Investigation of Plenum-to-Plenum Heat Transfer and Gas Dynamics under Natural Circulation in a Scaled-Down Dual Channel Module Mimicking Prismatic VHTR core using CFD

M. T. Kao*[1], P. Jain[2], and S. Usman[3]
mkao2@illinois.edu; jainpk@ornl.gov; usmans@mst.edu

I.A. Said[4], M. M. Taha[4]
iasvx4@mst.edu; mmt8t6@mst.edu

M. H. Al-Dahhan[4], Rizwan-uddin[1]
aldahhanm@mst.edu; rizwan@illinois.edu

1. Department of Nuclear, Plasma and Radiological Engineering, University of Illinois at Urbana-Champaign
2. Oak Ridge National Lab
3. Department of Mining and Nuclear Engineering, MST, Rolla
4. Department of Chemical Engineering, MST, Rolla

ABSTRACT

In this study, two CFD packages, STAR-CCM+ and COMSOL were used to simulate scaled experimental facilities mimicking a dual channel natural circulation behavior in VHTR's and mimicking prismatic VHTR core and plena operation under natural circulation conditions to simulate the natural flow conditions in P2P, identify stream line, and velocity variations in the upper plenum and the two channels at given initial conditions for two different cases of 0.8 MPa and 4.2 MPa. A dual channel module represents a scaled down prismatic block reactor mimicking OSU-HTTF with a scaling down ratio of 1/4 axially and 1/4 radially with a diameter similar to OSU-HTTF cooler channel (0.625 inch) and length of channel is identical to 1/4 to the length of channels of OSU-HTTF of reactor core diameter 1 foot with reference OSU-HTTF. Dual-channel module was meshed with 1 million cells for the two cases. Simulations were carried out for Helium at pressure values of 0.8 MPa and 4.2 MPa.

KEYWORDS
CFD; OSU-HTTF; Missouri S&T; Natural circulation; VHTR

1. INTRODUCTION

Very High Temperature Reactor (VHTR) is considered to be the leading candidate in the list of Next Generation of Nuclear Plants (NGNP) released by the Generation IV International Forum (GIF) in 2000. The NGNP are expected to be more safe, secure, sustainable, competitive, and versatile than previous generations. These designs are also expected to achieve higher burnup relative to current nuclear power plants. Moreover, some of these reactors are designed to burn existing nuclear waste while producing electricity [1].

Computational tools including CFD codes are being used in the design and analyses of these new reactor concepts. However, before proceeding with the use of these codes, they must be validated with appropriate and relevant experimental data. In addition to being used to validate codes, experimental data play an important role in understanding the geometry and design specific phenomena expected to occur in these new nuclear power plant designs. Once validated, computational tools enable us to optimize the design [2].

Our long term project is aimed at developing a database of experimental data to validate the CFD codes for the design and analyses of VHTR. At this early stage of our project, CFD codes are being used to help in the design of the experimental facility and its instrumentation.

Several CFD studies and code validation investigations have been reported for prismatic VHTR. The design features of prismatic VHTR allow the coolant to flow in the interstitial gaps between hexagonal fuel blocks. This phenomenon was analyzed by Sato et al. [3] and Yoon et al. [4]. Yoon et al. used experimental results to validate CFD codes for bypass flow analysis. The performance of the VHTR cooling system was assessed by Kim et al. by considering internal vessel cooling, external vessel cooling, and internal insulation during normal and accident conditions, using a system thermo-fluid analysis code, GAMMA+, and a commercial CFD code, CFX [5]. The experimental data obtained by Groehn [6] was used by Lee et al. to validate the CFD code CFX-12, in their study of the velocity and pressure distributions in the cross gaps [7].

Although CFD has received a great deal of attention for understanding and predicting the flow behavior in VHTR, sufficient validation of these codes has not been carried out [7]. In the current study, preliminary results obtained using CFD code, STAR-CCM+ and COMSOL, are reported. These results are being used to help design the experimental facility, and in determining suitable locations for placing novel and sophisticated sensors that will be used in the Missouri S&T scaled-down test facility.

2. MISSOURI S&T EXPERIMENTAL FACILITY

Missouri S&T is developing a new facility to mimic the natural circulation behavior in VHTR's prismatic core under accident conditions. It will be used for gas dynamics and heat transfer measurements. This dual-channel facility (phase one of the project) is a 1:4 scaled down facility based on OSU-HTTF design [8, 9]. Dimensions of the dual-channel reactor are listed in Table 1.

Table 1: Dimensions of (phase one) Missouri S&T facility (Scaled down model)

Parameter	Missouri S&T Facility
Tube Diameter (inch)	0.625
Tube length(inch)	23.228
Core diameter(inch)	11.78
Upper plenum height (inch)	9.41
Outer vessel diameter (inch)	15
Number of channels	Two (one for up-ward and one for down-ward flow)

This module is being designed to mimic one horizontal layer of a group of seven blocks, with one central block surrounded by six blocks. Two vertical cylindrical channels will be used (one for upward flow in the central block and one for downward flow in one of the six outer blocks) with a diameter similar to OSU-HTTF coolant channel (0.625 inch). Length of the channel is equal to 1/4 of the length of the channels in the OSU-HTTF facility (reactor core diameter of one foot).

A CAD drawing and schematic diagrams of the scaled down dual-channel Missouri S&T facility are shown in Figures 1 and 2.

