingress in the horizontal fan loft ducts. These air ingress issues were later addressed and reported under the Quality Assurance Plan (QAP), ANL-NSTF-000000-DAR-005-R0. The average wind results do however capture the flow perturbations caused by the wind (for example the perturbation caused by the wind gust at approximately 37 hours).

As expected, since the mass flow rate of the simulations closely matches the experimental data (the average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 3%), the average temperature rise of the air entering and exiting the riser ducts predicted by the simulations also agrees well with the experimental data (the

average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 4%). The time-dependent behavior of the pressure drop across the riser ducts and the chimney air velocity were captured well in the simulation results; however, the simulation results consistently under-predicted the absolute value of the experimental data. This issue is discussed in detail in Section 4.3.

4.5.2 Run020

Run020 was conducted from 09/05/15 to 09/07/15. The results presented in this section focus solely on the timeframe (approximately 52 hours) during which the heaters were operating. The chimney caps on the facility during the time period that Run020 was conducted were also the older style chimney caps, which were not intended to minimize the effects of wind on the facility. Due to the time of year the test was conducted, the outdoor air temperature was relatively warm (approximate range: 22.4°C to 33.4°C) compared to Run011. The average riser inlet air temperature (approximate range: 24.4°C to 31.1°C) was similar to the outdoor temperature range. The averaged wind speed range, as measured by a weather station located on the roof of building 308, (see discussion in Section 4.3 which provides details on how the weather station records data) was 0 m/s to 6.3 m/s.

The RELAP5-3D simulation results for both the no wind and averaged wind simulations are provided in Figure 4-10. The mass flow rate results for both simulations are very similar with only minor deviations. This is due to low (less than 2.5 m/s) averaged wind speed for the majority of the simulations. However, the results of the two simulations deviate slightly in the time range from 25 hours to 35 hours due to an averaged sustained wind speed of approximately 4 m/s (the effect of this wind speed is also evident in the experimental data during the same time frame). The predicted mass flow rates of both simulations over-approximated the average experimental data. For example the average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is approximately 5%. These results were expected for Run020 due to the mass flow rate of the system being slightly lower than normal (approximately 2%) due to outdoor air ingress in the horizontal fan loft ducts, as discussed above for Run011.

As expected, since the mass flow rate of the simulations over-predicts the experimental data, the average temperature rise of the air entering and exiting the riser ducts predicted by the simulations under-predicts the experimental data (the average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 6%). The time-dependent behavior of the pressure drop across the riser ducts and the chimney air velocity were captured well in the simulation results, however, the simulation results consistently under-predicted the absolute value of the experimental data. This issue is discussed in detail in Section 4.3.

4.5.3 Run024

Run024 was conducted from 04/08/16 to 04/14/15. The results presented in this section focus solely on the timeframe (approximately 110 hours) during which the heaters were operating and both chimney ducts were utilized (the Run024 test also investigated single chimney configurations, but those data are not presented here). It should be noted that the chimney caps on the facility during the time period that Run024 was conducted were the newer chimney caps, which were installed prior to Run022 and were added to minimize flow instabilities in the

chimney ducts due to wind effects. Due to the time of year the test was conducted, the outdoor air temperature ranged from cold to mild during the test (approximate range: -4.9°C to 10.3°C). The average riser inlet air temperature was significantly warmer than the outdoor temperature (approximate range: 17.0°C to 25.4°C) due to the building heater system. The averaged wind speed range, as measured by a weather station located on the roof of Building 308, (see discussion in Section 4.3 which provides details on how the weather station records data) was 0 m/s to 11.3 m/s.

The RELAP5-3D simulation results (provided in Figure 4-12) again highlight the effect of wind on the performance of the facility. Considering the comparison of the simulation results and experimental data for the mass flow rate of the system, the no-wind simulation results under-predict the average experimental data while the average wind simulation results provide better agreement overall with the experimental data. The average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 4%. The average wind results are also able to capture the flow perturbations caused by high wind periods which peaked at approximately 10 hours, 40 hours, and 70 hours.

As expected, since the predicted mass flow rate of the average wind simulation provides better agreement with the experimental data, it also provides better agreement with the experimental data for the average temperature rise of the air entering and exiting the riser ducts when compared to the no wind simulation results (the average absolute error between the “RELAP5 – w/Wind (Avg.)” simulation and the experimental data is less than 4%). The time-dependent behavior of the pressure drop across the riser ducts and the chimney air velocity were captured well in the simulation results, however, the simulation results consistently under-predicted the absolute value of the experimental data. This issue is discussed in detail in Section 4.3.

4.5.4 Importance of Capturing Ambient Effects in Base Case Simulations

Overall, the simulation results for Run011, Run020, Run022, and Run024 are able to capture the performance trends but not all absolute values of the facility parameters over a wide range of input conditions. The importance of modeling the effects of the ambient conditions (outdoor air temperature and wind speed) is evident in the simulation results. For very similar power profiles, for example Run011 and Run020, the performance of the system varied significantly due to the ambient conditions. In Run011, the approximate mass flow rate during the high power steady-state period was approximately 35 kg/min, while for Run020 it was approximately 28 kg/min. This difference is due to higher average wind speeds and colder outside air temperature during Run011 compared to Run020. Another example of the importance of capturing the ambient effects is shown in the simulation results for Run024. The mass flow rate of the system at approximately 10 hours into the test is higher than during the high-power steady-state period, due to elevated wind speeds. If these ambient effects were not accounted for, the simulation results among the various base case simulations would be very similar to each other and the model would fail to capture the overall behavior of the facility.

