Numerical simulation of turbulent forced convection in liquid metals

S Vodret, D Vitale Di Maio and G Caruso

“Sapienza” University of Rome – Dep. of Astronautic, Electrical and Energy Engineering (DIAEE) – Corso Vittorio Emanuele II, 244 – 00186 Rome, IT

Email: gianfranco.caruso@uniroma1.it

Abstract. In the frame of the future generation of nuclear reactors, liquid metals are foreseen to be used as a primary coolant. Liquid metals are characterized by a very low Prandtl number due to their very high heat diffusivity. As such, they do not meet the so-called Reynolds analogy which assumes a complete similarity between the momentum and the thermal boundary layers via the use of the turbulent Prandtl number. Particularly, in the case of industrial fluid-dynamic calculations where a resolved computation near walls could be extremely time consuming and could need very large computational resources, the use of the classical wall function approach could lead to an inaccurate description of the temperature profile close to the wall. The first aim of the present study is to investigate the ability of a well-established commercial code (ANSYS FLUENT v.14) to deal with this issue, validating a suitable expression for the turbulent Prandtl number. Moreover, a thermal wall-function developed at Université Catholique de Louvain has been implemented in FLUENT and validated, overcoming the limits of the solver to define it directly. Both the resolved and unresolved approaches have been carried out for a channel flow case and assessed against available direct numerical and large eddy simulations. A comparison between the numerically evaluated Nusselt number and the main correlations available in the literature has been also carried out. Finally, an application of the proposed methodology to a typical sub-channel case has been performed, comparing the results with literature correlations for tube banks.

1. Introduction

Differently than water, liquefied metals are characterized by a very low range of Prandtl numbers (of the order of $10^{-2}$ to $10^{-3}$ due to their high heat conductivity). This means that molecular transfer of momentum is significantly less intense than the molecular thermal diffusivity and so, for turbulent flows, the heat transfer in the channel could be essentially molecular.

Existing engineering computer codes used to simulate turbulent heat transfer adopt the turbulent Prandtl number model based on the Reynolds analogy, that states a coupling between velocity and temperature fields. Although this has already been validated for fluids with a Prandtl number of the order of unit or higher (such as water or air), the applicability for low Prandtl number fluids has to be reassessed.

Since experiments are scarce, expensive and difficult to perform, detailed data regarding turbulent heat transport at different Reynolds and Prandtl numbers are currently available in literature thanks to advanced simulation approaches such as direct numerical simulations (DNS) and large eddy simulations (LES). These data are then used to suggest new laws (wall functions) to be used in Reynolds Averaged Navier-Stokes (RANS) simulations, which provide for most cases the required accuracy.
accuracy and the lowest cost. An exhaustive list of issues to face within liquid metals Computational Fluid Dynamics (CFD) simulations may be found in literature [1].

A first issue of these simulations is the computation of heat diffusivity through the value of kinematic eddy viscosity $\nu_t$ and a turbulent Prandtl number. Since turbulent transport of momentum and heat or mass is due to the same mechanism (eddy mixing), the $Pr_t$ value is classically supposed to be approximately 1. From experimental data with water and air as working fluids, indeed, $Pr_t$ has an average value of 0.85 and ranges from 0.7 to 0.9 depending on the Prandtl number of the fluid. This is not valid for liquid metals and different expressions have been proposed for this parameter [2, 3, 4].

The first aim of the paper is to assess ANSYS FLUENT capability to perform wall resolved RANS simulations of heat transport in liquid metals, by using suitable expressions available in the literature for the turbulent Prandtl number. Moreover, the unresolved approach has been analyzed. When the wall function approach is chosen in RANS simulations a proper wall function must be selected to link correctly the value of a certain quantity at the first grid point to the one assumed at the wall (i.e. the temperature or the velocity). While for the velocity profile the linear and log laws are assessed in several cases, the same is not possible to say dealing with the temperature profile in low Pr flows, since no logarithmic region is visible. An original methodology to overcome this limit has been developed and validated using a new law for the thermal wall-function proposed by Bricteux [5, 6].

