NURETH14-143

INVESTIGATION OF THE INFLUENCE OF TURBULENCE MODELS ON THE PREDICTION OF HEAT TRANSFER TO LOW PRANDTL NUMBER FLUIDS

R. Thiele, W. Ma and H. Anglart
Royal Institute of Technology, Sweden, Stockholm

Abstract

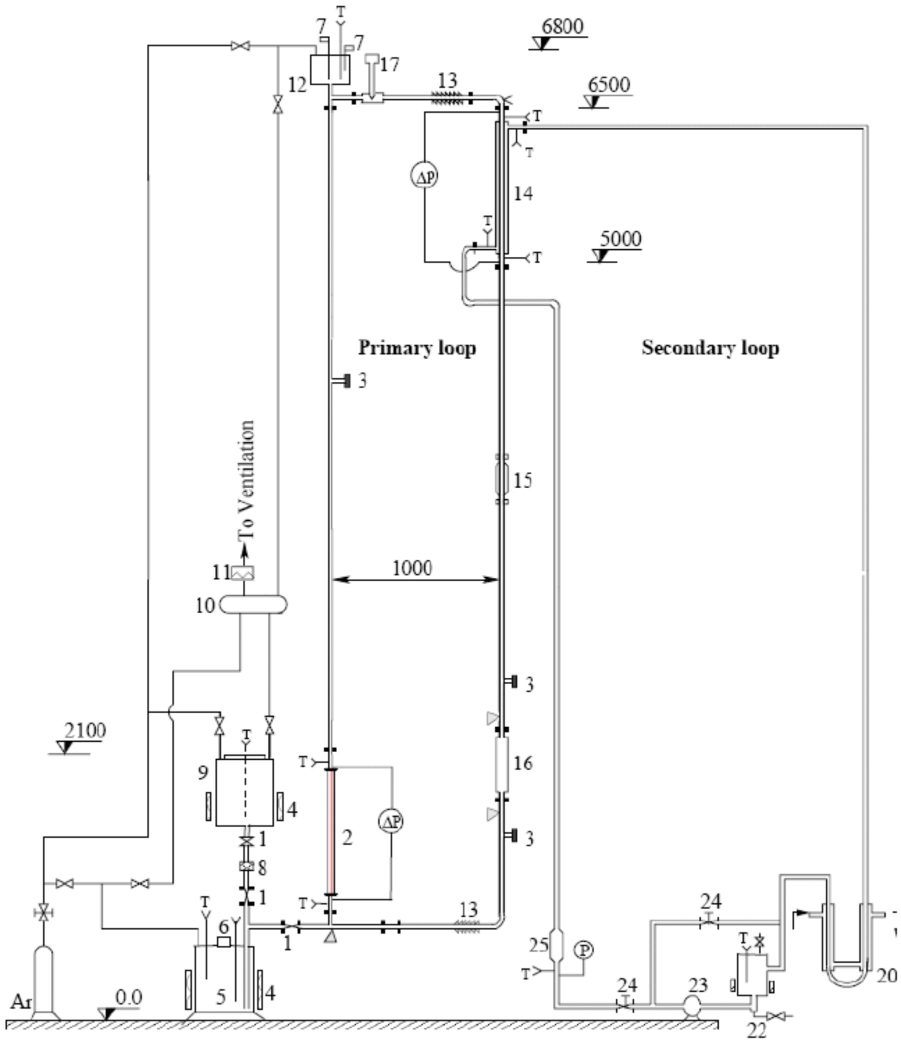
Despite many advances in computational fluid dynamics (CFD), heat transfer modeling and validation of code for liquid metal flows needs to be improved. This contribution aims to provide validation of several turbulence models implemented in OpenFOAM. 6 different low Reynolds number and 3 high Reynolds number turbulence models have been validated against experimental data for 3 different Reynolds numbers. The results show that most models are able to predict the temperature profile tendencies and that especially the k-ω-SST by Menter has good predictive capabilities. However, all turbulence models show deteriorating capabilities with decreasing Reynolds numbers.