Figure 4-7: Run011: RELAP5-3D model inputs estimated from experimental data

Figure 4-8: Run011: Comparison of RELAP5-3D simulation results and experimental data

Figure 4-9: Run020: RELAP5-3D model inputs estimated from experimental data

Figure 4-10: Run020: Comparison of RELAP5-3D simulation results and experimental data

Figure 4-11: Run024: RELAP5-3D model inputs estimated from experimental data

Figure 4-12: Run024: Comparison of RELAP5-3D simulation results and experimental data

4.6 Summary of RELAP5-3D Analyses

In FY16, efforts were directed at improving the RELAP5-3D model of the NSTF to capture the major system behavior in NSTF tests. Major model modifications were made to account for the significant effects of outdoor air temperature, wind, the presence of the building, and new chimney caps on the performance of the facility. The model was calibrated to experimental data from a single baseline test case (Run022) and then used to simulate three additional baseline test cases (Run011, Run020, and Run024). The time-dependent behavior of the pressure drop across the riser ducts and the chimney air velocity were captured well in the simulation results, however, the simulation results consistently under-predicted the absolute value of the experimental data for these two parameters. These under-predictions are most likely due to the location of the instrumentation in the facility and limitations of RELAP5-3D to capture local versus averaged quantities. The simulation results compared well with experimental data for all of the tests for the overall system mass flow rate (the average absolute error was less than 5% in all of the simulations using the average wind methodology) and the average temperature rise of air entering and exiting the riser ducts (the average absolute error was less than 6% in all of the simulations using the average wind methodology). These simulation results demonstrate the ability of the code to capture the effects of ambient conditions (outdoor air temperature and wind) on the performance of the facility during four experimental tests (outdoor temperatures ranging from -22.3°C to 33.4°C and wind speeds ranging from 0 m/s and 8.5 m/s). The ability of the model to capture these effects provides confidence that the natural circulation air-flow behavior in NSTF can be well captured by RELAP5 simulations and its usage could provide insights into the performance of the RCCS under various reactor system conditions, which would be required during reactor licensing. Overall, the ability of RELAP5-3D to simulate the performance of the NSTF demonstrates that it is an appropriate tool for use in transient analyses of air-cooled passive systems such as the RCCS.

References

- [4-1] The RELAP5-3D Code Development Team, "RELAP5-3D Code Manual," INEEL-EXT-98-00834, Rev. 4, Idaho National Laboratory, 2012.
- [4-2] M. Bucknor, R. Hu, D. Lisowski, A. Kraus, "Comparisons of RELAP5-3D Analyses to Experimental Data from the Natural Convention Shutdown Heat Removal Test Facility", *Proceedings of ICAPP'16*, San Francisco CA, April 17-20, 2016.
- [4-3] D. Lisowski, M. Farmer, et al., "Design Report for the ½ Scale Air-Cooled RCCS Tests in the Natural convection Shutdown heat removal Test Facility (NSTF)," ANL-SMR-8, Argonne National Laboratory, June 2014.

5 Air-Cooled NSTF CFD Analyses

5.1 CFD Model Description

5.1.1 Boundary Conditions and General Details

The three-dimensional modeling of the NSTF was performed using the computational fluid dynamics code STAR-CCM+ version 10.06 [5-1]. Earlier works published [5-2][5-3] by the NSTF program provide background and geometric details of the model description. All simulations performed within this work scope were steady state, with the objective of capturing the mean behavior of the system for a given power level.

The overall modeling approach used a second-order finite-volume approach with non-slip conditions on all walls. For the cases simulated here, a uniform heat flux boundary condition was applied to the heated wall for these tests, with values pulled from thermal power measurements of experimental test data. A symmetry boundary condition was applied across the domain centerline, which helps to reduce the problem size. The flow conditioner in the downcomer region was modeled with a porous media approach [5-1] to obtain the mean flow effect; with loss coefficients established with the aid of the experimental data. An ideal gas formulation was used for the equation of state for both the cavity and the main (RCCS) air. The Low-Mach (i.e. pressure-independent) form of this equation was used given the low flow speeds involved in this problem. Constant fluid specific heats were used, while viscosity and thermal conductivity were modeled using Sutherland's Law [5-4].

Different solver approaches were used based on the type of simulation being solved. For the forced-flow cases, a segregated solver approach was utilized to solve the continuity, momentum and energy equations. This approach is generally faster than the alternative coupled solver approach, where the aforementioned equations are solved simultaneously, and has given very similar results as those for the coupled solver for cases where the main flow is forced. For cases with natural convection-driven flow through the primary flow path of air, the segregated solver was used to obtain good initial conditions for the coupled solver in order to speed up the convergence rate. Convergence was generally 10^{-4} or lower for all normalized residuals. Convergence was also established via evaluating the energy balance of the riser ducts (minimization of the difference between the surface integral of heat flux over the outer duct surface versus the integral over the inner surface).

For the forced flow cases, a mass flow rate inlet boundary condition was used, incorporating the experimentally measured forced flow rate and inlet temperatures. For the natural convection cases, unlike in prior work, the model was updated to using a "pressure outlet" boundary at the inlet [5-3]. This allows for the indoor/outdoor pressure differential due to temperature differences and wind effects to be more easily applied. In both cases, a pressure outlet boundary was employed at the chimney exit.

Radiation was modeled within the cavity via a grey, surface-to-surface method. View factors were calculated within the code. Emissivities were taken to be 0.2 for the non-heated cavity walls and 0.8 for the heated wall and the duct walls. Conjugate heat transfer was modeled between the cavity air, steel duct walls, and the primary air flow.

The cases studied in detail here are Run22 and Run23. As stated above, Run22 is a baseline test with thermal powers of roughly 80 kW_e and 56 kW_e for the "high" power and "low" power

cases, respectively. Run23 is a forced flow case with flow rates matching those found in Run22. The heat removed from the system by the main flow was estimated as the product of the measured mass flow rate, the average specific heat of the air in the risers, and the temperature difference between the average of the twelve duct inlet thermocouples (TCs) and the average of the twelve DTS fibers at the riser outlets (Figure 5-1). The process was to first simulate Run23 to establish good modeling practices under the well-controlled forced flow conditions, such as appropriate turbulence models. Conclusions from those results were then used in simulating Run22, providing a more rigorous test of model applicability and limitations.

It is important to note that, in some regards, the forced-flow data is being used to “tune” the CFD model for more confident usage in full natural convection simulations. The NSTF is quite large and although it is insulated, there are non-negligible heat losses. Between 60-70% of the applied heat is removed by the air flowing through the risers, with the rest being lost to the surroundings. A significant portion of this is lost from the back of the heaters, but there are also non-trivial losses in the cavity sidewalls. To date, only one test case has been analyzed in detail with thermal imaging and other methods to experimentally investigate the heat loss distribution [5-3]. Simulations incorporating conduction losses through the sidewalls using the stated insulation resistance values have also far underestimated the true heat losses. For these reasons, modeling of the heat losses in the system is currently relatively simple. Other than the heated (East) cavity wall and side duct walls, all other walls in the system were treated here with an adiabatic boundary condition, and the heat applied is what is measured to be removed by the main flow. This should ensure that the amount of heat rejected by the system is correct, but it will affect some local temperatures, notably in the cavity. It should thus be expected that the predicted heated surface temperatures will be low, and the side cavity wall temperatures will be high. In light of this, the correct trends were emphasized in picking appropriate models over the correct magnitudes for Run23. The most appropriate models can be used in future work to help calibrate the heat loss distribution at different powers and improve those local estimates.

