SAUDI INTERNATIONAL CONFERENCE
ON NUCLEAR POWER ENGINEERING

Proceedings of SCOPE
13-15 Nov. 2023 – KFUPM
Paper 23111

Numerical Prediction of Flow and Heat Transfer in a Molten Corium Pool

Balabaid, Abdallah[1], Al-Gazlan, Osamah[1], Shams, Afaque[2,3], and Siddiqui, Osman[2]

[1]Chemical Engineering Department, King Fahd University of Petroleum & Minerals (KFUPM), Dhahran 31261, Saudi Arabia
[2]Mechanical Engineering Department, KFUPM, Dhahran 31261, Saudi Arabia
[3]Interdisciplinary Research Center for Renewable Energy and Power Systems (IRC-REPS), KFUPM, Dhahran 31261, Saudi Arabia

*Corresponding author: s201768910@kfupm.edu.sa

Abstract – *The integrity of the reactor pressure vessel is of the utmost importance in nuclear reactor safety. One of the cases that need to be studied more thoroughly is the formation of a corium pool which could happen during severe accident scenarios. The melted core is a mixture of multiple melted materials, giving the fluid a unique flow structure. Additionally, the presence of high-temperature variations within the corium pool causes an extremely high Rayleigh number for a natural convection flow regime. This unique problem requires significant research efforts. This paper studies a quarter-circular vessel with internal heat generation representing a high-temperature melted core and corresponds to the famous BALI experiments. In this context, a wide range of RANS (Reynolds-Averaged Navier-Stokes) based CFD (Computational Fluid Dynamics) simulations are performed to better understand the complex thermal-hydraulics phenomena in a corium pool. Additionally, a comparative study is performed to assess the prediction capabilities of different RANS models.*

Keywords: Corium Pool, Bali II, RANS,

I. INTRODUCTION

The fuel inside the nuclear reactor requires continuous cooling to absorb the heat generated. This requires the primary coolant as well as several secondary systems to ensure that there is enough heat extraction from the reactor vessel to avoid any excessive temperatures. A severe accident condition exists when there is a loss of coolant, loss of structural integrity, and a high risk of thermal runaway with high pressure. A corium pool is formed in the case of a severe accident, where the temperature of the core rises significantly high, causing the internals of the core, including the fuel, to melt and build up in the lower plenum of the reactor pressure vessel. This system now behaves as an internally heated natural convection case with an exceptionally high Rayleigh number (Ra) in the range of 10^{13}-10^{17}.

II. Literature Review

The numerical modeling of the heat transfer for a natural convection flow in the corium pools is the focus of the current study. In 1993, the BALI homogeneous pool experiment [1] was conducted with the aim of creating an experimental database of the heat transfer within the corium pools. The homogeneous pool had been designed for the BALI experiments to replicate the natural convection flow and heat transfer conditions in the corium pool at the lower head of the reactor pressure vessel [1].

The numerical modeling of turbulent flow under natural convection requires careful selection of the turbulence model used. The difficulties in the selection of the turbulence model were evident [2] for the common models utilizing the eddy diffusivity approach, especially for the modeling of mixed and natural convection.

Various scenarios in the nuclear industry involve turbulent natural convection heat transfer. One such scenario involves a severe accident in a light water reactor, where the molten core may relocate and accumulate in the lower reactor pressure vessel plenum. The decay heat in the molten core can lead to the formation of a core melt pool which is termed a corium pool and indicates the subsequent onset of natural convection. This turbulent natural convection within the corium pool can significantly impact the thermal loading in the vessel, which is crucial for vessel integrity [3], [4].

One critical aspect of corium retention is a thorough understanding of heat transfer in corium pools, as it defines the safety margin for vessel integrity. In pursuit of this knowledge, the BALI experiment was conducted in 1993 to create a database of heat transfer distribution at the boundaries of corium pools [1]. The BALI experiments explored various pool configurations, including homogeneous, stratified, and porous pools.