Introduction

Liquids with a low Prandtl number are encountered in many present and future nuclear reactor designs, such as liquid metal cooled fast breeder reactors and lead-cooled accelerator driven systems. The prediction of the temperature and the velocity field is of utter importance for safety analysis and component design. Despite many advances in CFD, heat transport modeling and validation of code for liquid metal flows needs to be improved. The poor predictive capabilities of today's code stem from the fact that there is no separation of heat and momentum transport within the code, where in fact there is a discrepancy between the thermal and the viscous length scales due to the low Prandtl number. The comparison of the Kolmogorov length scale, η, for the smallest momentum turbulence scales and the Corrsins temperature micro-scale, η_t, shows this [1]. Various ideas on how to solve this problem have been proposed [2, 3], where a separate system of equations is introduced in order to handle the temperature fluctuations. Presently used models of heat transfer to low-Prandtl fluids still need to be validated. The goal of this paper is to contribute with this type of development. To this end, several turbulence models implemented into the OpenFOAM code have been tested and validated against experimental data, that has been obtained in the TALL loop.

The TALL facility at the Royal Institute of Technology was constructed in order to study the flow and heat transfer behavior of lead-bismuth flows under forced and natural convection situations. The loop consists of two loops, a primary and a secondary one. The primary loop contains the test section, a flow meter, a pump, heating, piping, tanks and an intermediate heat exchanger, as can be seen in figure 1. It uses lead bismuth as the cooling liquid. The secondary loop on the other hand uses a high temperature oil (glycerol) as the cooling liquid. With a total height of 6.8 m, the facility is unique for simulation of natural circulation and dynamic responses [4].

The focus in this paper is on the steady state thermal hydraulics and heat transfer within a core



Primary loop:
Piping—33.4mm O.D. and 27.8mm I.D.
Material—AISI 316 stainless steel
Working fluid — LBE
Max flowrate — 15 liters/min
Preheating of piping — Rope heater

1–Valve for LBE
2–Core simulator, 22kW
3–Blind flange
4–Melting heater (6kW)
5–Sump tank for LBE
6–Observing window
7–Level sensor
8–Filter
9–Melting tank
10–Exhaust gas tank
11–HEPA filter
12–Expansion tank

Secondary loop:
Piping—26.7mm O.D. and 23.3mm I.D.
Piping material—carbon steel
Working fluid — Glycerol
Max flowrate — 260 liters/min
Preheating of piping — Band heater

13–Expansion tube
14–LBE–oil heat exchenger
15–EM Flowmeter
16–EM Pump
17–Oxygen meter

20–Oil–water heat exchenger
21–heater (3kW)
22–Sump tank for oil
23–Oil pump
24–Valve for oil
25–Flowmeter for oil

Figure 1 Schematic of the TALL loop facility [4] simulator with a capacity of 22 kW over a heated length of 0.87 m. The flow in the annulus is thermally developing and turbulent. The CFD calculations performed on the basis of the experiment were carried out with OpenFOAM.

The experimental work, which is used to compare the simulation results of section 1, was carried out at the TALL loop facility at the Royal Institute of Technology in Stockholm [4]. The test section was a fuel rod simulator with an overall heated length of 0.87 m and a diameter of 8.2 mm which formed an annulus channel with the housing which has an inner diameter of 19 mm. Several experimental simulations have been performed in order to investigate the impact of power and flow rate on the heat transfer. The inlet velocity was varied between 0.33 m/s and 2 m/s and the power was varied between 6 kW and 21 kW. The numerically investigated values are given in Table 1, which employ a constant power of 21 kW and velocities between 0.65 m/s and 2 m/s.

1. Modeling of TALL facility with OpenFOAM

For the computational simulation of the TALL facility, only the heated leg was considered. Therefore, just the 0.87 m long heated annulus was modeled with OpenFOAM, see part 2 in Figure 1. The mesh contains only hexahedral cells with smaller cells towards the walls and larger cells towards the center of the flow region. Since two different turbulence model classes were employed, which have different y^+ requirements, two different meshes are utilized. One, which keeps the dimensionless wall distance at $y^+>30$ and the other one keeping $y^+<1$. The latter is also the requirement that is recommended for low Prandtl number fluid simulations [4]. In order to keep the cell count at an acceptable level a 30° slab of the annulus region is meshed, see Figure 2. The mesh is adapted to the simulation set up, in order to stay within the requirements for y^+. For the high Reynolds number turbulence models, the requirements are not as strict, stating that $30 < y^+ < 150$ [5]. Therefore, the mesh is kept constant for the respective class of turbulence models.