Figure 5-1: Left: Locations of axial DTS fibers, at “GF” points vertically within the flow. Right: Example duct exit DTS fiber.

Prior work in this effort has shown that reasonable results can be obtained using relatively basic CFD methods [5-3], but that there is room for improvement. A focus for Run23 is made on investigating turbulence models and meshes in more detail. This should provide some insight into the accuracy achievable for a given computational cost, and help identify the current limitations of typical approaches for modeling this type of natural convection system. This should aid in using CFD as a predictive tool in the design and analysis of these systems.

5.1.2 Turbulence Modeling

The Reynolds-Averaged Navier-Stokes (RANS) approach was used for modeling turbulence in the NSTF. This was deemed appropriate for obtaining the mean values of interest. In prior analyses, turbulence modeling has generally been limited to basic two-equation models with adaptive wall modeling and relatively coarse “wall-function-type” (“high- y^+ ”) meshes given the physical size of the system and preliminary stages of the modeling effort. It is known however that refined wall meshes, sometimes called “low-Reynolds-type” (“low- y^+ ”) meshes, can provide improvements in wall temperature and shear stress estimates. These types of mesh are also in fact recommended for buoyancy-driven flows, as most wall functions are more appropriate for shear-driven flows [5-5]. For reference, y^+ is a non-dimensional distance from the wall, given by

$$y^+ = \frac{u_* y}{\nu} \quad (5-1)$$

where u_* is the friction velocity at the nearest wall, y is the distance from the wall, and ν is the kinematic viscosity of the fluid. Appropriate values of y^+ for a given mesh type are presented in Section 5.1.3.

The turbulence model used most in recent modeling is the Realizable k - ϵ Two-Layer model using the Xu Two-layer formulation [5-6][5-7]. This wall approach has been specifically designed for buoyancy-driven flows, and is recommended by the code manual to be used only in these cases [5-1]. For this reason, it is used as the “base” model for comparing against. In the current work, a number of other models are tested, as well as wall meshes. Adaptive “All- y^+ ” wall treatments were used throughout. The models tested include the Realizable k - ϵ Two-Layer model stated above, with either the Xu or Wolfstein wall treatments [5-8], the k - ω SST model with buoyancy production terms [5-9], the Standard k - ϵ low-Re model with a linear constitutive relation and the Yap correction (“Standard”) [5-10][5-11], and the Elliptic Blending k - ϵ model (“EB”) [5-12]. The last two models in particular have shown improvements for modeling flows with buoyancy over more basic models. They also were used only in conjunction with low- y^+ meshes.

Some expectations based on the physics can be established before simulation. In general the Two-Layer models and the high- y^+ meshes are more diffusive, and it should be expected that wall temperature estimates would be lower (i.e. predicted heat transfer coefficients will be higher). The flow in the cavity is completely buoyancy-driven and may benefit from a more advanced turbulence model. However, the flow in the ducts is high enough ($Re \sim 20,000$ and $Ri \sim 0.25$), that the heat transfer should be in the forced convection regime rather than the mixed or free convection regimes. This suggests that the coarse wall mesh approach could be successful for the primary air flow path. In the following sections, the air circulating within the cavity is termed the “cavity”, while the air flowing through the NSTF from the building to the outside is termed the “main”.

5.1.3 Meshing

The geometry used in the CFD simulations was generated directly from the design drawings, according to the “as-tested” specifications provided by ANL-SMR-8 [5-13]. This is shown in Figure 5-2. The flow inlet was placed at the start of the downcomer pipe, while the outlet was placed at the chimney pipe exit. STAR-CCM+ allows for unstructured meshing with polyhedral cells, which were used here. Appropriate mesh specifications were based on those established in prior work [5-3][5-14][5-15]. Two or more mesh cells were used across the riser duct width. A layer of prismatic cells was used orthogonal to all fluid-facing walls for improved modeling of heat transfer and turbulence. Conformal meshes were used at all interfaces, with an indirect mapping procedure used at a few non-conformal faces in certain cases.

Figure 5-2: Computational domain (L) and zoomed view of the outlet plenum (R).

Nearly all prior CFD work has employed “wall-function-type” meshes, with the first wall cell generally lying between $30 < y^+ < 300$. The current work uses some of these meshes, with y^+ generally closer to 30 in areas of interest, but “low-Re-type” meshes are also employed. These meshes have the goal of resolving the boundary layer, and generally have a first cell lying at $y^+ \sim 1$ or below. Four different meshes were used: 1) high- y^+ meshes in both the cavity and main air, 2) high- y^+ mesh in the cavity and low- y^+ mesh in the main, 3) low- y^+ mesh in the cavity and high- y^+ mesh in the main air, 4) low- y^+ meshes in both regions. These had roughly 5.4M cells, 26M cells, 22M cells, and 36M cells respectively. Examples of the mesh types can be seen in Figure 5-3. It should be noted that for the low- y^+ mesh in the main area, only the duct walls had this refined mesh; the rest of the domain (e.g. the chimney walls) was expected to be reasonably modeled with a wall mesh, and adaptive wall modeling should be able to handle the transition. Also it should be stated that only these specific types of meshes were tested; an advantage of many Two-layer models is the sufficient accuracy with reduced number of wall cells versus the low-Re models, but this aspect was not investigated in detail here.

Figure 5-3: Mesh examples for the low-y+ cavity high-y+ main (L) and high-y+ cavity low-y+ main (R) cases

5.2 Forced Flow Results and Discussion

5.2.1 Point Measurement Comparison

A listing of all of the turbulence cases run is provided in Table 5-1. The CFD simulations results are first compared with temperature measurements using thermocouples. The locations corresponding to TC numbers can be found in [5-13] (as shown in Figure 5-4 and Figure 5-5), but in general, increasing TC number corresponds to first lateral coordinate increases, then axial coordinate increases, with the largest temperature gradients axially.