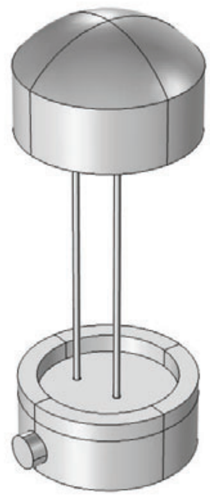

Figure 1. A CAD drawing of the facility

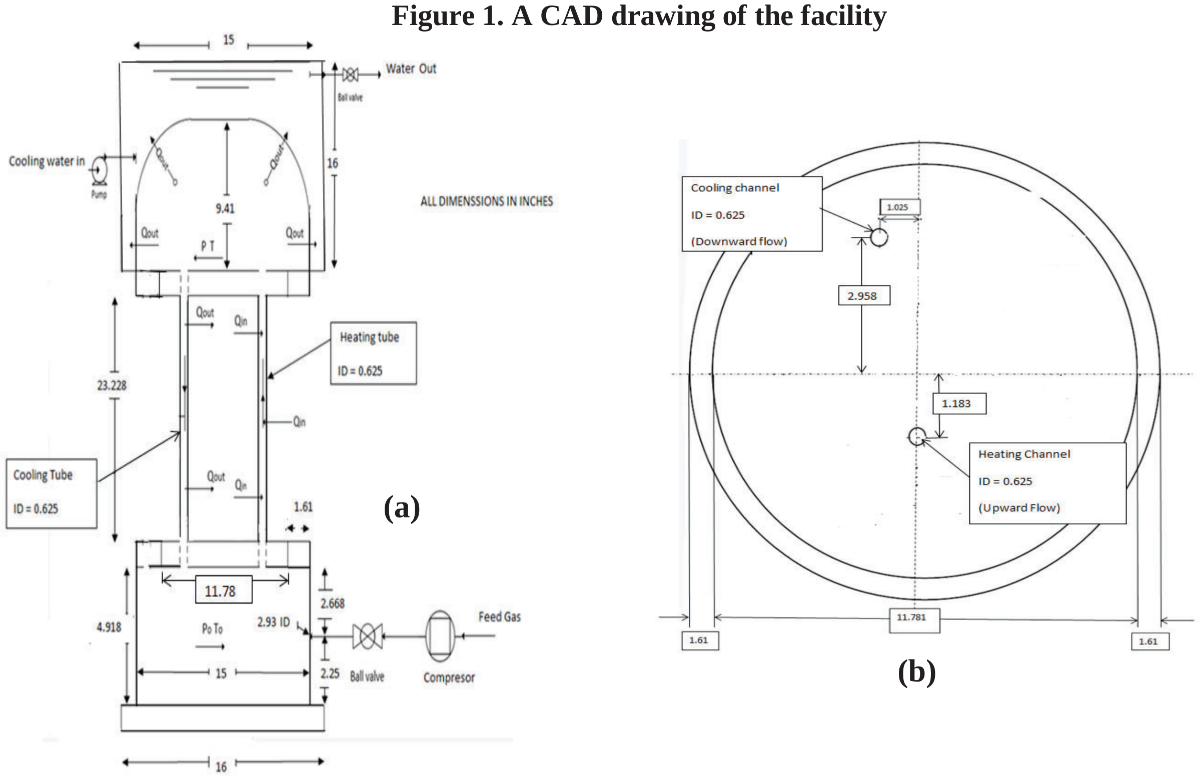

Figure 2. (a) Schematic diagram for Missouri S&T scaled-down dual-channel facility, (b) Top view of the Missouri S&T scaled-down dual-channel facility

A picture of the experimental facility under construction is shown in Fig. 3. Picture also shows the chiller to be used to extract the heat from the upper plenum.

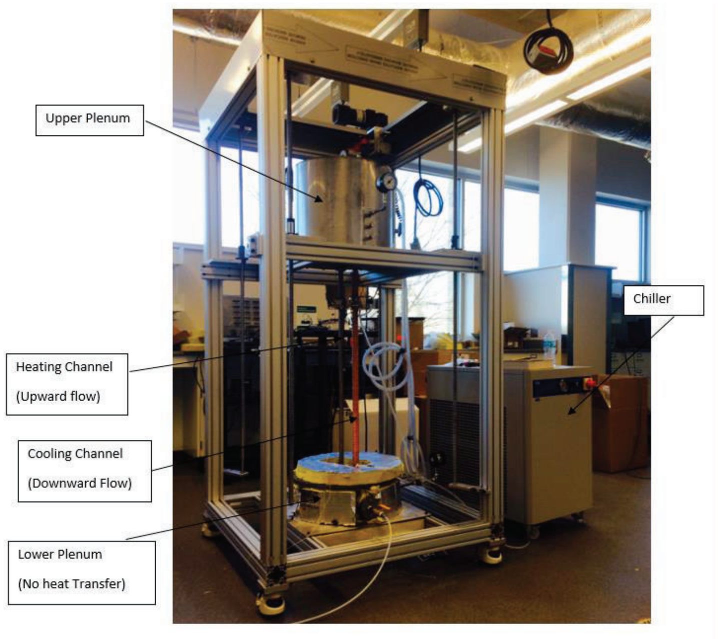

Figure 3. Experimental facility at Missouri S&T labs

3. DESCRIPTION OF WORK

We here report the results of our preliminary CFD studies carried out to help guide the design process; and to determine the best locations for instrumentation. In the current study the commercial CFD code, STAR-CCM+ and COMSOL were used to simulate the natural flow conditions between the two plena of the facility. Flow conditions are simulated for Helium for operational conditions at 0.8MPa and 4.2 MPa. The average velocity at the mid plane of the two channels and the total heat transfer rates at the top surface and in both channels were monitored during the iterations. When the variations of the observed quantities between two successive iterations are less than 0.05 %, the simulations are considered converged. For the case Helium under 0.8MPa, the Reynolds number in both channels is around 200 but at upper plenum region, the Reynolds number could be much higher. Hence, both laminar and turbulence models were used for 0.8 MPa case. For Helium under 4.2 MPa, the Reynolds number in both channels is around 3,300 and therefore only turbulence models were used in the simulation. Different turbulence models were used. Results reported in this work were obtained using the Realizable k-epsilon model with two-layer wall treatment approach. Realizable k-epsilon model includes a new transport equation for turbulent dissipation rate. (The term *realizable* is used to imply that the model satisfies certain mathematical constraints on the normal stresses. Instead of using the low-Reynolds number model, the two-layer approach allows the k-epsilon model to be applied in the viscous sublayer region.) The grid independent tests were conducted for all conditions to ensure that the differences between two successively refined meshes were below 4 %. 5.3 million and 5.4 million cells were used for 0.8 MPa and 4.2 MPa cases, respectively. The temperature and pressure dependent density is calculated using (Petersen, 1970) :

$$\rho = 48.14\frac{P}{T}\left[1+0.4446\frac{P}{T^{1.2}}\right]^{-1} \tag{1}$$

Results obtained using STAR-CCM+ are presented first in Sec 3.1. Results obtained using COMSOL are shown at the end in Fig. 24.

3.1. Results for system pressure of 0.8 MPa

Geometry of the dual-channel facility being constructed at Missouri S&T was meshed in STAR-CCM+ with 5.3 million cells. The mesh and a zoomed view of the upper and low plena of the dual-channel facility are shown in Figure 4. The postulated boundary conditions are outlined in Figure 5.

Results obtained using the laminar model for Helium at 0.8 MPa are presented below. The density distributions are shown in Figures 6 and 7. Since the surface temperature of the upper plenum (10 ℃) is much lower than the wall temperatures of hot (330 ℃) and cold (270 ℃) channels, the flow actually receives heat from both channels and loses heat only at the chiller surface (10 ℃) as shown in Figure 6. The upward flow coming out of the hot channel is fast enough that it reaches the top surface of the chiller dome, and then spreads out, losing heat to the cold chiller surface. It is noted that the highest velocity is near the exit of the hot channel. Average velocity in the hot channel is 0.31 m/s, a little higher than the average velocity of 0.29 m/s in the cold channel as shown in Figures 8 and 9. Figure 9 shows that the flow is, as expected, going upward in the hot channel and downward in the cold channel. The temperature distribution is presented in Figure 10. The average fluid temperature close to the top of the chiller surface is around 37 ℃ when using the laminar flow model.