Since the outer wall of the annulus was well insulated, an adiabatic boundary condition was applied at that wall. On the heated wall, the following boundary condition was applied,

$$q'' = \lambda \frac{dT}{dn} , \tag{1}$$

where λ is the molecular thermal conductivity of lead bismuth eutectic (LBE), q'' is the heat flux and dT/dn is the temperature gradient normal to the wall. The molecular thermal conductivity is used in this case, because close to the wall (first cell well within the thermal boundary layer) the mean temperature profile is dominated by the molecular heat transfer [6]. This condition is achieved due to LBE's very low molecular Prandtl number of about 0.0205 [7], which means that the thermal boundary layer is much thicker than the momentum boundary layer, which in this case is resolved due to a y^+ value of around 1.

The selected solver uses the Boussinesq approximation for density changes, whereas all other temperature dependent variables are kept constant. The reference temperature, T_{ref}, is obtained using the algebraic mean between the inlet and the outlet, calculated by iterating over a simple heat balance given by

$$T_{out} = T_{in} + \frac{P_{tot}}{W \cdot c_p} , \tag{2}$$

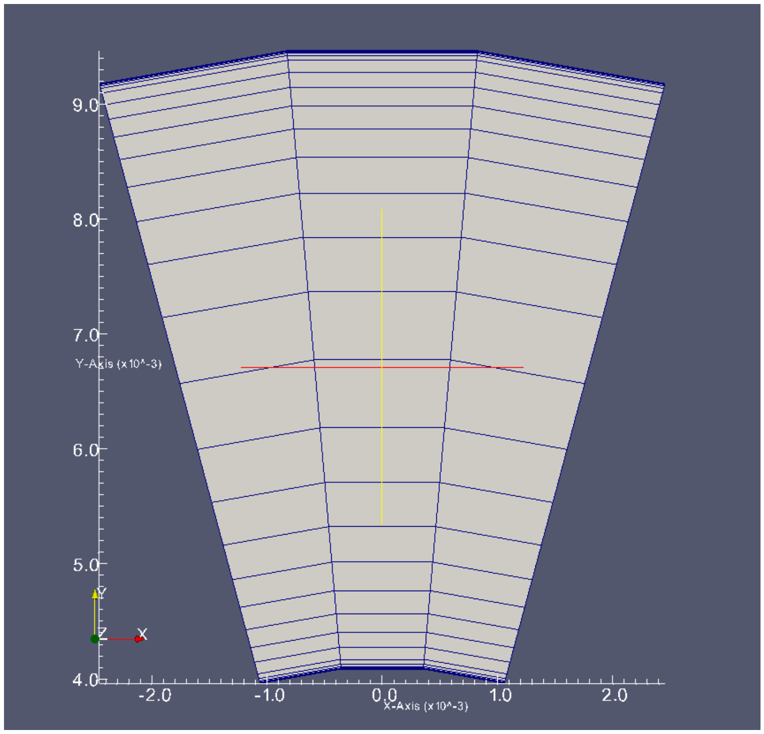


Figure 2 Cross section over the outlet of the modeled domain

where P_{tot} is the total power in the heated rod, T_{in} is the inlet temperature as given by the experiment, W is the mass flow rate through the annulus and c_p the specific heat capacity.

1.1 Simulation setups

The simulations were carried out with 6 different low Reynolds number turbulence models and 3 high Reynolds number turbulence models. The setup can be viewed in the test matrix in Table 1. The meshes employed have a cell count of 4800 and 36000 for the high Reynolds number turbulence models and the low Reynolds number turbulence models, respectively. In order to capture the performance of the turbulence models experiments with 3 different inlet velocities, u_{inlet}, were chosen. The following turbulence models were chosen for validation,

- Launder and Sharma k-ϵ (low-Re) [5],
- Lam and Bremhorst k-ϵ (low-Re) [5],
- Lien non linear cubic low Reynolds k-ϵ [8],
- Lien and Leschziner low Reynolds k-ϵ [9],
- Gibson and Dafa'Alla q-ζ (low-Re) [10],
- Menter k-ω-SST (low-Re) [5],

Table 1 Test matrix showing the different turbulence models and the input parameters.