Figure 5-6 to Figure 5-9 display the cavity wall surface temperature comparisons. Abbreviations are employed in the legends for the ensuing figures: for example, “MlowWolf_ChighXu” means that in the main region a low-y+ mesh was used with the Wolfstein Two-Layer k-epsilon model, while in the cavity region a high-y+ mesh was used with the Xu Two-Layer k-epsilon model. As anticipated, the predicted heated surface temperature is low and the side wall temperatures are high. The high-y+ mesh models have a flatter slope than in the experiment, while the more advanced models show better agreement. For the refinement in the main region, this is likely due to better capturing of the separation and developing flow at the entrance region of the riser. Most of the models are generally close to each other (within a spread of ~ 20 K) but there are two models whose trends are clear outliers: these are the cases using Xu with low-y+, either in the main or the cavity. Their temperature predictions are well below those of the other models because they are too diffusive, and hence over-predict the heat transfer.

Table 5-1: Database of all turbulence model and mesh refinement combinations used in the forced-flow modeling studies

Main Turbulence Model	Main Mesh y+	Cavity Turbulence Model	Cavity Mesh y+
Xu	High	Xu	High
Xu	Low	Xu	High
Xu	High	Xu	High
Wolfstein	Low	Xu	High
Wolfstein	High	Xu	Low
Wolfstein	High	Wolfstein	High
SST	High	Xu	High
SST	High	SST	High
SST	Low	Xu	High
EB	Low	Xu	High
Standard	Low	Xu	High
Xu	High	EB	Low
Xu	High	Standard	Low
Wolf	High	Standard	Low
Standard	Low	Standard	Low
EB	Low	EB	Low

Figure 5-10 and Figure 5-11 show the gas TC comparisons. The SST and Xu high-y+ main flow models are the closest to the TC values, with other advanced models reasonably close but consistently underestimating. Since the bulk exit temperature has essentially been fixed to be the same as in the experiment, this means that the temperature distributions within the duct are different for these models as compared to the others. Figure 5-13 and Figure 5-14 show the temperature measurements for TCs on the outer surface of Duct 9. Again, the two models mentioned previously give different results from the rest, with the main domain refinement case clearly being worse. Even for the advanced models, cavity refinement leads to lower duct surface temperatures. Using the Wolfstein model in the main domain works acceptably with cavity refinement. However, it is also seen that the cases with refinement in both areas are similar to the majority, so these lower temperatures appear to be due to a combination of the near-wall mesh itself in conjunction with the Xu model in the main domain.

Figure 5-4: TC locations for the heated cavity wall

Figure 5-5: TC locations for the cavity side wall

Figure 5-6: Heated surface temperatures for the Two-Layer and SST models

Figure 5-7: Heated surface temperatures for the Standard and EB models

Figure 5-8: Cavity side wall temperatures for the Two-Layer and SST models

Figure 5-9: Cavity side wall temperatures for the Standard and EB models

Figure 5-10: Duct outlet TC temperatures for the Two-Layer and SST models

Figure 5-11: Duct outlet TC temperatures for the Standard and EB models

Figure 5-12: Duct 9 outer surface TC locations.

Figure 5-13: Duct 9 outer surface temperatures for the Two-Layer and SST models

Figure 5-14: Duct 9 outer surface temperatures for the Standard and EB models

Pressure drop comparisons are provided in Figure 5-15 and Figure 5-16. These are a combination of gravity and frictional pressure contributions. It is clear that using the Xu model with refinement in the main gives very poor results, greatly over-predicting the shear stress and hence pressure drop. For the full natural convection simulations, this implies that this approach would underestimate the RCCS performance. Prior results did indeed underestimate the true mass flow rate in natural convection cases [5-3]. The pressure drop values from every other model are reasonably accurate, with all models underestimating. This should be somewhat expected since the ducts are modeled as smooth, while in actuality they will be somewhat rough, increasing the pressure drop. Note also that modeling the sidewall heat losses should decrease the temperature and increase the pressure drop in risers 1 and 12.

Figure 5-15: Duct pressure drops for the Two-Layer and SST models

Figure 5-16: Duct pressure drops for the Standard and EB models

Some conclusions can be drawn based on the results of the above comparisons. First, while using the Two-Layer Xu model with a high- y^+ mesh gives reasonable results, this serves only to hide the model's shortcomings that become apparent with refinement. The EB and Standard models give very similar results. High- y^+ and low- y^+ meshes produce more similar results for main refinement, while refinement in the cavity yields noticeable changes in temperatures throughout the system. The general results are relatively insensitive to the model selection with

the exception of only a few models. The Xu model appears to be the weakest of those tested, and at the very least should likely be abandoned in favor of the Wolfstein model in the main flow.

The shortcomings shown here of the Xu model, despite it being specifically designed for natural convection flows, led to further investigation of the validation for this model. It has been validated for parallel plate channel flow at a Rayleigh number lower than in the current system, and the rest of the validation was focused on vertical plates and enclosures [5-7]. This demonstrates that the model may need further developments and validations to be applicable to duct flows such as those found in the NSTF.

5.2.2 LUNA Fiber and 3D Comparison

Comparing the DTS fiber data with the simulation results allowed for further evaluation of modeling approaches. Figure 5-17 to Figure 5-19 provide comparisons with selected axial fibers in Duct 9 (refer to Figure 5-1). Again, with the exception of the low- y^+ main Xu model, the rest of the models generally display similar trends. The temperature increase is fairly linear going up the duct. The cases with main domain refinement show clear increased heat transfer near the inlet due to better capturing of the developing flow region. Agreement is good near the walls but worsens near the center.

Figure 5-20 to Figure 5-23 display comparisons with the outlet fiber across the Duct 9 exit. The placement of the fibers in these locations was specifically suggested by the analysis team, and the benefits for turbulence model evaluation are clear. It should be noted that the other ducts have similar experimental temperature profile shapes with only a few degrees' difference in magnitude. It is shown that the good agreement mentioned previously between the duct exit TCs with the SST models are because the SST models are too diffusive, and significantly under-predict the hot wall temperature. The Xu model is similar. The Wolfstein approach is promising as a method that can give reasonably good results quickly, as the shape is a clear improvement over the other models used with the high- y^+ mesh. There is little difference shown between the Wolfstein main cases when using either the Wolfstein or Xu models in the cavity.