A. Bali Experiments

Fig. 1 shows a slice of a typical pressurized water reactor (PWR) lower head. The slice is filled with a water-salt mixture to simulate the corium fluid. Electrodes were used to simulate the internal heat generation while cooling fluid was pumped around the pool for cooling. Thermocouples were installed to measure the temperature at various locations around the pool. Furthermore, particle image velocimetry was used to trace the movement of the fluid, thus creating an accurate reading of the flow structure inside the slice.

The natural convection in such a configuration resulted in three major flow zones within the test section. Fig. 2 shows these three major zones. First, a thermally stratified zone is seen in the lower part of the corium pool with low velocities. While in the top of the pool, an iso-thermal and unstable region exists which is characterized by the presence of large structure eddies caused by cold plumes plunging from the cooler surface on the top. The third and final region is the flow on the cooled curved wall, where maximum velocities occur due to external cooling and lower resulting densities.

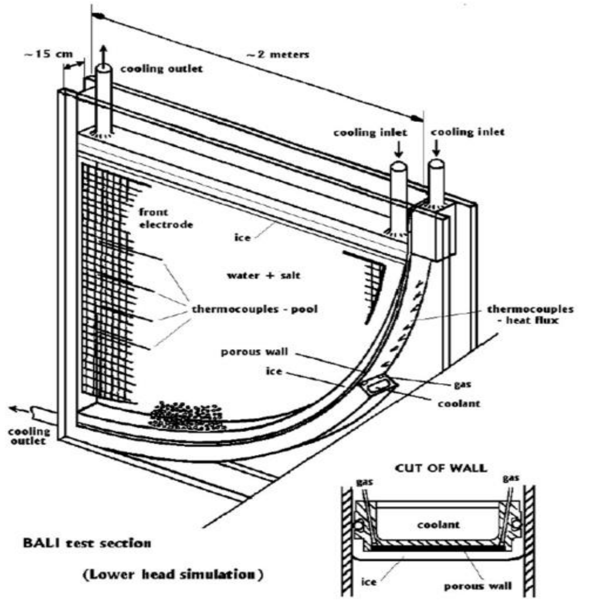

Figure 1: The BALI experiment test diagram, Bonnet, 1999 **[1]**

Several Computational Fluid Dynamics (CFD) studies have attempted to replicate similar experiments and provide insights into molten core phenomena [5]–[7](Le Guennic et al., 2017a,b; Fukasawa et al., 2008). Nevertheless, numerical simulations for this flow configuration are known to be challenging. Various efforts have been made to evaluate different RANS turbulence models for application in molten core and natural convection flow settings.

Dol and Hanjalic [8] studied the 2D and 3D natural convection within a closed domain filled with air. They used low Re k-ε model. Both iso-thermal and iso-flux boundary conditions were studied. They showed that the used model was able to predict the natural convection correctly, however the solution was found to be significantly affected by the mesh resolution near the wall. Thin boundary layer was required to be sufficiently resolved for an accurate solution. Similarly Aounallah et al. [9] performed a validation study of the k-ε and the k-ω SST models for turbulent natural convection flow in a differentially heated cavity containing a fluid with Prandtl number of 0.71. It was found that the k- ω SST model was better than the k-epsilon model. However, the velocities were not fully resolved by either of the models. Nourgaliev et al. [10] demonstrated that low Reynolds k-ε model is not suitable for high Rayleigh number flows that exist inside the nuclear reactors. They suggested modifying these turbulence models to account for anisotropic buoyancy effects that could enhance the prediction accuracy of these models. Fukasawa et al. [7] also assessed several turbulence models for one of the homogeneous BALI test cases, reaching conclusions consistent with Nourgaliev et al. [10].