Case #	u_{inlet} [m/s]	Power [kW]	T_{ref} [K]	Re
01	1.55	21	598.837	$9.91 \cdot 10^4$
02	2	21	594.597	$1.27 \cdot 10^5$
03	0.65	21	625.712	$4.37 \cdot 10^4$

- Renormalization group k-ϵ (high-Re) [5],
- Menter k-ω-SST (high-Re) [5] and
- standard k-ϵ (high-Re) [5].

The difference of the low Reynolds turbulence models and the high Reynolds turbulence models is the fact that the latter use wall functions as the boundary conditions for k, ϵ and ω. The turbulent Prandtl number, Pr_t, is set to the standard 0.85 in order to validate the standard models with standard input parameters.

2. Results and analysis of the numerical calculations

The predictions of the temperature field in dependence on the different turbulence models are compared to the experimental data from the TALL loop [4] and quantified using the root mean square error (RMSE) estimate over all 16 measurement points in the channel. The measurement points are located on 4 different levels in axial direction (0 cm, 29 cm, 58 cm and 87 cm) and in 4 different positions radially from the center (5.1 mm, 6.1 mm, 7.8 mm and 9.5 mm).

Due to measurement constraints, the wall temperature is not part of the data set [4]. However, the wall temperature is a crucial safety parameter. It was therefore approximated by extrapolation from the measurement data with help of the Nusselt number correlation for fully developed liquid metal flow in an annulus by Dwyer [7, pp. 456-457]. The predicted wall temperature does not contribute to the RMSE calculations.

The RMSE from the experimental data for each of the different turbulence models and different inlet velocities is presented in Table 2. As can be seen, the k-ω-SST model by Menter in low Re mode exhibits the best predictions at inlet velocities of 0.65 m/s and 1.55 m/s, whereas the k-ω-SST model in high Re mode shows superior predictions to the low Re mode at 2 m/s inlet velocity. It can be seen that the RSME for the 2 m/s cases are almost always half of the RSME of the 0.65 m/s cases.

In Figures 3 and 4 the radial temperature profiles for the two bounding inlet velocities, 0.65 m/s and 2 m/s, at the outlet of the annulus are presented. The four plots reflect the results shown in Table 2. The predictions for the lowest inlet velocity are far worse than the predictions for the highest inlet velocity, 2 m/s.

The Lam Bremhorst k-ϵ model neither follows the trend, nor predicts the experimental temperature

Table 2 RMSE estimate in Kelvin of the simulation data with respect to all measurement points, categorized by inlet velocity. Bold font highlighting the RMSE and italics highlighting the worst behavior.

Model	**0.65 m/s [K]**	**1.55 m/s [K]**	**2 m/s [K]**
Low Re models			
Menter k-ω-SST	**10.5013**	**5.9737**	5.0871
Lam and Bremhorst k-ϵ	*122.6314*	*24.6580*	*17.1864*
Launder and Sharma k-ϵ	14.4737	7.4536	6.1949
Lien non linear cubic k-ϵ	12.7421	6.7303	5.7361
Lien and Leschziner k-ϵ	16.3066	9.3606	7.8398
Gibson and Dafa'Alla q-ζ	13.2633	6.5422	5.4014
High Re models			
Menter k-ω-SST	13.2460	6.1045	**4.5155**
Renormalization group k-ϵ	16.3710	8.6811	7.2321
Standard k-ϵ	15.4358	8.2744	6.3319

as shown in 3a, which is very consistent with what the RMSE shows. For the high inlet velocity case, the trend of the temperatures is followed, but predictions of the temperatures fails.