### 2. Turbulent Prandtl number correlations

The enhanced heat transport in liquid metals, due to their high thermal conductivity, requires new correlations to link turbulent viscosity to turbulent heat diffusivity, by means of the turbulent Prandtl number ($Pr_t = \nu_t / \alpha_t$). The needing of assessment is evidenced by the numerous correlations proposed in the past through experimental campaigns or analytical studies. Despite this second approach allows a deeper understanding of the physics behind, the first one is more accurate and it is thus preferred. Models of Reynolds [2], Kays [3], Weigand [4, 7], Aoki [8], Dwyer [9] and Cheng and Tak [10] are only some of the suggested correlations and an overview of the other ones can be found in [11].

Among these correlations, two of them were chosen in this study for their assessment: the model of Reynolds [2] and the model of Kays [3], respectively:

$$Pr_t = (1 + 100 Pe^{-0.5}) \left( \frac{1}{1 + 120 Re^{-0.5}} \right)$$

$$Pr_t = 0.85 + \frac{0.7 \nu}{Pr \nu_t}$$

### 3. Wall resolved approach: RANS of turbulent heat transfer

Many channel flow data are available in literature thanks to numerous DNS and LES performed by several authors, such as Kawamura [12] and Abe [13], who performed DNS at $Re_t$ up to 1020 with Prandtl numbers ranging from 0.025 to 0.71, or Bricteux et al. [5], who made DNS at $Re_t = 180$ and 590 case with $Pr = 0.01-0.025$ together with LES up to $Re_t = 2000$ with the same range of Pr number. The works of Moser [14] and Tiselj [15] are also available. A 2D turbulent RANS analysis is performed at turbulent Reynolds number ($=u_c \delta / \nu$) of 395 with constant wall heat flux and $Pr = 0.025$ to be compared with the results of the DNS performed by Kawamura [16].

#### 3.1. The channel flow geometry and mesh

The channel flow geometry and mesh are reported in figure 1. The domain consists of two infinite plates separated by a distance $2 \delta$ and it is reproduced through a 2D section on the x-y plane. It is assumed that the fluid is flowing along the x axis. The geometry model is characterized by dimensions $L_x \times L_y = 2\pi \delta \times 2\delta$ (where $\delta$ is the half channel width) and it is nodalized with $N_x \times N_y = 64 \times 48$ divisions. The mesh is uniform among the streamwise direction while the “bias” tool was used to place more cells next to the wall, ensuring that $y^+ < 1$ as required for wall resolved simulations.
3.2. Equations and case setup

The RANS equations [17] are modified taking account the hypothesis of steady state and incompressibility. Since the steadiness is simulated through periodic boundary conditions in the streamwise direction the resolution of the energy equation does not converge to a steady state result since the cyclic boundary condition at the channel’s inlet and outlet results in a continuous increase of the fluid’s absolute temperature in the flow direction. The transformed form of the energy equation is thus employed to tackle this issue [17].

The realizable k-ε and the SST k-ω models are used for the specification of νt to solve RANS equations: two transport equations for these quantities are solved assuming μt isotropic. Further formulative details of these methods are available in [17].

Field variables (stored at cell centers) must be interpolated to the faces of the control volumes to solve the discretized transport equations of momentum, turbulent kinetic energy, rate of dissipation of turbulent kinetic energy and temperature. To do this, the second order upwind spatial discretization was used, which ensures a second order accuracy. Gradients of solution variables are also required in order to evaluate diffusive fluxes, velocity derivatives, and for higher-order discretization schemes. A least squares cell-based gradient interpolation method was used, which is recommended for regular meshes, as it is accurate and minimizes false diffusion. Cell-face pressures have to be computed when using the segregated solver. The linear interpolation scheme was used in this case.

3.3. Comparison with DNS results

Obtained profiles are reported in figure 2, together with the reference DNS. As one can see, for the realizable k-ε model there is a good agreement between the simulations, that match up to y+ ≅ 60, while the velocity is slightly underestimated in the center of the channel, where a wake is formed and the log law is not more valid. As it concerns the SST k-ω model, despite the perfect matching is up to y+ ≅ 10, the overall behavior is in good agreement with the reference DNS.