In summary, the results obtained by the computational fluid dynamics modeling of the natural convection has some deficiencies in modeling the corium pool experimental studies. Having said that, the CFD results are reported to be better than the 1-D models (e.g. MELCOR) used for simple estimation of pool behavior.

The focus of current work is to compare the experimental BALI-II reference and a specific developed model to the models built-in commercial numerical simulation software. The results are expected to show the comparison of obtained results with the published results. Section III will discuss the numerical methodology and flow parameters. Section IV will discuss the results of the study while the conclusion is presented in section V.

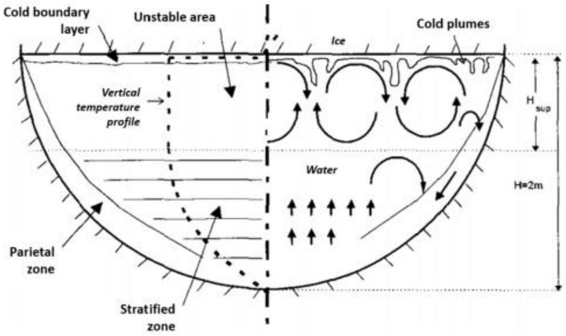

Figure 2: The three zones in the BALI experiment.

III. NUMERICAL METHODOLOGY

A. Computational Domain

A 3D quarter-circular geometry is created based on the BALI-II test in which it has a width of 2 m and height of 2 m. and a thickness of 0.75 m. In addition, the lower 0.1 m is truncated as it represents the ice build-up section as discussed by Shams [11]. The mesh was created with a boundary layer to provide sufficient resolution and capture the viscous sublayer. The y+ was kept at 0.46, this led to the first layer height of 4.87×10^{-4} m. The growth rate was set at 1.2 for 12 total layers. For the bulk volume, a triangular mesh with a size of 0.01 m was used. The resulting mesh is shown in Fig. 3

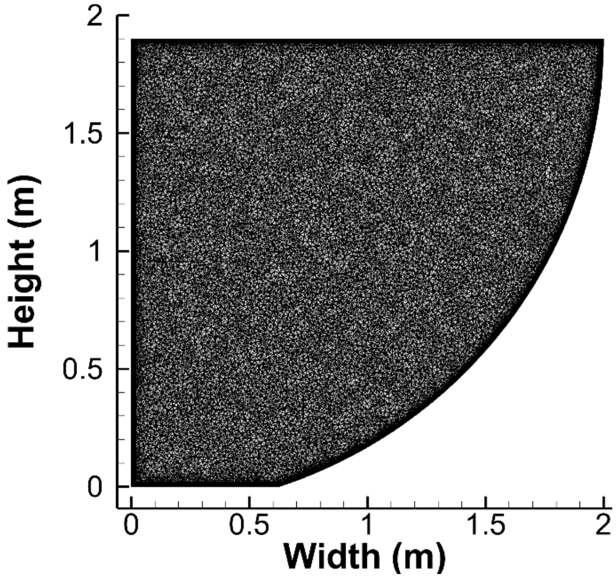

Figure 3: The 2D sectional view of the 3D mesh used.

B. Boundary Conditions

The boundary conditions used in the study are discussed in this section. First, an internal heat generation of 32,755 W/m^3 is used to simulate the heat released from the melted core and components. This correlates to the total power of 15.7 W used in the original BALI experiments. The thermal boundary conditions are listed in Table 1, where an isothermal temperature is imposed on the right, bottom, and top of the quarter-circular structure. Additionally, adiabatic boundary conditions are used for the front, back, and left walls. Finally, a no-slip condition is used for all walls.