The benefits of mesh refinement are clear when comparing the individual models for main and cavity refinements. Confirming the above conclusions, refinements with the Xu model lead to different results than for any of the other models, although this is more pronounced for the main region than the cavity. Mesh refinements greatly improve the shape of the temperature profile and increase the predicted duct wall temperatures. It is important to note that these results show that the specific turbulence model chosen has significantly less impact than the level of mesh refinement.

Figure 5-17: Axial fiber temperature comparison for the high-y+ mesh models. Solid lines are for fiber GF_E; dashed lines are for fiber GF_C

Figure 5-18: Axial fiber temperature comparison for models with main low-y+ meshes. Solid lines are for fiber GF-E; dashed lines are for fiber GF-C

Figure 5-19: Axial fiber temperatures for models with cavity low- y^+ meshes. Solid lines are for fiber GF-E, dashed lines are for fiber GF-C

Figure 5-20: Duct 9 exit fiber temperatures for the high- y^+ mesh models

Figure 5-21: Duct 9 exit fiber temperatures for models with main low- y^+ meshes

Figure 5-22: Duct 9 exit fiber temperatures for models with cavity low- y^+ meshes

Figure 5-23: Comparison of mesh and model approaches for Duct 9 outlet temperatures

3D figures of temperature and velocity at different points in the system are also provided to aid in understanding the effects of modeling choices on system behavior. There are no comparable 3D experimental data, so these plots are only for comparing different computational models to each other. The example comparisons presented here are between the MhighWolf_ChighXu model and the MlowEB_ClowEB model, and attempt to show some of the differences between the high- y^+ and low- y^+ approaches.

Figure 5-24 shows the velocity along the duct 9 exit DTS fiber line. Velocity is higher on the heated side. The average velocity is higher for the low- y^+ case. This is due to the fact that, as suggested by the higher pressure drop in the ducts nearest the cavity wall, there is more flow directed to these edge ducts in the high- y^+ case. The cases with main region refinement better capture the separation and flow development near the riser inlet, and should be expected to be more accurate in predicting the pressure drop. The fact that these models imply a more uniform distribution of pressure loss suggests that the edge ducts have elevated pressure drop due largely to thermal effects. As stated above, temperatures are overestimated for the cavity side walls in the simulations. Applying a more appropriate heat loss boundary condition would decrease those temperatures, and in turn decrease heat transfer to the edge ducts. This would decrease the bulk temperatures in the edge ducts and increase the density, leading to the elevated pressure drop seen in experiment.

Figure 5-24: Duct 9 riser outlet velocity comparison.

Figure 5-25 and Figure 5-26 demonstrate the profiles at the riser midplane for velocity magnitude and temperature, respectively. The most notable difference between the simulations is that for cases with cavity refinement, both the momentum and thermal boundary layers are significantly thinner. Heat transfer coefficient is reduced, leading to the higher duct wall temperatures. Similar plots for Duct 9 are provided in Figure 5-27. While there are some differences in the separation region near the riser inlet, the Duct 9 results are actually quite similar, as was also suggested by the DTS comparisons.

Figure 5-25: Velocity distribution at the riser axial midplane for the MhighWolf_ChighXu model (left) and the MlowEB_ClowEB model (right)

Figure 5-26: Temperature distribution at the riser axial midplane for the MhighWolf_ChighXu model (left) and the MlowEB_ClowEB model (right)

Figure 5-27: Axial velocity and temperature distributions at the Duct 9 midplane for the MhighWolf_ChighXu model (left) and the MlowEB_ClowEB model (right). Plots are 1/5 axial scale.

5.2.3 Conclusions from Turbulence Modeling and Mesh Studies

The Run023 forced flow test investigated here was helpful in choosing an appropriate approach for wall meshing and turbulence. It was shown that the Xu Two-Layer model used in prior work is not a good choice for the main air flow, as it gives poor results with refinement. This is suspected to be due in part to the flow in the risers being more in the forced convection heat transfer mode than the free convection mode. Additionally, the model may need more development and validation to be used confidently in high-Rayleigh flows. The Wolfstein model gives better results than the Xu model for similar computational effort, and should be considered for any future scoping or parametric tests given its simplicity. Despite its added complexity from solving additional differential equations, the Elliptic Blending model yielded nearly identical results to the Standard k-epsilon Low-Re model. It was also shown that a refined wall mesh is necessary to obtain the most accurate local results compared to the experimental data, but that pressure drops and bulk heat transfer to the main air can be reasonably well predicted using high- y^+ wall meshes. The refined mesh results also predict lower heat transfer coefficients and hence should be more accurate (and conservative) in predicting the heat removal capabilities of the RCCS, although this comes at a significantly increased computational cost.

5.3 Full Natural Convection Simulations

5.3.1 Base Case

The lessons learned from the turbulence and meshing studies were applied to Run022 at the baseline, 56 kW_t power level. This run was chosen because it features the full range of facility improvements, including the anti-downdraft chimney caps. The simulation was completely driven by natural convection, and serves to more rigorously gauge the appropriateness of the modeling approaches. Results shown here are for 0 Pa of pressure difference between the inlet and outlet, which essentially means that the case is entirely driven by the heat source effects on buoyancy, with no outside effects from wind or indoor/outdoor temperature differential. The hydrostatic head due to the height difference between inlet and outlet was included in the STAR-CCM+ simulations. The Wolfstein model with a high- y^+ mesh was used in the main flow, while the Xu model with high- y^+ mesh was used in the cavity. This is expected to provide acceptably accurate global values with moderate computational cost, based on the results presented in Section 5.2.

Results are presented and compared with the forced flow case results in Figure 5-28 to Figure 5-33. Most of them are quite similar, with the differences largely due to there being a roughly 2 K difference for the inlet flow temperatures. The simulation mass flow rate was 0.605 kg/s, which is less than 1% less than the experimental flow rate of 0.606 kg/s and is within the measurement uncertainty. Pressure drops in the risers also matched reasonably well with the experimental data (5.4% root mean square (rms) relative error), with there being only slight differences in the risers between forced and natural flow. Hence the code is successful at predicting the RCCS performance, specifically the flow rate and bulk temperature increase, for the base case. Cavity side wall temperatures are again significantly overestimated and heated surface temperature significantly underestimated, mainly due to the adiabatic boundary conditions. The results confirm that the current modeling approaches agree with the trends in the experimental data, and that the dual-pressure outlet approach for full natural convection simulation is appropriate. DTS temperature measurements also compare well with the CFD results, with 7.0% average differences for the GF-E fiber, 10% differences for the GF-C fiber, and 6.6% differences for the Duct 9 outlet fiber.