As expected, the linear sub-layer extends up to y+ ≅ 5, while for 10 < y+ < 40, where both viscosity and turbulence play an important role to the total momentum transfer, there is the buffer layer, which forms the transition between the viscous wall region and the logarithmic wall region of the boundary layer. At least, the log-law layer starts from y+ > 50. The temperature field has been solved through the classic, Reynolds and Kays correlations for Prt (setting a value of 0.85 for the first one). The reported dimensionless temperature profile is built using θ+ = (Tw - T)/Tτ, where Tτ is the friction temperature, = qw/(ρ cp uτ). The profiles of the Reynolds’ and Kays’ turbulent Prandtl numbers for the realizable k-ε model are reported in figure 3, together with the computed Prt using data from Kawamura [16]. The three temperature profiles are reported in figure 4 together with the results of the DNS by Kawamura [16] and the linear law for the laminar thermal sub-layer. The dashed curve represents the linear relation θ+ = Pr y+ valid in the immediate vicinity of the wall.

As one can see, the classic approach fails to predict the temperature profile for this very low Prandtl number, underestimating the temperature difference with a relative error of 29.62 %. Despite the good agreement of the turbulent Prandtl number of the Kays correlation with the DNS’ one (see figure 3), the θ+ profile is slightly underestimated in the center of the channel. This might be because
of the enhanced wall function of the realizable k-ε model used, which gives a too low value of the wall temperature. Obtained profile of $\theta^+$ using the SST k-ω model is reported in figure 5.

Figure 2. Dimensionless velocity profile for $Re_\tau = 395$.

Figure 3. Values of $Pr_\tau$ used for the realizable k-ε simulations.

In this case the Kays correlation gives a more accurate prediction of the temperature profile, while the Reynolds’ one gives a slightly overestimation of $\theta^+$.

Figure 4. Mean temperature profile at $Re_\tau=395$ and $Pr=0.025$ - realizable k-ε model.

Figure 5. Mean temperature profile at $Re_\tau=395$ and $Pr=0.025$ - SST k-ω model.

In table 1 the obtained Nusselt numbers are reported, computed using the following formula:

$$Nu = \frac{q_w 2\delta}{k(T_w - T_h)}$$  \hspace{1cm} (3)

Table 1. Computed Nusselt number for $Pr=0.025$.

| Case            | Nu  | Case          | Nu   |
|-----------------|-----|---------------|------|
| DNS             | 6.52| Classic approach | 8.19 |
| Reynolds correlation | 6.06 | Kays correlation | 6.52 |

The classic approach clearly overestimates the heat transfer coefficient, while the other results obtained with the Reynolds and Kays correlations are in good agreement with the reference data.
3.4. RANS calculations for \( \Re \tau = 590 \) and 2000: comparison with LES results

Similar calculations for RANS at \( \Re\tau = 590 \) and 2000 (using LES performed by Bricteux et al. \[6, 18\] as reference) have been performed and just the velocity and temperature profiles are reported below. The mesh was modified each time in order to maintain always the wall resolved approach, ensuring \( y_+ \leq 1 \). Comparing these results with the same obtained for \( \Re\tau =395 \), a better agreement with literature data is detectable as \( \Re \tau \) increases, especially for velocity profiles. This is due the better behavior at higher \( \Re \tau \) of the turbulence models. As it concerns the temperature profiles, if at \( \Re\tau =395 \) both Reynolds and Kays’ correlations gave similar results, here the second one always gives better results and thus might be chosen for future simulations.

4. Wall function approach

So far simulations on a very fine grid have been performed, ensuring that the first node along the \( y \) axis was positioned in such a way that \( y_+ \leq 1 \). This leads to very heavy simulations, which are not feasible in industrial field because this method devotes a substantial fraction of the computational time to the near wall region, where velocity and temperature profiles are steep due to viscous effects.

Another way to perform RANS simulations is the so called “wall function approach”. This method allows to use an uniform mesh, putting the first point of the grid among the \( y \) axis at \( y_+ \) much greater than one and using semi empirical formulas called “wall functions” to link the viscosity affected region between the wall and the fully-turbulent region. In this section, the wall function proposed by Bricteux et al. \[5\], applicable for low \( \Pr \) numbers, is assessed.