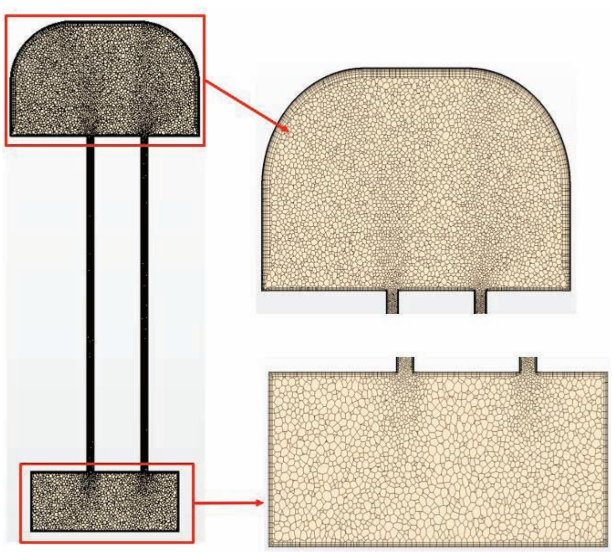

Figure 4. Mesh structure of the dual-channel facility (P = 0.8 MPa)

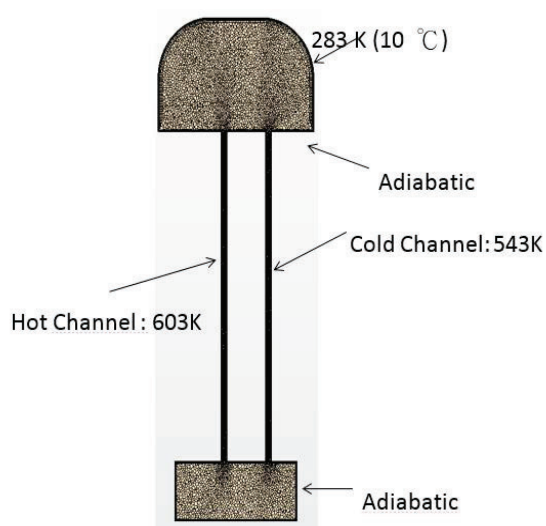

Figure 5. Boundary conditions for plena and tubes

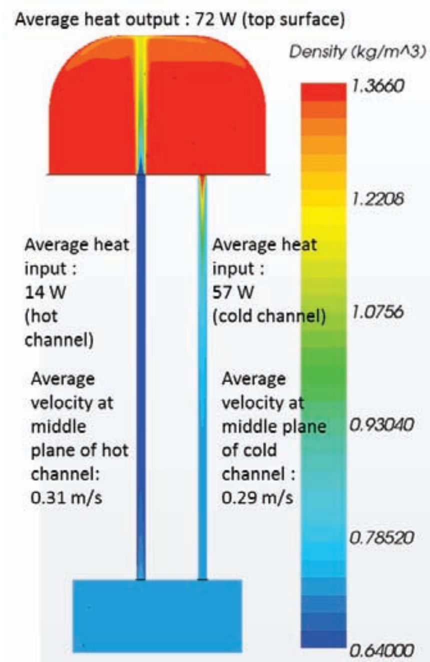

Figure 6. Density distribution obtained using laminar flow model (P = 0.8 MPa)

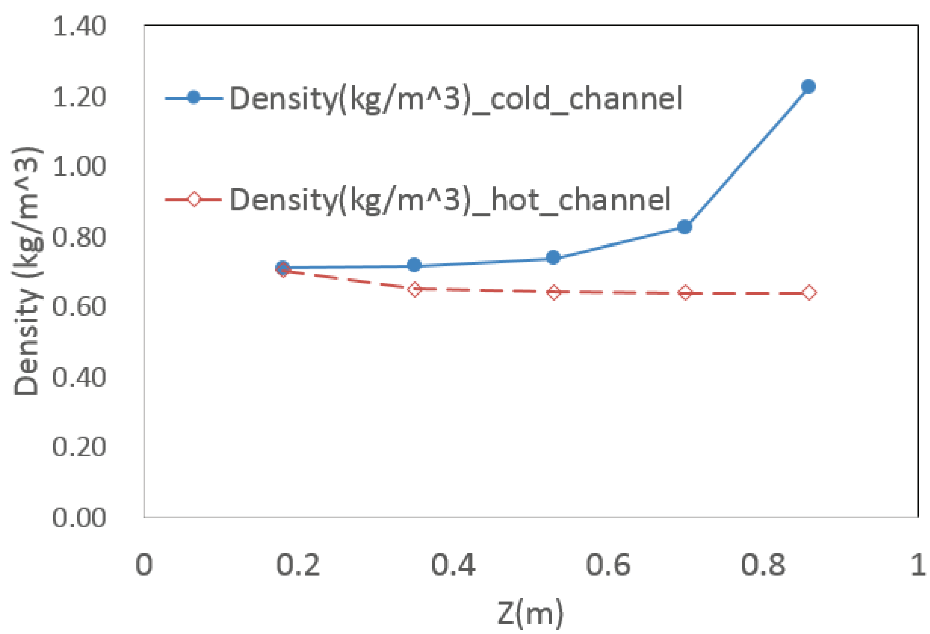

Figure 7. Density distribution along the cold and hot channels obtained using the laminar flow model (P = 0.8 MPa)

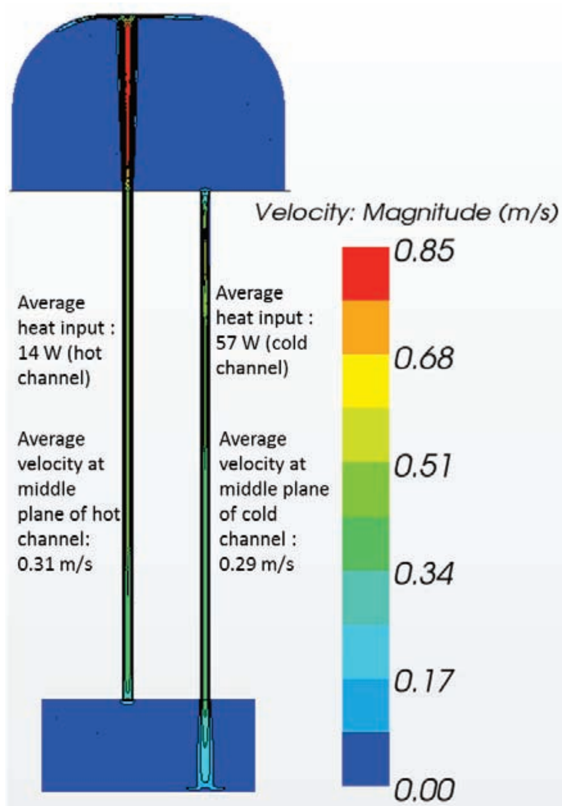

Figure 8. Velocity distribution obtained using the laminar flow model (P = 0.8 MPa)