Figure 5-28: Heated surface temperatures for the Run22 high power case

Figure 5-29: Cavity side wall temperatures for the Run22 high power case

Figure 5-30: Duct outlet TC temperatures for the Run22 high power case

Figure 5-31: Duct pressure drops for the Run22 high power case

Figure 5-32: Axial fiber temperatures for the Run22 high power case. Solid lines are for fiber GF-E, dashed lines are for fiber GF-C

Figure 5-33: Duct 9 exit fiber temperatures for the Run22 high power case

5.3.2 Low Power Case

An additional simulation was performed at the lower power steady-state of the baseline test Run022. This further case is important to show that trends in power changes reproduce the appropriate trends in flow rate for the computational model. Results, provided in Figure 5-34 to Figure 5-40, again show relatively similar trends, albeit at lower temperature magnitudes, to those of the high power case.

A notable difference here is that the mass flow rate for the low power case is comparatively lower than for the high power case. The simulation flow rate was calculated to be 0.543 kg/s, which is more than 8% lower than the experimental value of 0.595 kg/s. For this reason, the main air temperatures are relatively higher than would be seen in the forced-flow low power case. It should be noted, however, that the ambient effects (as discussed in Section 3.2) have not been included in this model, which would enhance the system natural circulation flow rates (as seen in Section 4.2). It is also important that although there is a non-trivial change in the mass flow rate, cavity surface temperatures are very close to those of the forced flow case. This further supports the experimental conclusion that the heated surface temperatures are only minimally impacted for moderate changes in the main air flow rate. The DTS comparisons are again reasonable for the level of modeling fidelity chosen, and the forced-flow and natural convection temperature distributions are similar. There is 6.0% error for the GF-E fiber, 7.3% error for the GF-C fiber, and 5.3% error for the Duct 9 outlet fiber.

Figure 5-34: Heated surface temperatures for the Run22 low power case

Figure 5-35: Cavity side wall temperatures for the Run22 low power case

Figure 5-36: Duct outlet TC temperatures for the Run22 low power case

Figure 5-37: Duct 9 outer surface temperatures for the Run22 low power case

Figure 5-38: Duct pressure drops for the Run22 low power case

Figure 5-39: Axial fiber temperatures for the Run22 low power case. Solid lines are for fiber GF-E, dashed lines are for fiber GF-C

Figure 5-40: Duct 9 exit fiber temperatures for the Run22 low power case

5.4 Building Pressure and Weather Effects

It was seen in Section 5.3 that the mass flow rate was under-predicted in the computational results, notably in the low power case. Accounting for the outside weather will increase the flow rate in these cases. The outside temperature is significantly lower than the inside temperature during the steady-state periods in Run022, which will create a positive pressure head due to the stack effects in the NSTF building.

The theoretical basis of the weather effects is described in Section 3.2. Some adjustments are needed to apply it to the CFD model. The main issue is that there are significant pressure losses that are not modeled in the CFD simulation, notably the substantial losses for air leaking into the building. Thus the net pressure head must be modified to

$$\Delta P_{net} = P_s + \Delta P_w - \Delta P_{loss} = \rho\beta(T_{in} - T_{out})gH + \frac{C_w\rho_{out}V_w^2}{2} - \frac{K}{2}\rho_{in}V_{in}^2 \quad (5-1)$$

where K is the loss coefficient representing the building losses. In the CFD model, this loss is applied as a porous baffle in STAR-CCM+ [5-1] at the exit of the porous region. The two left-most terms in ΔP_{net} are calculated based on the weather data and applied as a boundary condition at the pressure inlet.

An appropriate value for the loss coefficient was ascertained from forced-flow simulations. The difference in piezometric pressure between the domain inlet and outlet was obtained for both the high and low power cases. As an illustration, if the piezometric pressure difference were zero, this would mean that the mass flow rate would be completely governed by the buoyancy source from the heated plate and the pressure losses of the NSTF. Thus, the weather and loss effects would cancel. If there was a higher piezometric pressure at the inlet in the forced flow case, this would mean that the weather effects would necessarily have to produce that amount of pressure head in excess of that subtracted by the loss coefficient in order to achieve the experimental flow rate, and vice-versa. Thus the piezometric difference can be regarded as ΔP_{net} in Equation 5-1. Using this equation, the average loss coefficient was calculated to be $K = 14.61$.

The effects on mass flow rate from applying the weather modeling are provided in Table 5-2. LUNA and TC data are very similar to the forced flow data presented above (as expected), thus are not plotted here. The impact shown on mass flow rate demonstrates that the weather modeling improves the flow rate estimates, particularly for the low power case. Understanding and application of the weather effects improves the potential for CFD to be a predictive tool in RCCS design and analysis.

Table 5-2: Mass Flow rate comparisons for natural convection simulation cases.

	High Power			Low Power		
	Experi- ment	Natural	Natural (Weather effects)	Experi- ment	Natural	Natural (Weather effects)
$\dot{m}$ (kg/s)	0.6058	0.6043	0.6078	0.5948	0.5431	0.5845
% difference	-	0.3	0.3	-	8.7	1.7

5.5 Summary of Findings and Accomplishments from CFD Modeling

The turbulence and meshing studies showed that under controlled forced-flow boundary conditions, the NSTF could be well modeled with CFD. The Wolfstein model with high- y^+ mesh in the main flow was found to be sufficient for predicting the bulk behavior of the system, while refinements in the cavity and ducts showed improved local temperature predictions, notably in the temperature profile of the air in the riser ducts. One recommendation is thus that the high- y^+ meshes can be used for scoping and design calculations, but a refined case should be run for more accurate (and conservative) wall temperature predictions.

The extension of modeling to full natural convection more rigorously confirmed that the NSTF could be well simulated by CFD methods. This was more of an *a priori* test, given that only the system geometry, wall heat flux, and indoor and outdoor temperatures were needed. The predicted mass flow rates were low for the low-power test, but were shown to be improved by accounting for the effects of ambient conditions. It is important to note, though, that the CFD simulations corroborated the experimental findings that changes in the main air flow have only a minor impact on the heated surface peak temperatures. This helps to demonstrate the robustness of the RCCS concept under normal operation. The findings from this work indicate that CFD can be used as a reliable modeling tool for RCCS design and analysis.