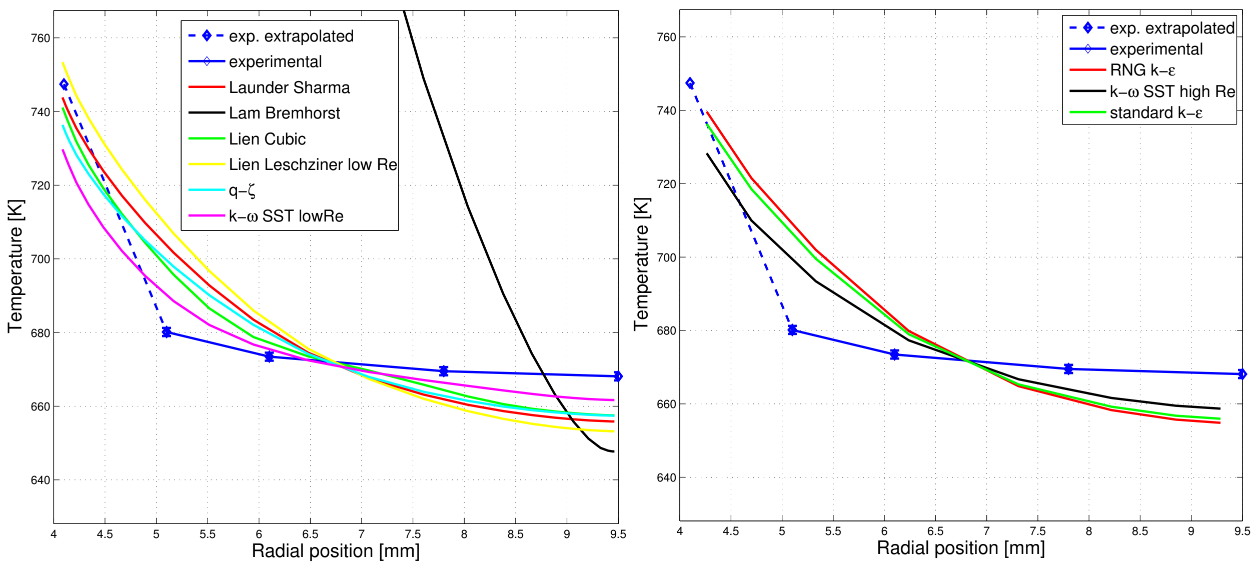
An analysis of the Froude number, $Fr = Gr/Re^2$, shows that all cases are forced convection. The ratios are $2.52 \cdot 10^{-3}$, $3.22 \cdot 10^{-4}$ and $1.56 \cdot 10^{-4}$ for inlet velocities 0.65 m/s, 1.55 m/s and 2 m/s, respectively.

3. Conclusions

Most turbulence models are able to predict the temperature distribution to a certain extend, with the models of the k-ω-SST series exhibiting the best behavior under all conditions tested. The Lam Bremhorst k-ϵ model completely fails to predict the temperatures and the trend of the temperatures for the low velocity case and only captures the trend for the high velocity case. In this series of tests it is by far the model with the worst prediction capabilities.

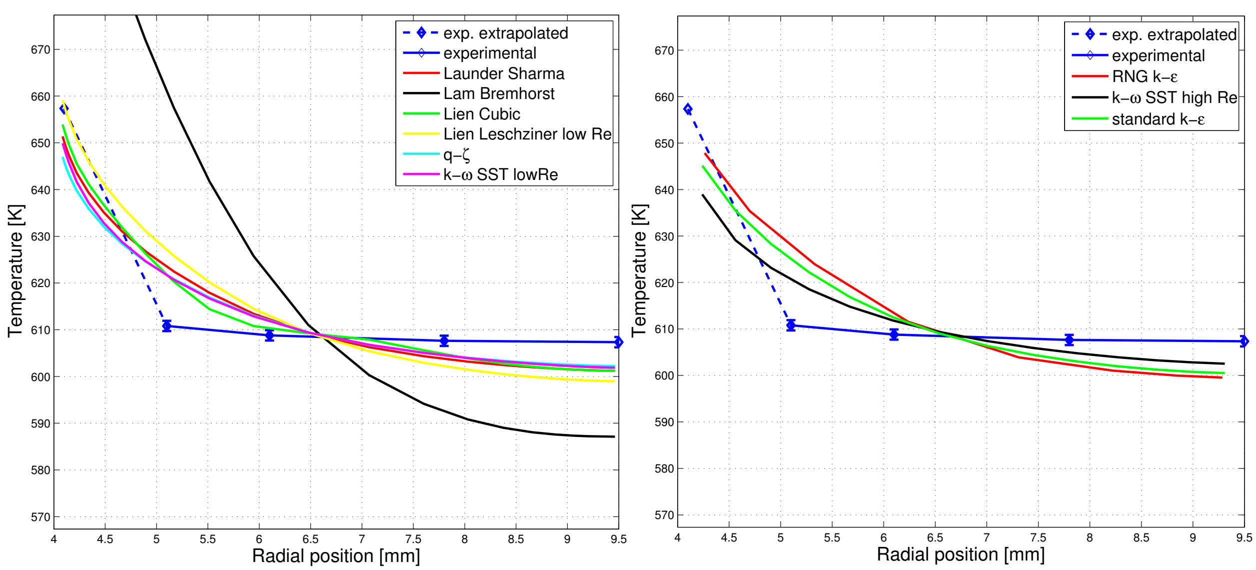
The comparison between low and high Reynolds number turbulence models of the same class (k-ϵ and k-ω) showed that in general the low Reynolds number models are superior to the high Reynolds number cases, except for the highest inlet velocity case, where the high Reynolds number k-ω-SST model outperforms the low Reynolds number k-ω-SST model.

Overall, there is a trend that accompanies all turbulence models in this test series, namely deteriorating predictive capabilities with decreasing Reynolds number. With decreasing inlet velocity the temperature gradient between inlet and outlet, as well as from the inner to the outer wall, increases. Therefore, the density gradient increases, leading to larger buoyancy forces. These buoyancy forces might influence the behavior of the turbulence models, since the models are not specifically designed for buoyant flows.



(a) Low Re models (b) High Re models

Figure 3 Comparison of turbulence models with experimental data for the case of 21 kW power and 0.65 m/s inlet velocity, showing the radial temperature profile at the exit.



(a) Low Re models (b) High Re models

Figure 4 Comparison of turbulence models with experimental data for the case of 21 kW power and 2 m/s inlet velocity, showing the radial temperature profile at the exit.

Furthermore, it should be investigated how the turbulent Prandtl number influences the predictive capabilities. DNS simulations by Kawamura et al. show that it is not constant over the whole domain, although close to unity towards the core of the flow for high turbulence and low molecular Prandtl numbers [11].

4. References

[1] G. Grötzbach, “Revisiting the resolution requirements for turbulence simulations in nuclear heat transfer”, Nuclear Engineering and Design, 2011 Feb.

[2] L. Carteciano, Entwicklung eines Turbulenzmodells für Auftriebsströmungen, Ph.D. thesis, Forschungszentrum Karlsruhe, Umwelt und Technik, Karlsruhe, Germany, 1996 Nov.

[3] M. Youssef, Y. Nagano and M. Tagawa, “A two-equation heat transfer model for predicting turbulent thermal fields under arbitrary wall thermal conditions”, International Journal of Heat and Mass Transfer, 1992 Nov, 35, no. 11, 3095–3104.

[4] W. Ma, A. Karbojian, T. Hollands and M. K. Koch, “Experimental and numerical study on lead–bismuth heat transfer in a fuel rod simulator”, Journal of Nuclear Materials, 2011 Apr.

[5] D. Wilcox, Turbulence modeling for CFD, DCW Industries, La Canada Calif., 3rd ed., 2006.

[6] M. Ould-Rouiss, L. Redjem-Saad, G. Lauriat and A. Mazouz, “Effect of Prandtl number on the turbulent thermal field in annular pipe flow”, International Communications in Heat and Mass Transfer, 2010 Oct, 37, no. 8, 958–963.

[7] Handbook on lead-bismuth eutectic alloy and lead properties, materials compatibility, thermal-hydraulics and technologies, OECD Nuclear Energy Agency, Issy-les-Moulineaux France, 2007 ed. ed., 2007.

[8] F. Lien, W. Chen and M. Leschziner, “Low-Reynolds-number eddy-viscosity modelling based on non-linear stress-strain-vorticity relations”, in Proc. 3rd Symp. on Engineering Turbulence Modelling and Measurements, Crete, Greece, 1996 May .

[9] F. Lien and M. Leschziner, “A pressure-velocity solution strategy for compressible flow and its application to shock/boundary-layer interaction using second-moment turbulence closure”, Journal of fluids engineering, 1993, 115, no. 4, 717–725.

[10] M. M. Gibson and A. A. Dafa’Alla, “Two-equation model for turbulent wall flow”, AIAA Journal, 1995 Aug, 33, no. 8, 1514–1518.

[11] H. Kawamura, H. Abe and Y. Matsuo, “DNS of turbulent heat transfer in channel flow with respect to Reynolds and Prandtl number effects”, International Journal of Heat and Fluid Flow, 1999 Jun, 20, no. 3, 196–207.