Any solver offers several wall functions selectable, giving the user the possibility to build its own. In this case, FLUENT replaces with customized wall functions the standard ones, based on the work of Launder and Spalding \[19\] using \( U^* \) and \( y^* \), defined below:

\[
U^* = \frac{\rho U_p C_{\mu}^{0.25} \kappa_{\mu}^{0.5}}{\tau_w} \quad \text{and} \quad y^* = \frac{\rho C_{\mu}^{0.25} \kappa_{\mu}^{0.5} y_p}{\mu}
\]

These functions are based on the universal linear and log laws for velocity previously showed and, being built for conventional fluids, the law for temperature is dependent from the velocity one according to the Reynolds analogy. Since no log-law is visible in the temperature profile dealing with liquid metals, a new wall function has to be used. The law proposed by Bricteux et al. \[5\] is:

\[
\theta^* = \frac{\Pr}{\kappa} \log \left( 1 + \frac{\kappa}{\Pr y^*} \right)
\]

This law was compared to DNS and LES data for a range of Péclet numbers (\( \Pr = 0.025-0.01 \) and \( \Re\tau =180-2000 \)) and it fits well the data in the range \( y^* =70-300 \) where the logarithmic law for the velocity profile is applicable.

Since ANSYS FLUENT does not allow the user to define a custom wall function for the temperature field, a strategy to overcome this limit is proposed in this study and consists in a gradual solution of the flow field, as explained below.

First, the velocity field is computed using the enhanced wall treatment. Once the solution is converged, a user defined wall function for the velocity is hooked up in the solver, thus the momentum and turbulence equations are frozen, while the energy model is activated. When the calculation is run again, the user defined wall function will not affect the velocity field, but will be used by the solver to compute the temperature at the wall. Obviously this procedure is applicable only with the hypothesis of incompressibility of the fluid and considering constant its thermal properties.

The UDF employed is coded reasoning on the standard wall functions (since these wall functions are replaced by custom’s one).

Since this approach has been tested using a \( y_+ =100 \), only the \( \Re\tau =2000 \) case has been solved, as a too coarse mesh would have been necessary to solve the other two cases. The obtained temperature
profile is reported in figure 6, solving the energy equation using the Kays correlation for Pr, since it was more rewarding than the Reynolds’ one in the wall resolved approach.

From the results it is evident that the thermal wall function predicts accurately the wall temperature profile, providing results as similar to the reference ones as those obtained using the wall resolved approach, despite the coarse and uniform mesh which drastically reduces the computational time.

5. Forced convection heat transfer in liquid metals

The turbulent heat transfer in circular and flat ducts is one of the most investigated configuration of the liquid metal heat transfer with lead, Lead Bismuth Eutectic (LBE), mercury or sodium.

For forced convection flows of liquid metals, dimensional analysis shows that \( \text{Nu} = \frac{h D}{k} = f(\text{Pe}) \), where \( \text{Pe} \) is the Peclet number \( (\text{Pe} = \rho c_p D (u_{av})/k = \text{Pr} \cdot \text{Re}) \) [20]. A very large number of correlations, both theoretical and empirical, has been developed in the last sixty years for the constant wall temperature and constant heat flux cases, evidencing the complexity and difficulties related to the study of this phenomenon. Three models are considered in this study.

The first one was proposed by Lyon [21] and it is suitable for pure metals with a perfect wettability of the walls:

\[
\text{Nu}_{\text{Lyon}} = 7 + 0.025 \text{Pe}^{0.8}
\]

The second correlation was obtained by Lubarsky and Kaufman [22] and it should be more representative of real facilities whereby liquid metals are seldom pure:

\[
\text{Nu}_{\text{Lub}} = 0.625 \text{Pe}^{0.4}
\]

This correlation was recently reviewed by Cheng and Tak [10], leading to:

\[
\text{Nu}_{\text{Chen}} = A + 0.018 \text{Pe}^{0.8}
\]

Where

\[
A = \begin{cases} 
4.5 & \text{for } \text{Pe} < 1000 \\
3.6 & \text{for } \text{Pe} > 2000 
\end{cases}
\]

Global heat transfer performances of the considered cases are analyzed in this section through the obtained values of the Nusselt number. A comparison of the numerically calculated Nu with the reference and the computed through the most famous literature expressions ones is made. As reference Nu parameters, the one obtained through the DNS of Kawamura [16] for the \( \text{Re}_\tau = 395 \) case and the two obtained through the LES of Bricteux et al. [6, 18] for the \( \text{Re}_\tau = 590 \) and 2000 are used. The Lyon, Lubarsky and Kaufmann [22] and Cheng and Tak [10] correlations were used. All the considered parameters are reported in table 2 and in figure 7.