Some future work opportunities can still be considered for the NSTF CFD modeling. These primarily concern the “off-normal” cases, such as those with the blocked riser tubes or azimuthal power skew. While the normal behavior of the NSTF can be accurately modeled, simulating these other cases would result in a more encompassing view of modeling the range of phenomena and conditions that could be found in a full RCCS.

References

- [5-1] CD-Adapco, “STAR-CCM+ 10.06 Manual,” Melville, NY (2015).
- [5-2] A. Kraus, D. Lisowski, M. Bucknor, R. Hu, “Thermal-Hydraulic Simulation Framework for the NSTF and Preliminary Comparisons with Experimental Data,” *Proceedings of the American Nuclear Society Winter Meeting 2015*, Washington, D.C., USA, Nov. 8-12, 2015.
- [5-3] A. Kraus, R. Hu, D.D. Lisowski, M.D. Bucknor, “Simulation of Buoyancy-Driven Flow for Various Power Levels at the NSTF,” *Proceedings of ICONE-24*, Charlotte, NC, June 26-30, 2016.
- [5-4] Sutherland, “The Viscosity of Gases and Molecular Force,” *Philosophical Magazine*, 5, 36, pp. 507-531 (1893).
- [5-5] OECD/NEA, Committee on the Safety of Nuclear Installations, “Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications – Revision,” NEA/CSNI/R(2014)11 (2015).
- [5-6] Shih, T.-H., et al., “A New k - ϵ Eddy Viscosity Model for High Reynolds Number Turbulent Flows -- Model Development and Validation,” National Aeronautics and Space Administration Technical Memo 106721 (1994).

- [5-7] W. Xu, Q. Chen, F. T. M. Nieuwstadt, “A New Turbulence Model for Near-Wall Natural Convection”, *Int. J. Heat Mass Transfer*, **41**, pp. 3161-3176 (1998).
- [5-8] M. Wolfstein, “The Velocity and Temperature Distribution in One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient”, *Int. J. Heat Mass Transfer*, **12**, pp. 301-318 (1969).
- [5-9] Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Modeling for Engineering Applications”, *AIAA J.*, **32** (8), pp. 1598-1605 (1994).
- [5-10] F. S. Lien, W. L. Chen, M. A. Leschziner, “Low-Reynolds Number Eddy-Viscosity Modelling Based on Non-Linear Stress-Strain/Vorticity Relations,” *Proceedings of the 3rd Symposium on Engineering Turbulence Modelling and Measurements*, Crete, Greece, May 27-29 (1996).
- [5-11] C. Yap, “Turbulent Heat and Momentum Transfer in Recirculating and Impinging Flows”, Ph.D. Thesis, Faculty of Technology, University of Manchester, U.K. (1987).
- [5-12] F. Billard and D. Laurence, “A Robust $k\text{-}\epsilon\text{-}v^2/k$ Elliptic Blending Turbulence Model Applied to Near-Wall, Separated and Buoyant Flows”, *Int. J. Heat Fluid Fl.*, **33** (1), pp. 45–58 (2012).
- [5-13] D. Lisowski, M. Farmer, et al., “Design Report for the ½ Scale Air-Cooled RCCS Tests in the Natural convection Shutdown heat removal Test Facility (NSTF),” ANL-SMR-8, Argonne National Laboratory, June 2014.
- [5-14] S. Lomperski, W. D. Pointer, C. P. Tzanos, et al., “Generation IV Nuclear Energy System Initiative: Air-Cooled Option RCCS Studies and NSTF Preparation,” ANL-GenIV-179, Argonne National Laboratory Report (2011).
- [5-15] R. Hu and W. D. Pointer, “CFD Analyses of Natural Circulation in the Air-Cooled Reactor Cavity Cooling System,” *Proceedings of the International Conference on Mathematics and Computational Methods Applied to Nuclear Science & Engineering (M&C 2013)*, Sun Valley, Idaho, USA, May 5-9, 2013.

6 Conclusions

The NSTF analysis effort has contributed to the design, operation, and evaluation of the RCCS concept for passive safety in advanced reactors. To support the experimental NSTF program, significant computational modeling efforts were performed including the direct experimental program support, study of ambient condition effects, 1D system-level and 3D CFD benchmark simulations of the conducted tests. The computational models were improved in FY16 based on the findings from the ambient condition effect study and the results from previous simulations. Through the analysis effort, important findings have been found in the computational modeling of the NSTF and RCCS. With the improved understanding of the physics phenomena in NSTF and the improved modeling approaches, good agreements have been achieved between the simulation results and experimental data.

Significant efforts were devoted to improve the understanding of the weather effects on NSTF system flow behavior. It was found that temperature difference between the outside and building interior as well as wind speed have strong impacts on NSTF system flow rates. An empirical model was developed to model the stack pressure effects and the wind effects on system flow rates, in which the coefficients were obtained based on best fitting the experimental data. The predictive capability of the empirical model is demonstrated by applying it to the new set of experimental data.

The RELAP5-3D model of the NSTF was then updated to properly capture the effects of outdoor air temperature, wind, building temperature, and chimney caps on the performance of the facility. The overall behavior of the facility was captured well by the model in all of the scenarios simulated, which demonstrated the robustness of the model for a wide range of ambient conditions. The ability of the model to capture these effects provides confidence that the natural circulation air-flow behavior in NSTF can be well captured by RELAP5 simulations and its usage could provide insights into the performance of the RCCS under various reactor system conditions, which would be required during reactor licensing.

An in-depth study of refined wall meshes and turbulence models has been undertaken in the CFD analysis of the NSTF with controlled force-flow boundary conditions. The turbulence and meshing studies showed that some of the models used in prior work were actually too diffusive, and under-predicted the duct wall temperatures. Recommendations were then made for the best models for either coarse wall mesh or fine wall meshes, given the level of accuracy desired and computational time available. The extension of the CFD modeling to full natural circulation cases more rigorously confirmed that the NSTF could be well simulated by CFD methods. Accounting for the effects of ambient conditions, the CFD simulations predicted the natural circulation flow rates very well. Additionally, CFD simulations corroborated the experimental findings that changes in the main air flow have only a minor impact on the heated surface peak temperatures, which helps to demonstrate the robustness of the RCCS concept under normal operation.