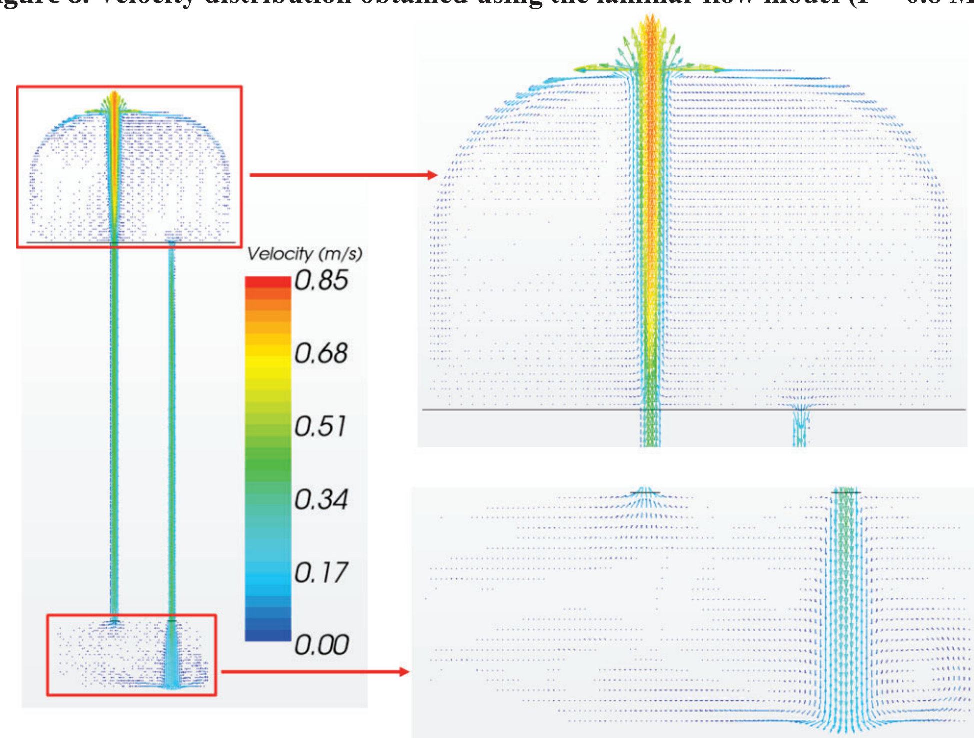

Figure 9. Velocity vectors obtained using the laminar flow model (P = 0.8 MPa)

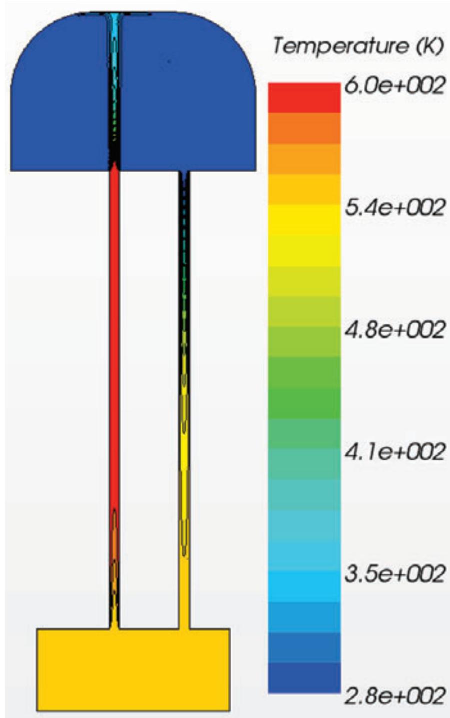

Figure 10. Temperature distribution obtained using the laminar flow model (P = 0.8 MPa)

Simulation results shown in Figures 11 through 23 are obtained using the Realizable Two-Layer k-ε turbulence model. The density distribution through the two channels and the two plena are shown in Figures 11 and 12. It can be seen that the trend of density variations in the channels is the same as that obtained using the laminar model. In addition, heat transfer rates on the top surface and in both channels are a little lower than the values predicted using the laminar flow model. This is because of the fact that, as seen in Figure 13, the average fluid temperature near the top of the chiller surface when using the turbulence model is around 25 ℃, which is lower than the average fluid temperature (37 ℃) predicted when using the laminar model. .

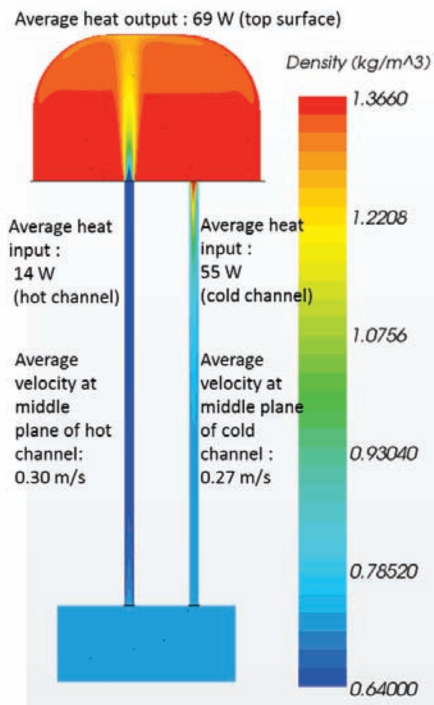

Figure 11. Density distribution obtained using the Realizable Two-Layer k-ε model (P = 0.8 MPa)

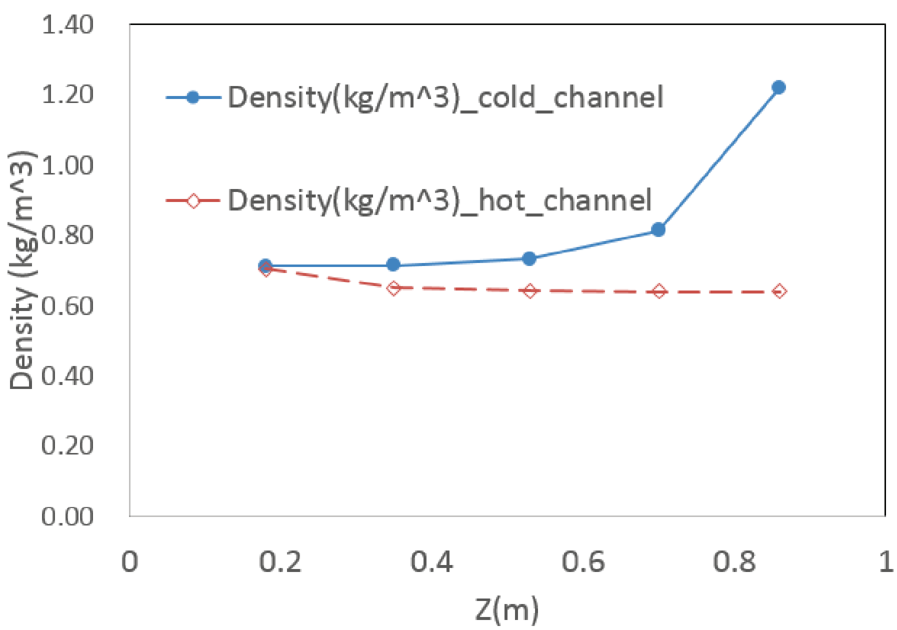

Figure 12. Density distribution obtained using the Realizable Two-Layer k-ε model (P = 0.8 MPa)

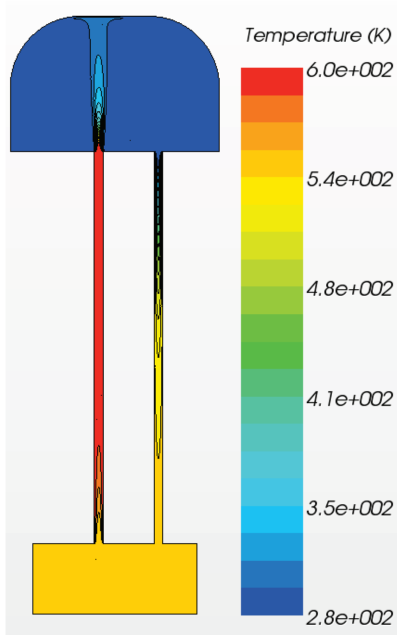

Figure 13. Temperature distribution obtained using the Realizable Two-Layer k-ε model (P = 0.8 MPa)

Figures 14, 15 and 16 show the velocity distributions. Strong lateral velocity (~0.3 m/s) can be observed close to the top surface. Here, the predicted average velocities (0.30 and 0.27 m/s) in the two channels are a little lower than the values predicted when using the laminar flow model.

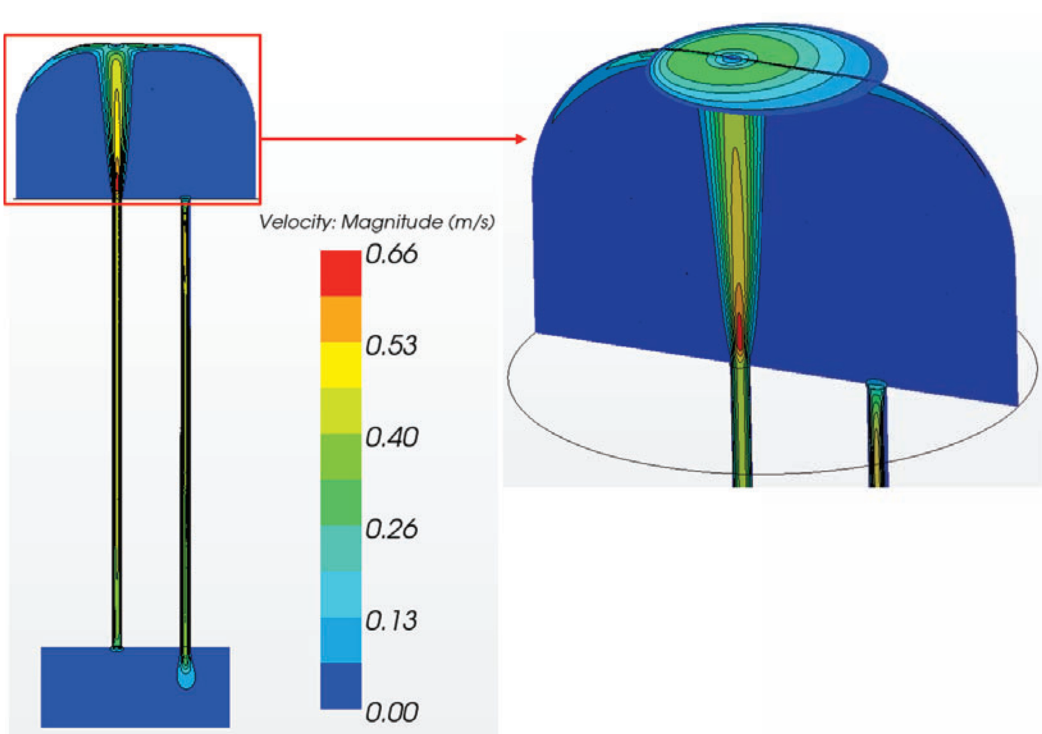

Figure 14. Velocity distribution obtained using the Realizable Two-Layer k-ε model (P = 0.8 MPa)

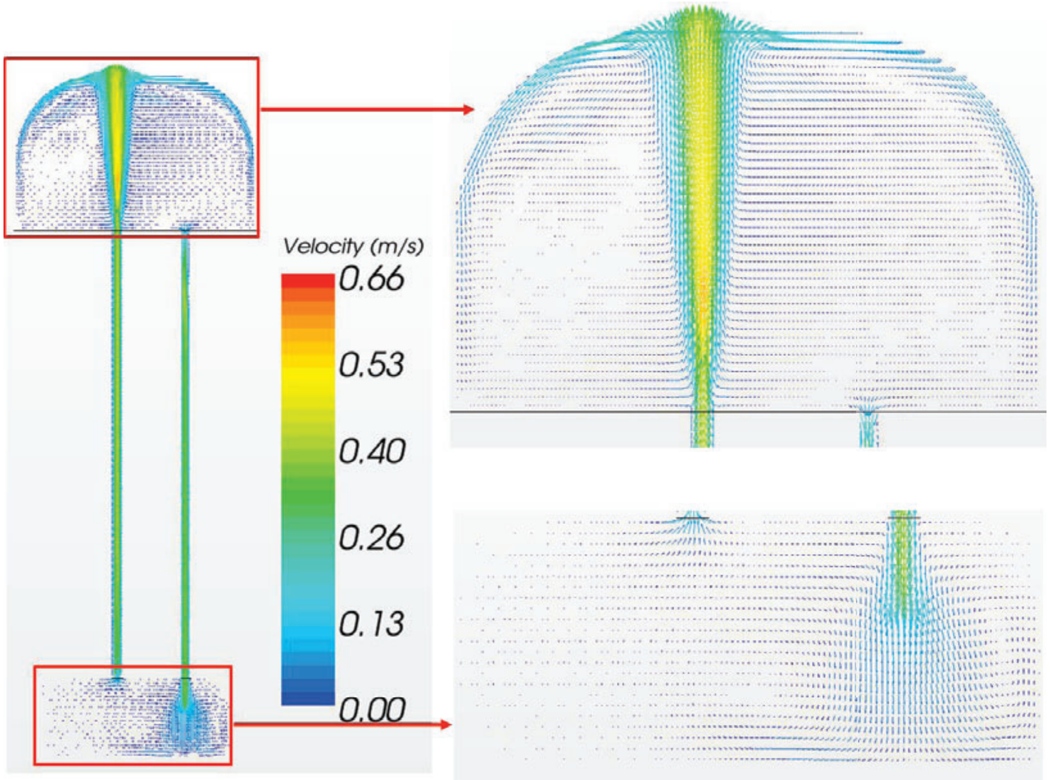

Figure 15. Velocity vectors obtained using the Realizable Two-Layer k-ε model (P = 0.8 MPa)

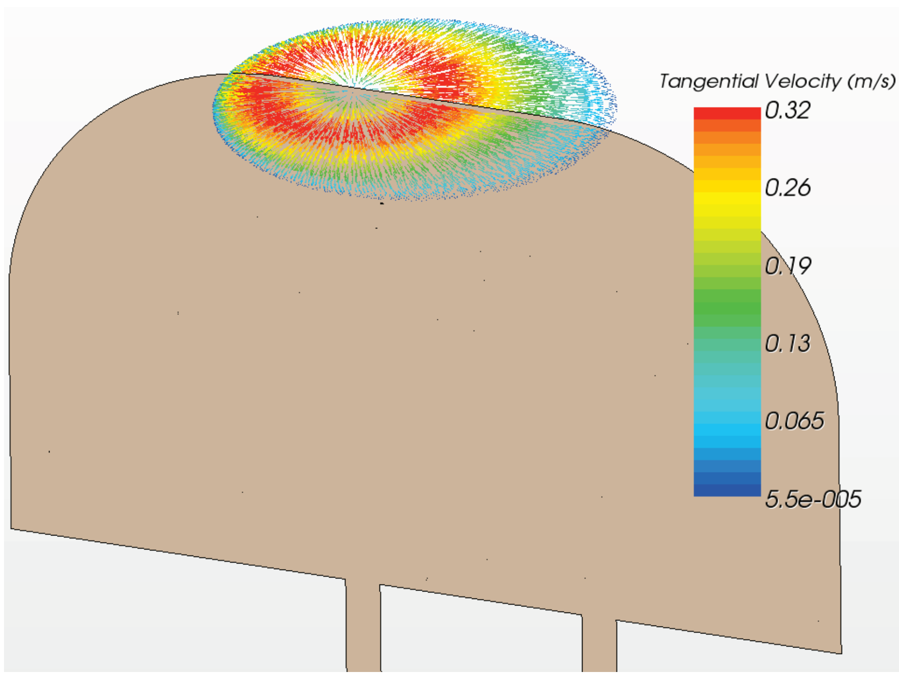

Figure 16. Lateral velocity near the top surface obtained using the Realizable Two-Layer k-ε model (P = 0.8 MPa)

3.2. Results for system pressure of 4.2 MPa

Simulations at the higher pressure required a finer mesh for convergence (5.4 million). Mesh for the 4.2 MPa case is shown in Figure 17.

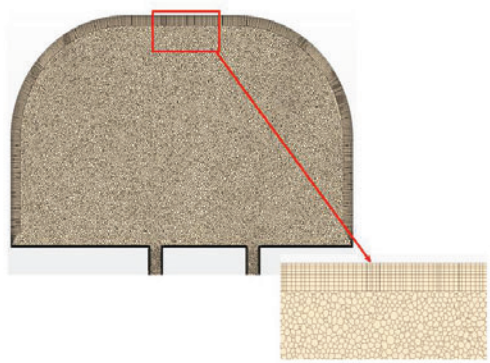

Figure 17. Mesh in the upper plenum for the Realizable Two-Layer k-ε model for 4.2 MPa case (5.3 million)

The simulation is carried out using the Realizable Two-Layer k-ε turbulence model. A summary of the major results are shown in Figures 18 and 19. It is noteworthy to mention that the density is 5 ~ 7 times higher than in the 0.8 MPa case.

The average velocity is found to be 0.81 m/s in the hot channel and 0.64 m/s in the cold channel, both of them are 2.6 times higher than in the 0.8 MPa case. Therefore, the simulation results show that the total heat transfer rate for the 4.2 MPa case (Figure 18, heat output = 1076 W) is 15-16 times higher than in the 0.8 MPa case (Figure 11, heat output = 69 W) due to the denser Helium gas and higher velocities in the two channels.

The temperature distribution obtained using the Realizable Two-Layer k-ε model is shown in Figure 20. It is found that at 4.2 MPa, the heat transfer rate in the cold channel (701 W) is almost twice as high as in the hot channel (372 W).

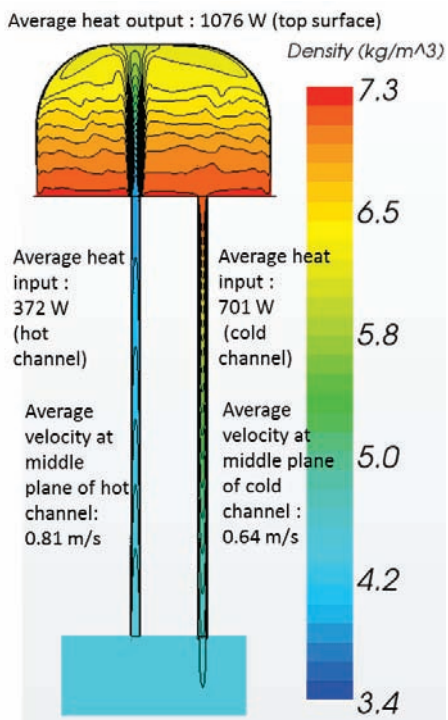

Figure 18. Density distribution for the 4.2 MPa case, obtained using the Realizable Two-Layer k-ε model

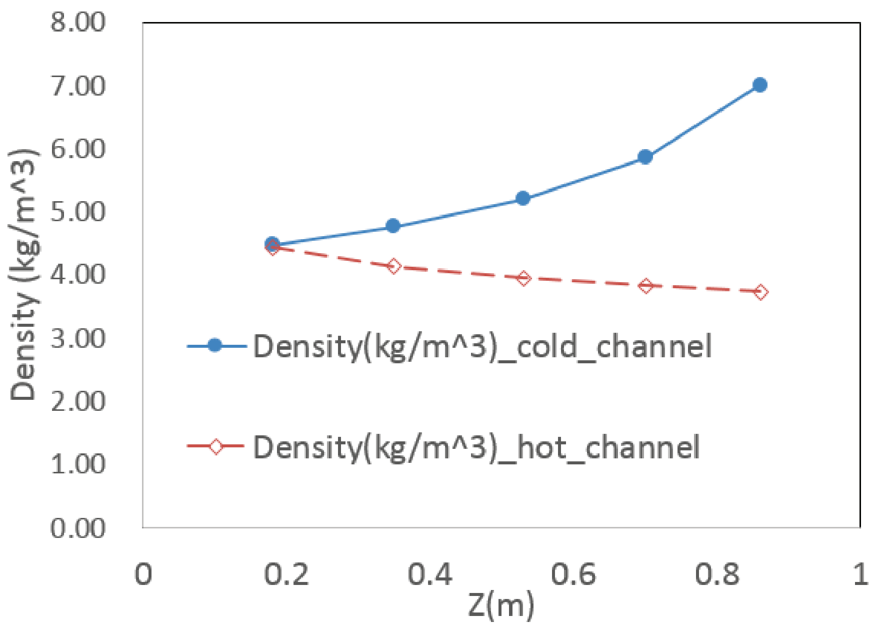

Figure 19. Density distribution along the two channels for the 4.2 MPa case, obtained using the Realizable Two-Layer k-ε model

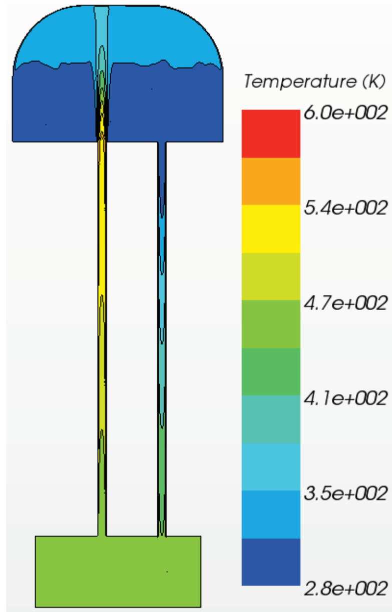

Figure 20. Temperature distribution obtained using the Realizable Two-Layer k-ε model (P = 4.2 MPa)

Figures 22 and 23 show the velocity distribution in the dual-channel facility, with average velocity of 0.64 m/s in the cold channel and 0.81 m/s in the hot channel. Average velocities in both channels are higher than the 0.8 MPa case and the lateral velocity near the top surface is around 0.25 m/s.

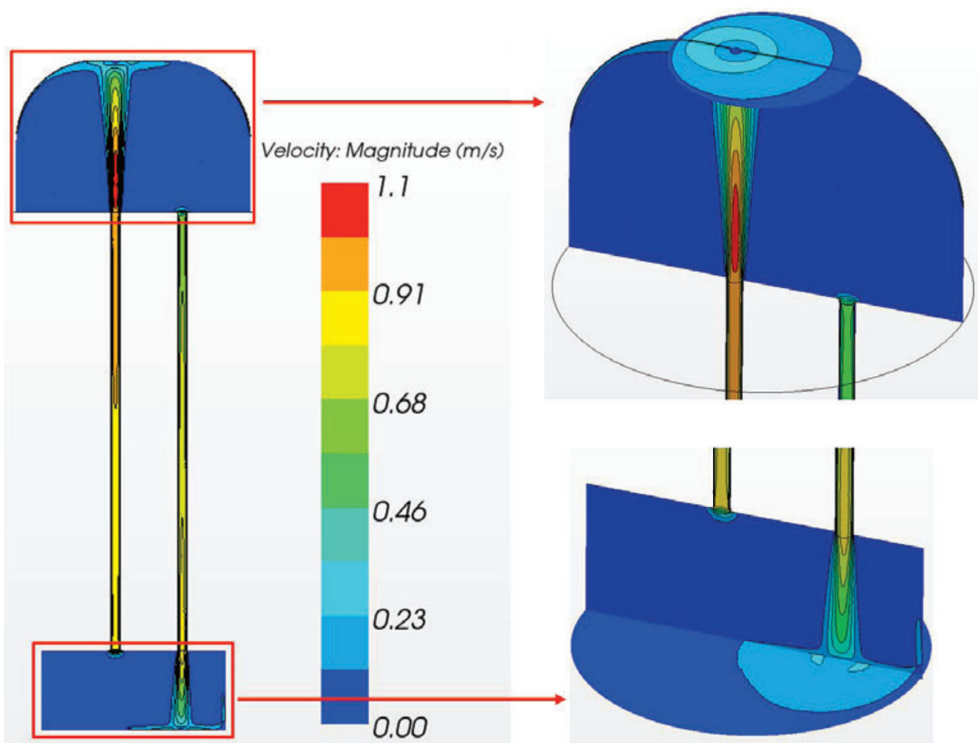

Figure 21. Velocity distribution obtained using the Realizable Two-Layer k-ε model (P = 4.2MPa)

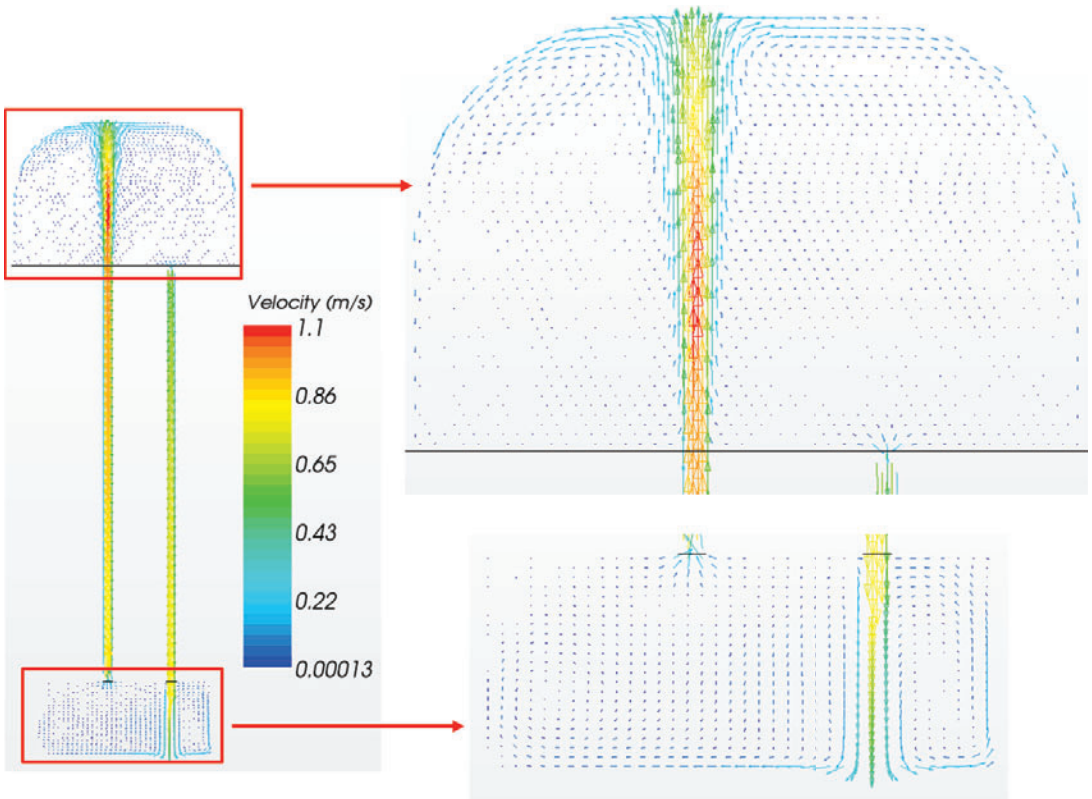

Figure 22. Velocity field obtained using the Realizable Two-Layer k-ε model (P = 4.2 MPa)

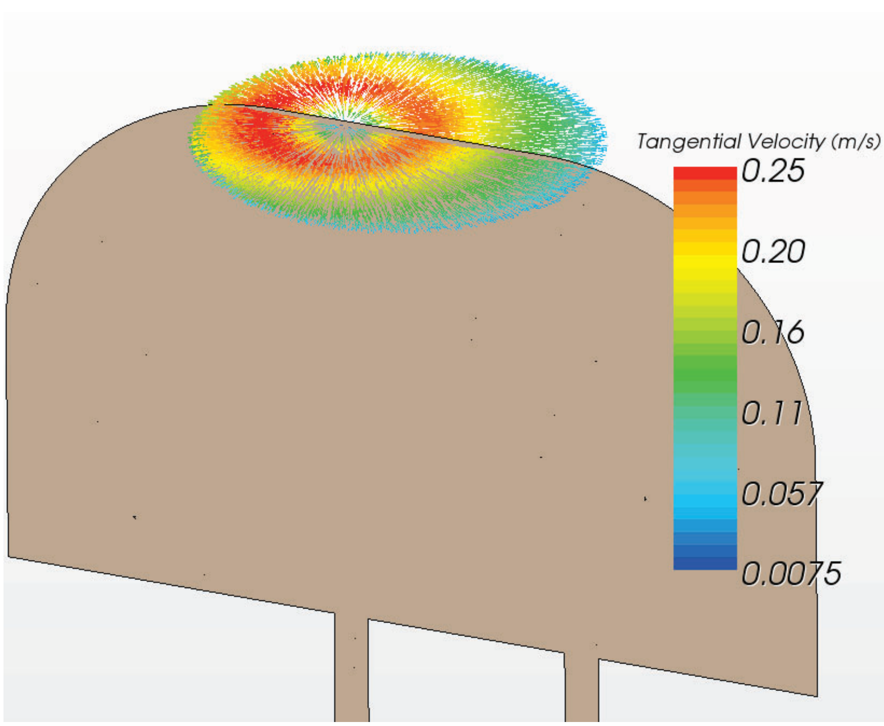

Figure 23. Lateral velocity near the top surface obtained using the Realizable Two-Layer k-ε model

Natural circulation in the dual channel model for the higher pressure case (P = 4.2 MPa) was also simulated using COMSOL. The purpose of the COMSOL model was to run an independent "confirmatory" analysis on the primary CFD results obtained using the Star-CCM+ code. Therefore, a standard k-epsilon RANS model was chosen as the turbulence model in the COMSOL simulation, with sufficient mesh density to keep the run times reasonable. The COMSOL simulations were carried out in the transient mode (till steady-state is reached). Results of the transient simulations can be used to estimate the time scale for the natural circulation flow to reach steady-state. Simulations are carried out using the same boundary conditions as those used in the Star-CCM+ model. Evolutions of the vertical component of the velocity at the mid-height of the hot and cold channels are shown in Figure 24.

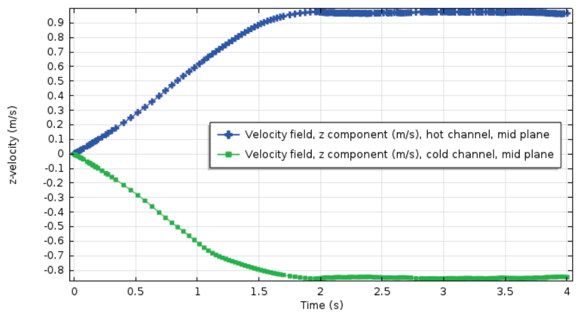

Figure 24. Evolution of the vertical component of the velocity at the mid-height of the cold and hot channels (P = 4.2 MPa).

Steady-state velocities predicted by COMSOL (1.0 m/s and 0.87 m/s for hot and cold channels respectively) are a little higher than those obtained using STAR-CCM+ (0.81 m/s and 0.64 m/s for hot and cold channels respectively). It takes the flow about two seconds to reach steady-state.

4. DISCUSSION AND REMARKS

Missouri S&T, University of Illinois at Urbana-Champaign and Oak Ridge National Laboratory are partners in a project aimed at studying the plenum to plenum (P2P) natural circulation phenomena in VHTR. Missouri S&T will apply a set of novel and sophisticated measurement techniques to measure the flow and temperature fields in a scaled model. These techniques include: the integrated fast response flash mounted heat transfer probes that enable us to directly measure heat transfer coefficient [10]; and hot wire anemometry technique to determine the local velocity in various location of interest in the core. In addition, the gaseous tracer (GT) technique is also available to accurately measure the residence time distribution (RTD) in complex flow structures by injecting pulses of gas tracers and then monitoring its concentration at various locations in the geometry.

CFD code STAR-CCM+ and general purpose code COMSOL are used to simulate the flow and thermal conditions in the dual channel scaled model of VHTR design. These preliminary simulations are being carried out using existing models in the codes to help in the design and instrumentation of the scaled facility, which is being constructed to understand flow conditions under accidental scenarios as well as to gather data for code validation. Results of these preliminary analyses of the Missouri S&T facility are being used to identify the most suitable locations to place the sensors of the novel sophisticated measurement techniques.

5. CONCLUSIONS

Simulations have been carried out at two different system pressure values of 0.8 MPa and 4.2 MPa using STAR-CCM+ and general purpose COMSOL code. Reynolds number for the 0.8 MPa case is near the transition value, and therefore the density distributions through the two channels and in the two plena lead to the same trend when using the laminar or (Realizable Two-Layer k-ε) turbulence models. However, heat transfer rates predicted on the top surface and in both channels when using the Realizable Two-Layer k-ε turbulence model are up to 4% lower than the values predicted when using the laminar flow model (Figures 6 and 11).

Under the 4.2 MPa condition, average velocities in the two channels are higher and the density is 5 ~ 7 times higher than in the 0.8 MPa case. Total heat transfer rate is consequently significantly (about nine times) higher than that in the 0.8 MPa case.

Velocities and temperatures are within expected ranges. Locations for high velocities and temperatures as well as locations where the gradients are high have been identified. Velocity fields and temperature distributions are being used to guide the instrumentation placement in the experimental setup. Selection of appropriate locations for the sensors, informed by a prior simulation of the expected conditions, will help in judicious use of sensors and thus acquiring reliable benchmark data for code validation.

ACKNOWLEDGMENTS

The authors acknowledge the financial support provided by the U.S. Department of Energy project number (NEUP 13-4953 (DE-NE0000744)) for the 4th generation nuclear energy, which made this work possible.

REFERENCES

1. Doug Chapin, S.K., and Jim Nestell, The very hight temperature reactor: A technical summary. 2004, MPr Associates, Inc.
2. Smith, R.C., Uncertainty quantification: theory, implementatation, and applications. 2014, Philadelphia: SIAM.
3. Sato, H., R. Johnson, and R. Schultz, Computational fluid dynamic analysis of core bypass flow phenomena in a prismatic VHTR. Annals of Nuclear Energy, 2010. 37(9): p. 1172-1185.
4. Yoon, S.-J., et al., Study on the flow distribution in prismatic VHTR core with a multi-block experiment and CFD analysis. Nuclear Engineering and Design, 2011. 241(12): p. 5174-5182.
5. Kim, M.-H., N.-i. Tak, and H.-S. Lim, Thermal-fluid assessment of the design options for reactor vessel cooling in a prismatic core VHTR. Annals of Nuclear Energy, 2010. 37(12): p. 1774-1782.
6. Groehn, H.G., Estimate of Cross Flow in High Temperature Gas-Cooled Reactor Fuel Blocks. Nuclear Technology, 1982. 56(2): p. 392-400.
7. Jeeng-Hun Lee, S.-J.Y., Eung Soo Kim, and Goon-Cherl Park, CFD analysis and assessment for cross-flow phenomena in VHTR prismatic reactor. Heat Transfer Journal, 2014. 35(11-12): p. 1152-1160.
8. B. G. Woods, R. B. Jackson., B. L. Nelson, S. R. Cadell, J. N. Reyes. (2015). Scaling Analysis for the Very High Temperature Reactor Test Facility at Oregon State University OSU-HTTF-000000-TECH-001-R0: Oregon State University.
9. Richard R. Schultz, P. D. B., Richard W. Johnson, Glenn E. McCreery, William Taitano, and James R. Wolf. (2010). Studies Related to the Oregon State University High Temperature Test Facility: Scaling, the Validation Matrix, and Similarities to the Modular High Temperature Gas-Cooled Reactor: Idaho National Lab
10. Abdulmohsin, R., Gas dynaimcs and heat transfer in a packed pebble-bed reactor for the 4th generation nuclear energy, in Chemical & Biochemical Engineering. 2013, Missouri University S&T: Rolla, MO.
11. Petersen, H., The Properties of Helium: Density, Specific Heats,Viscosity, and Thermal Conductivity at Pressures from 1 to 100 bar and from Room Temperature to about 1800 K. (1970). Ris¨o Report 224.Danish Atmoic Energy Commission, Research Establishment Ris¨o, Copenhagen.