Table 1: The boundary conditions

Condition	Faces
Adiabatic: 0 W/m^2	Front, Back, Left
Isothermal: 273.15 K	Right, Bottom, Top

C. Flow Parameters

The operating parameters concerning the flow are based on the following two parameters namely, the Rayleigh number and the Prandtl number.

$$Ra_i = \frac{g\beta Q_v H^5}{k\nu\alpha} \quad \text{(Eq. 1)}$$

$$Pr = \frac{\nu}{\alpha} \quad \text{(Eq. 2)}$$

where g is the gravity constant, β is the thermal expansion coefficient, Q_v is the heat generation, H is the characteristics length, k is the thermal conductivity, ν is the kinematic viscosity, and α is the thermal diffusivity. All fluid properties are assumed constant.

Table 2: Fluid Properties.

Property (unit)	Value
ρ (Kg/m^3)	998.3
μ (Pa.s)	0.001
ν (m^2/s)	1. ×10^{-06}
α (1/K)	1.44 ×10^{-07}
K (W/m*K)	0.6
β (K)	2.07×10^{-04}

The natural convection flow is driven by the buoyancy force due to density generation and shear forces at the walls. Boussinesq approximation is used to model the density change proportional to the temperature through the thermal expansion coefficient β.

D. Numerical Model

This section outlines the computational setup of the simulation and the turbulence modeling methodology. The corium pool study presented in this paper was numerically modeled using the Ansys Fluent [12] computational fluid dynamics software. To model the flow within the corium pool, various RANS modeling approaches are used to compare with the experimental data.

The 4 selected models include both linear and non-linear models for a thorough study. The models include k-ε standard, k-ω SST, k-ω GEKO, and Reynolds stress model (RSM). The selection of these 4 models is reasoned with the following.

k-ε standard is a linear model that depends on two transport variables, namely, the turbulent kinetic energy (k) and the turbulent dissipation (ε). The model, theoretically, should perform well for wall-bounded flows. However, due to the high Rayleigh number of the flow, the model may struggle to capture the large-scale structures developed in the corium pool.

k-ω SST is a linear model that relates the turbulent kinetic energy (k) with the specific dissipation rate (ω). This model is a combination of k-ω near the wall and k-ε in the bulk which theoretically combines the best of the two models and is expected to perform better for wall-bounded flows.

As for the k-ω GEKO model, it is a linear model but provides results similar to non-linear models. This is because it has multiple coefficients that can be tuned to the model specifications, providing the necessary customization of the model. The GEKO model customizability, especially the coefficients of mixing (CMIX) and near wall (CNW), should in theory produce favorable results in the presence of cold plumes and flow near the wall of the lower head.

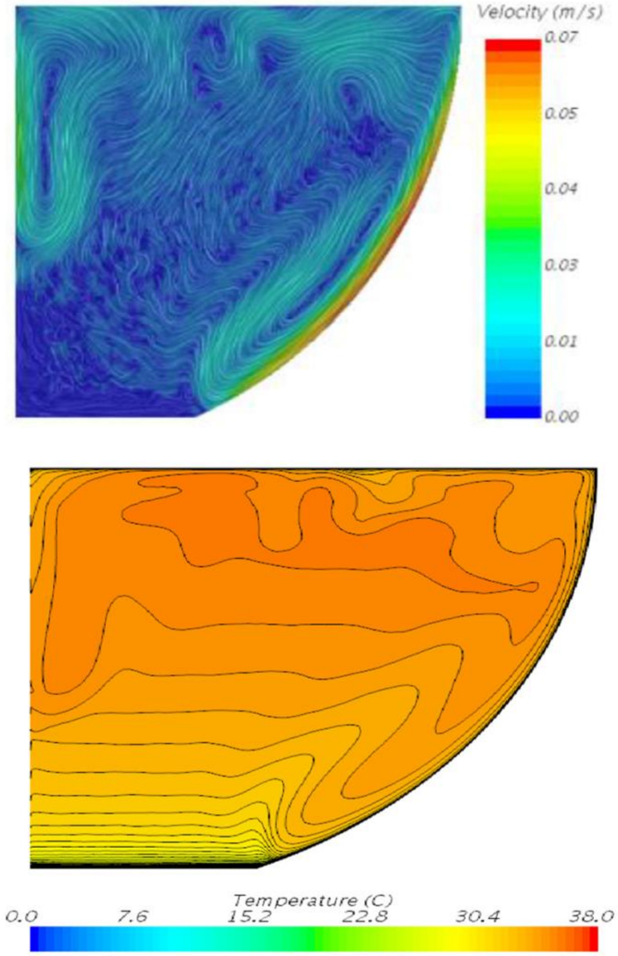

Figure 4. The velocity and temperature contours for the model AHFM-SC [4]

Finally, the RSM is a 7-equation model namely, the mass conservation equation, the momentum conservation equation, the turbulence kinetic energy equation, the turbulence dissipation rate equation, the Reynolds stress transport equation, the turbulence viscosity equation, and the turbulence length scale equation. RSM is a non-linear model meaning that the evolution of k and ε are dependent on the specific flow conditions. The fact that RSM is non-linear should be a great resource to face the intricate nature of a high Rayleigh number flow.

IV. RESULTS AND DISCUSSION

The results presented in this paper are compared with the results presented by Shams [4] and the model AHFM-SC [13]. The velocity and temperature contours are presented in Figure 4. It can be seen that the highest velocity occurs at the right curved cooled wall. A circulation region is also present owing to this high velocity flow. In addition, another circulation zone is visible near the top left wall, where the fluid cooled by the top wall moves downward. The temperature contours show the lowest temperature at the bottom of the pool and higher values near the top.

The velocity contours for all cases are presented in Figure 5. The three regions, discussed earlier, are clearly visible for all the cases with stratified low-velocity regions at the bottom of the pool. The bulk and top of the pool show isothermal flow eddies as circulation regions. The third region is high-velocity flow along the curved surface. The recirculation zones exist due to the constant cooling by the top surface and the cold fluid moving toward the bottom of the pool due to higher density. However, the internal heat generation increases the temperature again and flow recirculates near the top wall. Relatively higher velocities are noted for the k-ω SST and k-ω GEKO. The natural flow is highly unstable especially due to the various recirculation zones.

Fig. 6 shows the corresponding temperature contours for all the four cases. It can be seen that lower temperatures exist for the k-ε model and k-ω GEKO. A few regions of lower temperature can be seen near the top surface where the cold fluid moves downwards. Several such regions exist which shows the unstable nature of such flows.

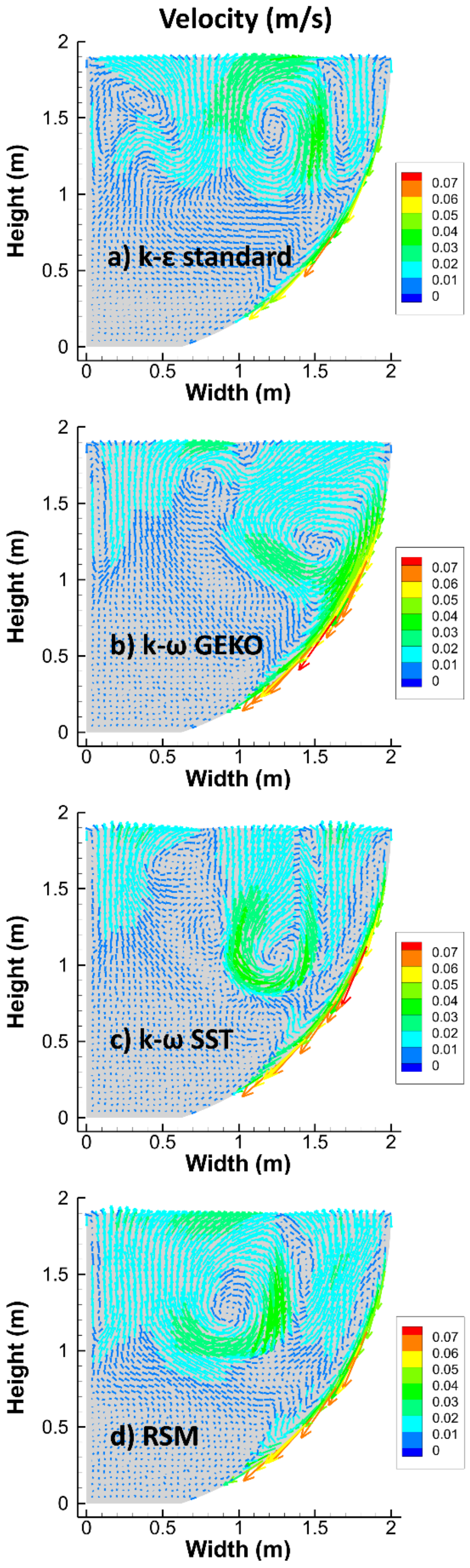

Figure 5 Velocity vectors of all models a) k-ε standard, b) k-ω GEKO, c) k-ω SST, and d) RSM)

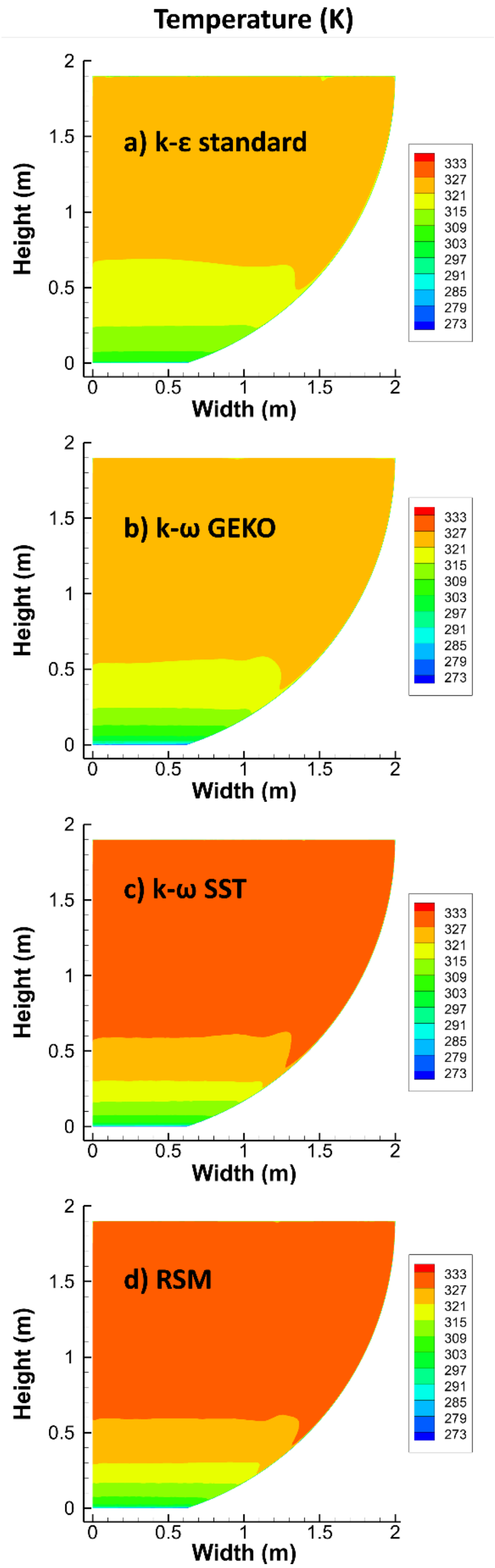

Figure 6. Temperature contours for all models a) k-ε standard, b) k-ω GEKO, c) k-ω SST, and d) RSM)

Figure 7 shows that all the models currently used are overpredicting the temperature at the location 0l2m from the left wall. k-ω SST and RSM show the highest temperatures of over 55 °C, with k-ω GEKO and k-ε showing temperatures over 50 °C while the experimental values are below 40 °C.

Figure 8 shows the comparison of the heat flux at the right curved wall for all the models under consideration. It shows the higher values at the top corner, in comparison to the experimental values and the AHFM-SC, and then lower values for the middle section, while approximately similar values near the bottom section.

This study shows that direct implementation of the models available in the numerical modeling software should taken for granted especially in the case of high Rayleigh number free convection conditions as part of the current work related to the corium pool. The AHDM-SC model has been previously calibrated for such conditions and has shown results closer to the experimental work of BALI experiments.

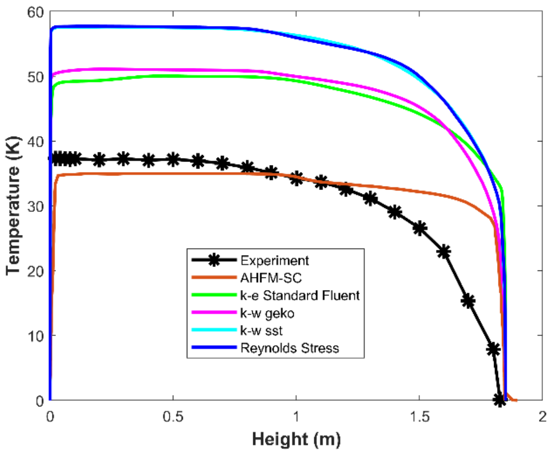

Figure 7. Quantitative comparison of temperature between all models and AHFM-SC and the experimental results.

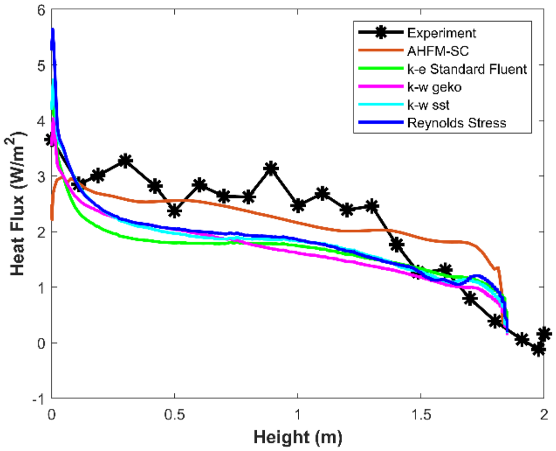

Figure 8. Quantitative comparison of heat flux between all models and AHFM-SC and the experimental results.

V. CONCLUSIONS

In this paper, the melted core of the reactor has been modeled to analyze the flow and heat transfer characteristics under natural convection. The high internal heat generation and external cooling lead to highly buoyant flow conditions and require extensive studies to increase the understanding and predictions to ensure safe containment and cooling under severe accident conditions. This corium pool was simulated and compared to the experimental results of BALI-II and a calibrated AHFM-SC model. Four models were used, namely, k-ε standard, k-ω GEKO, k-ω SST, and RSM. All models show the low velocity, large circulation region, and high-velocity region near the curved walls.

All classic models overestimate the temperature profile throughout the quarter-circular while AHFM-SC was able to match the temperature of the experimental values for a significant portion of the quarter-circular which is attributed to its calibration for high Rayleigh flows.

This shows the importance of ensuring the utility of the CFD methods by validating them with experimental results. The overprediction of temperature can be a great concern in highly critical applications like the corium pool where the safety margin is of utmost importance.

VI. ACKNOWLEDGMENT

The authors acknowledge the support provided by King Fahd University of Petroleum and Minerals (KFUPM).

VII. Nomenclature

g	gravity constant
β	thermal expansion coefficient
Q_v	heat generation
H	characteristics length
k	thermal conductivity
v	kinematic viscosity
α	thermal diffusivity
Ra	Rayleigh number
Pr	Prandtl number

VIII. REFERENCES

[1] J. M. Bonnet, “Thermal hydraulic phenomena in corium pools: the BALI experiment, in Proc. Workshop on Severe Accident Research (SARJ-98), Tokyo, Japan, Nov. 4–6, 1998, pp. 79–86.,” 1999, DOI:

[2] OECD/NEA, Handbook on Lead–bismuth Eutectic Alloy and Lead Properties, Materials Compatibility, Thermal-hydraulics and Technologies. 2007, DOI:, OECD NEA No. 6195, ISBN 978-92-64-99002-9.

[3] X. Gaus-Liu, A. Miassoedov, T. Cron, and T. Wenz, “In-vessel melt pool coolibility test—Description and results of LIVE experiments,” *Nucl. Eng. Des.*, vol. 240, no. 11, pp. 3898–3903, Nov. 2010, DOI:10.1016/J.NUCENGDES.2010.09.001.

[4] T. G. Theofanous, C. Liu, S. Additon, S. Angelini, O. Kymäläinen, and T. Salmassi, “In-vessel coolability and retention of a core melt,” *Nucl. Eng. Des.*, vol. 169, no. 1–3, pp. 1–48, 1997, DOI:

[5] C. Le Guennic, L. Saas, R. Le Tellier, Y. Wu, M. Guingo, and J. Laviéville, “Contribution of CFD studies to IVR assessment,” in *8th European Review Meeting on Severe Accident Research-ERMSAR-2017, Warsaw, Poland*, 2017.

[6] C. Le Guennic, E. Skrzypek, L. Vyskocil, M. Skrzypek, A. Shams, and L. Saas, “Analysis of in-vessel corium pool behaviour using CFD tools,” in *Proceedings of ERMSAR-2017 conference, Warsaw (Poland)*, 2017.

[7] S. H. Masanori FUKASAWA and M. SAITO,

"Thermal-Hydraulic Analysis for Inversely Stratified Molten Corium in Lower Vessel," *J. Nucl. Sci. Technol.*, vol. 45, no. 9, pp. 873–888, 2008, DOI:10.1080/18811248.2008.9711489.

[8] H. S. Dol and K. Hanjali, "Computational study of turbulent natural convection in a side-heated near-cubic enclosure at a high Rayleigh number," *Int. J. Heat Mass Transf.*, vol. 44, no. 12, pp. 2323–2344, Jun. 2001, DOI:10.1016/S0017-9310(00)00271-4.

[9] M. Aounallah, Y. Addad, S. Benhamadouche, O. Imine, L. Adjlout, and D. Laurence, "Numerical investigation of turbulent natural convection in an inclined square cavity with a hot wavy wall," *Int. J. Heat Mass Transf.*, vol. 50, no. 9–10, pp. 1683–1693, May 2007, DOI:10.1016/J.IJHEATMASSTRANSFER.2006.10.015.

[10] R. R. Nourgaliev, T. N. Dinh, and B. R. Sehgal, "Effect of fluid Prandtl number on heat transfer characteristics in internally heated liquid pools with Rayleigh numbers up to 1012," *Nucl. Eng. Des.*, vol. 169, no. 1–3, pp. 165–184, Jun. 1997, DOI:10.1016/S0029-5493(96)01282-4.

[11] A. Shams, "Towards the accurate numerical prediction of thermal hydraulic phenomena in corium pools," *Ann. Nucl. Energy*, vol. 117, pp. 234–246, Jul. 2018, DOI:10.1016/J.ANUCENE.2018.03.031.

[12] "Product Documentation, Fluent 15, ANSYS Inc." ANSYS Inc., Pennsylvania, United States, 2014.

[13] A. Shams, "Understanding the need for proper turbulent heat flux modelling for non-unity Prandtl number fluids," in *Computational Fluid Dynamics for Nuclear Reactor Safety (CFD4NRS-9)*, 2023, pp. 1–13.