The collaboration between the experimentalists and computational analysts has been vital for improving the knowledge of the system, as well as strengthening the experimental program. Improvements on the instrumentation, test procedures, and the test facility were identified through the analysis efforts. This mutually beneficial relationship has become integral to the overall program objective of examining the heat removal performance of the RCCS concept, and will continue to benefit the water-cooled RCCS study.

Acknowledgements

The authors sincerely thank our former colleagues for their previous work on the system thermal-hydraulics and CFD analyses of the RCCS and NSTF, including Drs. Steve Lomperski, Tom Wei, W. Dave Pointer, Constantine Tzanos, and many summer students. The authors would like to express their gratitude to Drs. Mitch Farmer and Craig Gerardi for their valuable comments and advice throughout this work. This work is funded by U.S. Department of Energy Office of Nuclear Energy's Advanced Reactor Technologies (ART) program. We gratefully acknowledge use of the Blues cluster in the Laboratory Computing Resource Center at Argonne National Laboratory.

Bibliography

- Billard, F., and Laurence, D., “A Robust $k-\varepsilon-v^2/k$ Elliptic Blending Turbulence Model Applied to Near-Wall, Separated and Buoyant Flows”, *Int. J. Heat Fluid Fl.*, **33** (1), pp. 45–58 (2012).
- Bucknor, M., Hu, R., Lisowski, D., Kraus, A., “Comparisons of RELAP5-3D Analyses to Experimental Data from the Natural Convention Shutdown Heat Removal Test Facility”, *Proceedings of ICAPP'16*, San Francisco CA, April 17-20, 2016.
- CD-Adapco, “STAR-CCM+ 10.06 Manual,” Melville, NY (2015).
- Hu, R., Kraus, A, Bucknor, M., et al., Argonne National Laboratory, unpublished information, September 2015.
- Hu, R., and Pointer, W. D., “CFD Analyses of Natural Circulation in the Air-Cooled Reactor Cavity Cooling System,” *Proceedings of the International Conference on Mathematics and Computational Methods Applied to Nuclear Science & Engineering (M&C 2013)*, Sun Valley, Idaho, USA, May 5-9, 2013.
- Incropera, F. P., and DeWitt, D. P., *Fundamentals of Heat and Mass Transfer*. New York: J. Wiley, 2002.
- Kraus, A., Hu, R., Lisowski, D.D., Bucknor, M.D., “Simulation of Buoyancy-Driven Flow for Various Power Levels at the NSTF,” *Proceedings of ICONE-24*, Charlotte, NC, June 26-30, 2016.
- Kraus, A., Lisowski, D., Bucknor, M., Hu, R., “Thermal-Hydraulic Simulation Framework for the NSTF and Preliminary Comparisons with Experimental Data,” *Proceedings of the American Nuclear Society Winter Meeting 2015*, Washington, D.C., USA, Nov. 8-12, 2015.
- Lien, F. S., Chen, W. L., Leschziner, M. A., “Low-Reynolds Number Eddy-Viscosity Modelling Based on Non-Linear Stress-Strain/Vorticity Relations,” *Proceedings of the 3rd Symposium on Engineering Turbulence Modelling and Measurements*, Crete, Greece, May 27-29 (1996).
- Lisowski, D., Farmer, M., et al., “Design Report for the 1/2 Scale Air-Cooled RCCS Tests in the Natural convection Shutdown Heat Removal Test Facility (NSTF)”, ANL-SMR-8, Argonne National Laboratory, June 2014.
- Lisowski, D., Gerardi, C., Kilsdonk, D., et al., Argonne National Laboratory, unpublished information, September 2015.
- Lisowski, D.D., Kraus, A.R., Bucknor, M.D., Hu, R., Farmer, M.T., “Experimental observations of natural circulation flow in the NSTF,” *Nuclear Engineering and Design*, Vol. 306, 124-132, (2016). <http://dx.doi.org/10.1016/j.nucengdes.2016.01.014>
- Lomperski, S., Pointer, W. D., Tzanos, C. P., et al., “RCCS Studies and NSTF Preparation Air-Cooled Option,” ANL-GenIV-142, Argonne National Laboratory, September 2010.
- Lomperski, S., Pointer, W. D., Tzanos, C. P., et al., “Generation IV Nuclear Energy System Initiative: Air-Cooled Option RCCS Studies and NSTF Preparation”, ANL-GenIV-179, Argonne National Laboratory, September 2011.
- Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Modeling for Engineering Applications”, *AIAA J.*, **32** (8), pp. 1598-1605 (1994).

OECD/NEA, Committee on the Safety of Nuclear Installations, “Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications – Revision,” NEA/CSNI/R(2014)11 (2015).

Shenoy, A., General Atomics (retired), Personal Communications, February 2016.

Shih, T.-H., et al., “A New k - ϵ Eddy Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation,” National Aeronautics and Space Administration Technical Memo 106721 (1994).

Sutherland, “The Viscosity of Gases and Molecular Force,” *Philosophical Magazine*, 5, 36, pp. 507-531 (1893).

The MathWorks, Inc., MATLAB and Statistics Toolbox Release 2013b, Natick, Massachusetts, United States, 2013.

The RELAP5-3D Code Development Team, “RELAP5-3D Code Manual,” Idaho National Laboratory, Idaho Falls, ID, INEEL-EXT-98-00834, Rev. 4, 2012.

Tsilingiris, P.T., “Thermophysical and transport properties of humid air at temperature range between 0 and 100 °C,” *Energy Conversion and Management*, Vol. 49, pp. 1098–1110, 2008.

Walker, I.S., and Wilson, D.J., “Evaluating Models for Superposition of Wind and Stack Effect in Air Infiltration,” *Building and Environment*, Vol. 28, No. 2, pp. 201-210, 1993.

Wolfstein, M., “The Velocity and Temperature Distribution in One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient,” *Int. J. Heat Mass Transfer*, **12**, pp. 301-318 (1969).

Xu, W., Chen, Q., Nieuwstadt, F. T. M., “A New Turbulence Model for Near-Wall Natural Convection,” *Int. J. Heat Mass Transfer*, **41**, pp. 3161-3176 (1998).

Yap, C., “Turbulent Heat and Momentum Transfer in Recirculating and Impinging Flows”, Ph.D. Thesis, Faculty of Technology, University of Manchester, U.K. (1987).

Nuclear Engineering Division

Argonne National Laboratory
9700 South Cass Avenue, Bldg. 208
Argonne, IL 60439

www.anl.gov

**U.S. DEPARTMENT OF
ENERGY**

Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC