Proceedings of SCOPE
13-15 Nov. 2023 – KFUPM
Paper 23008

Numerical Modeling of Mixed Convection Flow Regime in Low-Prandtl Number Fluids

Alsubhi, Abdulaziz[1], Meri, Yazan[1], Al-Athel, Khaled[1,2], Shams, Afaque[1,3] and Siddiqui, Osman[1]*

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

*Corresponding author *email:* abdulaziz.a.alsubhi@gmail.com

Abstract – *Turbulent heat transfer is an extremely complex phenomenon and is critical in scientific and industrial applications. It becomes much more challenging in a buoyancy-influenced flow regime, particularly for non-unity Prandtl number (Pr) fluids. In this article, an effort has been put forward to assess the prediction capabilities of different Reynolds-Averaged Navier-Stokes (RANS) based turbulence models for a mixed convection flow regime. In this regard, a mixed convection flow in channel is considered for three different Richardson numbers(Ri= 0.25, 0.5, and 1). The considered flow configuration is a parallel plate arrangement with differentially heated side walls. Two different turbulent heat flux models, are compared with the available reference Direct Numerical Simulation database. The prediction capabilities for these modeling approaches are assessed and will be extensively discussed in this paper.*

Keywords: Mixed convection, Nuclear energy, Liquid metal, Turbulence modelling.

I. Introduction

Mixed convection is a fluid flow phenomenon where fluid motion is induced by natural and forced convection sources. It is commonly observed in various thermofluidic applications characterized by pressure and temperature variances. The forced component of the mixed convection is produced by external sources such as pressure and frictional effects. Whereas natural convection is driven by an internal property of fluids which is the density variance as a function of temperature. The dominance of each component in the flow is dependent on multiple factors including the fluid properties, temperature gradients, and the external forces effects. With the development of fourth-generation nuclear reactors, the study of mixed convection has gained significant importance. Nuclear reactor development requires extensive design considerations, particularly with the advancements of non-unitary Prandtl number fluids, e.g., Liquid Metal Cooled Reactor (LMR) and Sodium-Cooled Fast Reactor (SFR). Consequently, studying mixed convection flows and their behaviour as a function of changes in fluid properties became a significant aspect of nuclear power development.

The presence of mixed convection in such critical applications has triggered extensive research efforts to understand and study this phenomenon. The novel experimental work of Poiseuille-Rayleigh-Bénard (PRB) flow initiated a reference case for the analysis development and validation. The PRB case is a pressure-driven Poiseuille flow in which buoyancy effects are created by a temperature difference between the bounding walls of the channel. This case was set to be a milestone reference as it presents the key characteristics of forced and natural convection combinations. Researchers have extensively studied the PRB case through multiple experiments since the work of [1], [2]. Subsequently, several experimental and numerical simulation analyses have served as a key influence in understanding and developing the phenomenon applications [3]–[5].

A. Analysis Methodology and Numerical Modeling

Mixed convection can be analyzed through multiple flow parameters such as temperature and velocity variations besides the non-dimensional flow representing parameters. Through this, Richerdson number quantifies the mixed convection flows and represents their components' dominance. Other flow quantities are considered in the study due to their impact on the flow such as Prandtl number (Pr) as it has a crucial impact on the convection properties.

The approaches to study flow and heat transfer are experimental, numerical, and computational methods. Experimental methods have stayed the essence of thermofluidic science throughout history as they are a reliable research methodology and many current theories have been constructed on an experimental basis. However, they have limitations in explaining several scientific advances. The limitations could be constituted in measurement ability and measurement resolution as well as experiment construction and replicability, practicability, and accuracy. Analytical solutions are possible with several simplifications and assumptions to result in simple fundamental studies, however, as the research statements advance with higher complexities, the application of the analytical methods becomes restricted. This brought focus to the computational tools that aim to solve numerical models for complex domains. Computational fluid dynamics (CFD) is a model approach that solves complex problems based on the fundamental numerical models implemented on smaller elements of the domain. Turbulent fluid flow models are mathematical models to simulate and predict the disturbance effects for real-life flows with practical results. One of the CFD approaches is Direct Numerical Simulation (DNS), which solves the actual Navier–Stokes equations even for turbulent flows without requiring a turbulence model. Thus, having a reference solution allows the validation and development of various turbulence models. Reynolds-Averaged Navier-Stokes (RANS) and other CFD models partially solve the flow governing equations with an influencing use of turbulence modelling. RANS models performance is constrained to the prediction of mean flow behaviour as of the adoption of the Reynolds averaging technique. However, RANS models are favored as they provide practical solutions that can be implemented in complex geometries. There is a variety of RANS models that consider multiple turbulence modelling and diffusion hypotheses. This brought discussion, especially with the notable effect of fluid properties variation. Several researchers considered the implementation of robust and well-known RANS turbulence models adopting the Simple Gradient Diffusion Hypothesis (SGDH), utilizing the advantage of adjusting the turbulent Prandtl number. This approach had brought strong simplifying assumptions, however, its modelling improvement limitations are concerned, particularly in non-unitary Prandtl fluids. The assumptions relating the Turbulence Momentum Flux (TMF) to Turbulence Heat Flux (THF) can produce a significant effect in the presence of natural convection. This is due to the attribution of the convection to the forced flow regime. Alternatively, Algebraic Heat Flux Models (AHFM) are developed with separate calculations of TMF and THF. Through that, AHFM are able to calculate the flow buoyancy production effect. It is worth noting that a developed version of AHFM based on Shams correlation has been established and calibrated for low Pr fluids, see [6].

Notable research projects that have contributed to the study of the PRB case and its implications in mixed convection include the simulation and assessment of turbulence models for the PRB horizontal flow channel. [7] has performed a DNS simulation for a three-dimensional PRB case studying variable Prandtl number. In a similar manner, [8] studied the fluid behaviour at multiple Richardson numbers, establishing reference cases at multiple fluid properties. Through the RNAS validation work, [9] performed a thorough analysis considering four RANS turbulence models at the three convection modes (i.e. forced, natural and mixed convection), where it is concluded that a THF closure model is essential to produce accurate results. Similarly, [10] reported the advancement of the AHFM-NRG model simulating low Prandtl fluids in complex geometries. It is found that a promising approach to the mixed convection cases is through the utility of anisotropic algebraic heat flux along with a second moment closure. As for the mixed convection vertical channel flow, [11] initiated a DNS reference dataset considering multiple mixed convection states at Pr = 0.025. [12] has investigated the validation of SGDH and AHFM models for variable convection modes. It is illustrated that SGDH models produced higher errors as the buoyancy effect increased, whereas the AHFM-SC model showed a very good estimation compared to a DNS database. [13]–[17] had implemented high-fidelity simulations studying low Prandtl fluids mixed convection for complex geometries. [18] had performed RANS SGDH for flow inside a concentric annual studying the mixed convection phenomenon at Pr=0.021. These cases focused on the analysis of low-Prandtl flows, which constitute the base design of Gen-IV reactors such as LMR. As of the use of the RANS model is constrained by the validation with specific applications, this research builds reference databases to which RANS turbulence models can be analyzed and validated. This study provides further investigation of RANS models' performance in a vertical channel subjected to a variable mixed convection mode. It studies Low Prandtl fluid considering the advancement of nuclear power plants.

B. Implications of the nondimensional numbers

The Prandtl number defines the ratio of momentum diffusivity to thermal diffusivity. It infers the thickness and development of thermal and momentum boundary layers. As a fluid's Prandtl number deviates from unity, the difference in boundary layer thickness gets expanded resulting in one boundary layer being small when compared to the other as illustrated in Figure 1. Examples of low Prandtl numbers include liquid metal flows (i.e. small momentum boundary layer). Capturing the boundary layer is one requirement to be addressed in computational modelling. This process is highly dependent on the discretization elements' size as smaller elements can model more accurate flow features [19]. The dimensionless wall distance y^+ is used to denote the wall regions namely: the viscous sublayer ($y^+ < 5$), the buffer layer ($5 < y^+ < 30$), and the logarithmic layer ($y^+ > 30$). The y^+ for this flow is kept below 0.5 to capture the flow within the viscous sublayer near the wall and is calculated as:

$$y^+ = \frac{\rho y u_\tau}{\mu} \tag{1}$$

where ρ is density, μ is dynamic viscosity, y is wall distance and u_τ is the friction velocity.

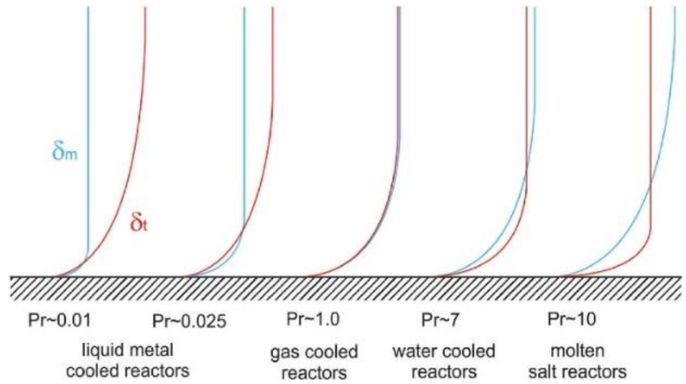

Figure 1. Momentum and thermal boundary layer development for multiple Prandtl numbers [20].

Richardson number quantifies the significant relation of natural flow to the forced flow. It defines the dominance ratio of the natural convection providing an assessment of the flow stability and stratification. Richardson number is defined as the ratio of the potential energy associated with buoyancy forces to the kinetic energy associated with shear forces. As the number increases, it indicates the increasing dominance of natural convection. Ri number has a significant effect in turbulence modelling, particularly with the models that are based on the Reynolds analogy. These models are embedded with assumptions that aid in decreasing computational power. One of these assumptions that are directly related to the Ri number is the attribution of THF to TMF. As an effect of this assumption, RANS models ignore the thermal effect of the natural convection, leading to inaccuracy in calculating mixed convection flows.

II. CASE DESCRIPTION

This study analyzes mixed convection cases with a focus on evaluating RANS models' performance on variable mixed convection flow combinations. The case consists of a vertical channel with differentially heated side walls where the buoyancy acts in the streamwise direction. The analyzed domain for the cases consists of a 2D rectangular planner channel as illustrated in Figure 2. The geometry is specified with the wall-normal dimension ($L_Y = 2\delta$) and streamwise dimension ($L_X = 4\pi\delta$) with δ representing the boundary layer thickness. The flow is imposed with periodic boundary conditions to ensure a fully developed boundary layer. The no-slip condition is applied to the channel sides. A constant wall temperature boundary condition is applied at the side walls where the right wall represents the cold plate with a temperature (T_C) and the left wall represents the hot plate with a temperature (T_H). The difference in temperatures ($\Delta T = T_C - T_H$) induces a buoyancy effect across the x-axis direction where the hot side aids the flow and the cold side opposes the flow.

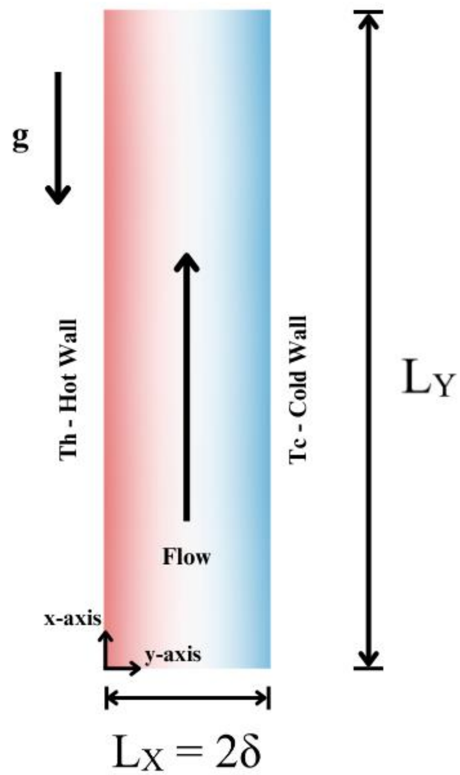

Figure 2. An illustration of the computational domain for the studied case.

Table 1 Side walls operating temperatures.

Ri value	Tc (K)	T_H (K)	ΔT (K)
0.25	548.85	545.14	3.71
0.50	550.71	543.28	7.43
1.00	554.43	539.56	14.87

A. Flow parameters

A forced flow is applied at the x-axis direction with a constant flow rate. The non-dimensional flow parameters are specified as follows: Re_b = 4,667 and Pr = 0.025, with a resulting flow bulk velocity of U_b = 0.025 m/s. The buoyancy impact is varied by controlling Richardson number from the nearly fully forced convection flow regime to multiple degrees of buoyancy strengths as Ri = 0.25, 0.5, and 1. The variation in Richardson number is obtained as a direct function of the side walls' temperature differential. The specified wall temperatures are listed in Table 1 where the increment in the temperature difference induces a higher buoyancy effect leading to a higher Richardson number. The resultant effect generates a variation in the streamlines' velocities due to aiding and opposing buoyancy effects.

Table 2. Summary of the simulation fluid parameters

Symbol	Definition	Value
Re_b	Bulk Reynolds number	4,667
Pr	Prandtl Number	0.025
Ri	Richardson Number	0.25, 0.5, 1
U_b	Bulk velocity (m/s)	0.025
μ	Dynamic viscosity (Pa·s)	2e-3
K	Thermal conductivity (W/m·K)	11.44
ρ	Density (kg/m^3)	10,358
β	Thermal expansion coefficient (K^{-1})	1e-4

A. Numerical Solution

All numerical simulations have been conducted utilizing the commercially available software Ansys Fluent 2022 [21]. Ansys Fluent is a CFD software that is known for its advanced physics modelling capabilities and accuracy. The selected turbulent flow models were based on the Reynolds analogy, which relates heat transfer to turbulent momentum. The particular turbulent models studied are:

- k-ω SST (Shear Stress Transport)
- Generalized k-ω (GEKO)
- Reynolds Stress Models (RSM)

k- ω is among the most common models that use two additional transport equations for turbulent kinetic energy and turbulent dissipation. It has several extensions in the field including k-ω SST which is a two-elemental model that combines the best of k-ω and k-ε formulations. It uses the k-ω formulation close to the wall (low Reynolds within the viscous sublayer) while using the k-ε formulation away from the wall in the free stream where k-ω has a high sensitivity to the turbulence properties. The GEKO model is another extension of k-ω. It is developed to have higher tuning flexibility. The RSM is a second-order closure model, and its formulation follows the precise Reynolds stress transfer equation.

C. Mesh Generation

The mesh has been produced for the computational domain utilizing the meshing software provided by ANSYS, Inc. As demonstrated in Figure 3, the mesh is constructed with inflation layers near the walls in the wall-normal direction. The main purpose of inflation is to capture momentum and thermal boundary layers across the side walls. The mesh is generated considering y^+ less than 0.7. The inflation between layers extends with a stretching ratio of 1.2. Along flow direction, the mesh is of 0.001 m element size resulting in a total elements of 20,900.

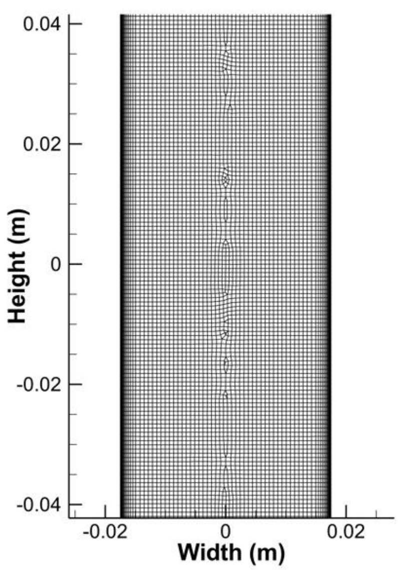

Figure 3. A cross-sectional capture of the case upper half mesh.

III. RESULTS AND DISCUSSION

In this section, the results obtained with the use of k-ω SST, GEKO, and RSM turbulent models will be presented with a thorough assessment of the simulation results for the mixed convection cases. The cases will be evaluated through a qualitative review and a quantitative data validation with the DNS [11] and AHFM-SC [12] taken as reference data. The qualitative results are set to provide insight into CFD models' capability in modelling turbulence eddies. Quantitative comparisons are an essential part of assessing the accuracy of CFD models in predicting the flow behaviour of a system. Quantitative comparisons involve the comparison of numerical simulation data with experimental data or reference simulations such as DNS simulations. Additionally, advanced and calibrated CFD turbulence models can be used to validate the results. AHFM-SC is the most advanced and up-to-date RANS model available in the nuclear industry and is able to predict mixed convection behaviour better than the other RANS-based models. The evaluation will study THF considering the effect of the Richardson number at a low Prandtl value.

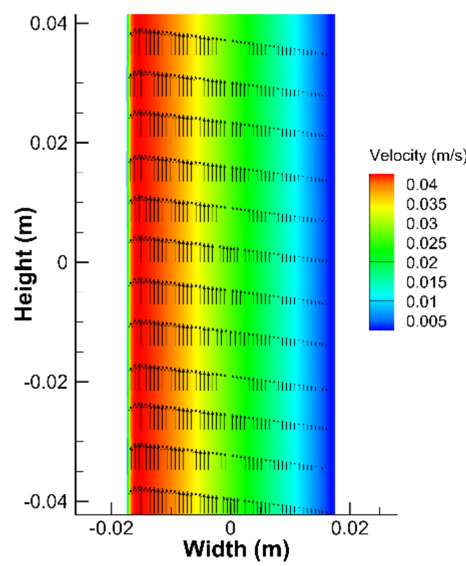

Figure 4. Velocity field capture of RANS simulation.

A. Qualitative Results Assessment

The obtained qualitative results for the Ri=0.5 case with both TMF represented in the velocity distribution and THF represented in the temperature distribution are shown in Figures 4 and 5. The RANS model k-ω SST was able to predict the average behaviour of the flow. A typical turbulent flow where the bulk flow has maximum speed is shown with the no-slip condition effect in the near-wall region. Because of the limitations of the RANS models, they cannot capture turbulence eddies, which is due to the averaged effect of RANS. In a similar manner, the heat diffused through the right and left walls can be visualized in Figure 5. The temperature shows a similar effect to the velocity gradient. This effect is mainly due to the shortcoming of the Reynolds analogy-based models where the calculations relate heat transfer to the momentum flux.

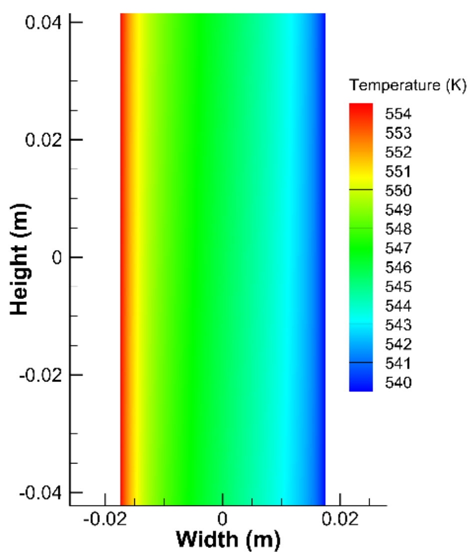

Figure 5 Temperature field capture of RANS simulation.

B. Quantitative Results Assessment

Quantitative comparisons are performed to the temperature field at variable Richardson numbers. The temperature field studies the heat dissipation to the flow besides the mixed convection effects in the channel. It represents the THF situation indicating the models' effectiveness in calculating heat transfer. Results are compared with the reference DNS database [11] as shown in Figure 6. The selected parameter for comparison is the normalized horizontal temperature profile calculated as follows:

$$T^* = \frac{(T - T_{cold})}{T_{hot} - T_{cold}} \tag{2}$$

Where T represents the local mean temperature. Results are extracted at the mid-vertical cross-section of the flow domain. Normalization is done for the horizontal width as well.

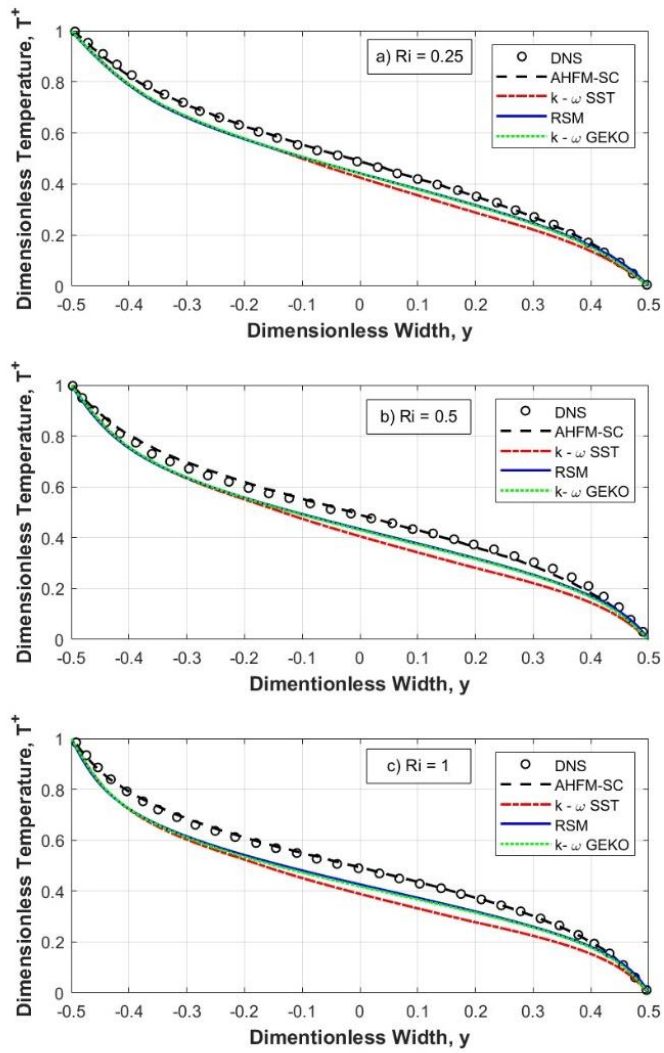

Figure 6 The normalized temperature variation along the horizontal cross-section - a) Ri = 0.25, b) Ri = 0.5, and c) Ri = 1.

Figure 6 shows the temperature profile along the horizontal flow axis for different models. As the DNS model solves the full Navier–Stokes equation without using any turbulence modelling, it is taken as the reference to compare other models. The AHFM-SC model has shown a matching result compared to the DNS simulation as well as a symmetrical gradient on the streamwise axis. It is also evident from Figure 6 that this model is able to predict the temperature within the thermal boundary layer as well as the bulk of the fluid in all mixed convection cases. The AHFM-SC model has been calibrated for a wide range of test cases, as highlighted in [22]. It is characterized by its compatibility with mixed convection cases through the explicit modelling of turbulent heat flux. Alternatively, the analyzed RANS models k-ω SST, k-ω GEKO, and RSM underestimated the thermal heat flux represented in the temperature distribution. GEKO and RSM show a matching temperature gradient, while greater error is detected from k-ω SST. The error in the three models increases at higher Richardson number values. This is attributed to the limitations of RANS models predicting the natural convection phenomenon. In addition, the presence of the aiding and opposite buoyancy effect increases turbulence in the flow with eddies generated on the side walls. It results in an intrinsically unsteady flow causing numerical models to struggle to generate converged results in non-calibrated models.

IV. CONCLUSIONS

This work focuses on the mixed convection flow regime that exists in almost all real-world applications and the buoyancy-driven, i.e., natural and mixed convection elements, making it challenging especially for turbulent flows. CFD simulations have been performed to address the ability of RANS-based turbulence models to predict the mixed convection regime at non-unity Prandtl numbers. A 2D forced flow was modeled between differentially heated walls inducing the buoyancy effect. The mixed convection was studied at multiple Richardson numbers and validated against the reference DNS database. The results obtained from three different turbulence models are also compared with an advanced turbulent heat flux model, called AHFM-SC, to assess different turbulence models and their respective limitations. It has been found that the AHFM-SC shows superior results compared to the other RANS models (k-ω SST, GEKO and RSM). This is mainly because the AHFM-SC takes into account the explicit modelling of turbulent heat flux and is well-calibrated for low-Prandtl number fluids. Whereas the other considered models calculate the THF based on the simplest assumption of the Reynolds analogy.

REFERENCES

[1] J. K. Platten and M. Lefebvre, “A preliminary experimental investigation of the stability of flows with an imposed temperature gradient,” *Physica*, vol. 51, no. 2, pp. 330–332, Jan. 1971.

[2] J. M. Luijkx, J. K. Platten, and J. C. Legros, “On the existence of thermoconvective rolls, transverse to a superimposed mean poiseuille flow,” *Int J Heat Mass Transf*, vol. 24, no. 7, pp. 1287–1291, Jul. 1981.

[3] C. Bonnefoi, C. Abid, M. Medale, and F. Papini, “Poiseuille–Benard instability in a horizontal rectangular duct water flow,” *International Journal of Thermal Sciences*, vol. 43, no. 8, pp. 791–796, Aug. 2004.

[4] R. Taher and C. Abid, “Experimental determination of heat transfer in a Poiseuille-Rayleigh-Bénard flow,” *Heat and Mass Transfer/Waerme- und Stoffuebertragung*, vol. 54, no. 5, pp. 1453–1466, May 2018.

[5] R. Taher, M. M. Ahmed, Z. Haddad, and C. Abid, “Poiseuille-Rayleigh-Bénard mixed convection flow in a channel: Heat transfer and fluid flow patterns,” *Int J Heat Mass Transf*, vol. 180, Dec. 2021.

[6] A. Shams, F. Roelofs, E. Baglietto, S. Lardeau, and S. Kenjeres, “Assessment and calibration of an algebraic turbulent heat flux model for low-Prandtl fluids,” *Int J Heat Mass Transf*, vol. 79, pp. 589–601.

[7] D. De Santis, A. De Santis, A. Shams, and T. Kwiatkowski, “The influence of low Prandtl numbers on the turbulent mixed convection in an horizontal channel flow: DNS and assessment of RANS turbulence models,” *Int J Heat Mass Transf*, vol. 127, pp. 345–358, Dec. 2018.

[8] W. Guo, A. Shams, Y. Sato, and B. Niceno, “Influence of buoyancy in a mixed convection liquid metal flow for a horizontal channel configuration,” *Int J Heat Fluid Flow*, vol. 85, Oct. 2020.

[9] A. Shams, A. De Santis, L. K. Koloszar, A. Villa Ortiz, and C. Narayanan, “Status and perspectives of turbulent heat transfer modelling in low-Prandtl number fluids,” *Nuclear Engineering and Design*, vol. 353, Nov. 2019.

[10] A. Shams, A. De Santis, and F. Roelofs, “An overview of the AHFM-NRG formulations for the accurate prediction of turbulent flow and heat transfer in low-Prandtl number flows,” *Nuclear Engineering and Design*, vol. 355, Dec. 2019.

[11] W. Guo and H. M. Prasser, “Direct numerical simulation of turbulent heat transfer in liquid metals in buoyancy-affected vertical channel,” *Int J Heat Mass Transf*, vol. 194, Sep. 2022.

[12] A. Pucciarelli, A. Shams, and N. Forgione, “Numerical prediction of turbulent flow and heat transfer in buoyancy-affected liquid metal flows,” *Ann Nucl Energy*, vol. 186, Jun. 2023.

[13] L. Marocco, “Hybrid LES/DNS of turbulent forced and aided mixed convection to a liquid metal flowing in a vertical concentric annulus,” *Int J Heat Mass Transf*, vol. 121, pp. 488–502, Jun. 2018.

[14] A. Shams, B. Mikuž, and F. Roelofs, “Numerical prediction of flow and heat transfer in a loosely spaced bare rod bundle,” *Int J Heat Fluid Flow*, vol. 73, pp. 42–62, Oct. 2018.

[15] A. Pucciarelli, “Results of a LES Application to LBE Turbulent Flow in a Wire-Wrapped Single Rod Channel,” in *28th International Conference on Nuclear Engineering*, American Society of Mechanical Engineers, Aug. 2021.

[16] L. Marocco, M. Sala, G. Centurelli, S. Straub, and L. Colombo, “LES simulations and Nusselt number decomposition of turbulent mixed convection of liquid metals flowing in a vertical pipe,” *Int J Heat Mass Transf*, vol. 182, Jan. 2022.

[17] P. Zhao, J. Zhu, Z. Ge, J. Liu, and Y. Li, “Direct numerical simulation of turbulent mixed convection of LBE in heated upward pipe flows,” *Int J Heat Mass Transf*, vol. 126, pp. 1275–1288, Nov. 2018.

[18] L. Marocco, A. Alberti di Valmontana, and T. Wetzel, “Numerical investigation of turbulent aided mixed convection of liquid metal flow through a concentric annulus,” *Int J Heat Mass Transf*, vol. 105, pp. 479–494, Feb. 2017.

.

[19] K. H. Hanjali´c, “One-Point Closure Models for Buoyancy-Driven Turbulent Flows,” *Annual Review of Fluid Mechanics,* vol. 34, pp. 321–347, 2002.

[20] A. Shams *et al.*, “A collaborative effort towards the accurate prediction of turbulent flow and heat transfer in low-Prandtl number fluids,” *Nuclear Engineering and Design*, vol. 366, Sep. 2020.

[21] “Ansys fluent | Fluid Simulation Software,” Nov. 20, 2022

[22] A. Shams, “Understanding the need for proper turbulent heat flux modelling for non-unity Prandtl number fluids,” College Station, Texas, USA., Feb. 2023.