### Table 2. Nusselt numbers and main parameters involved.

| \( \text{Re}_\tau \) | \( T_w \) | \( T_b \) | \( k \) | \( U_{av} \) | \( \text{Re} \) | Pe | \( \text{Nu}_{\text{CFD}} \) | \( \text{Nu}_{\text{ref}} \) | \( \text{Nu}_{\text{Lyon}} \) | \( \text{Nu}_{\text{Lub}} \) | \( \text{Nu}_{\text{Chen}} \) |
|---|---|---|---|---|---|---|---|---|---|---|---|
| [-] | [K] | [K] | [W/m K] | [m/s] | [-] | [-] | [-] | [-] | [-] | [-] | [-] |
| 395 | 303.027 | 300 | 101.26 | 17.59 | 13897 | 347.43 | 6.52 | 6.49 | 6.44 |
| 590 | 303.789 | 300 | 67.79 | 18.94 | 22347 | 558.67 | 7.89 | 7.85 | 7.34 |
| 2000 | 306.773 | 300 | 20.00 | 22.67 | 90686 | 2267.14 | 14.39 | 13.74 | 12.30 |

As expected, the Nu numbers from CFD calculations \( \text{Nu}_{\text{CFD}} \) and \( \text{Nu}_{\text{ref}} \) (in good agreement with each other) are slightly higher than Lubarsky and Kaufman and Cheng ones, since CFD does not consider the occurrence of impurities and the wettability problem, leading to higher Nu values. Lyon’s correlation is however too optimistic, since it always overestimates the values of Nu (from 48.62 % to 32.66% for higher Re number). Despite it considers the same thermo-fluid dynamic conditions of CFD simulations, the disagreement might be because of the few set of experimental data used to extrapolate Lyon’s formulation.
6. The subchannel case

The evaluation of the heat transfer in a real subchannel with constant and cosine heat flux is carried out, on the basis of the previous study of the channel flow. Both the wall resolved and wall function approaches have been used to compute the average Nusselt number in the facility, making use of both the enhanced and user defined wall function for the latter case. The models of Mikityuk [23] and Ushakov [24] were used to assess the obtained results.

The simulations performed are based on the ALFRED (Advanced Lead Fast Reactor European Demonstrator) triangular core lattice geometry. This lead cooled reactor is the third considered within the LEADER (Lead - cooled European Advanced Demonstration Reactor) project, and is thought to be realized in the short term, relying on presently available technology. This scaled down reactor is thus designed using solutions as much as possible close to the adopted reference conceptual design but considering the essential need to proceed to construction in a short time frame. Many design details of this reactor can be found in [25], here the main ones are reported in table 3. The domain used in the following simulations consists of 1/6 of a subchannel 0.7 m long (composed by the asset zone of 0.1 m and the active length of 0.6 m).

| Parameter (Core)       | Value   | Unit |
|------------------------|---------|------|
| Thermal power          | 300     | MWh  |
| Coolant mass flow rate | 25984   | kg/s |
| Coolant average velocity, $\bar{u}$ | 1.4     | m/s  |
| Coolant inlet temperature, $T_{in}$ | 401.5   | °C   |
| Coolant outlet temperature, $T_{out}$ | 480     | °C   |

| Parameter (Fuel pin)                          | Value   | Unit |
|-----------------------------------------------|---------|------|
| Total number of FA/Pins per FA               | 171 / 127 | -    |
| Relative rugosity, $\varepsilon$             | 5.17$\cdot$10^{-4} | -    |
| Cladding outer radius, $r$                   | 5.25$\cdot$10^{-3} | m    |
| Pitch to diameter ratio, p/D                 | 1.32    | -    |
| Asset length / Active height                 | 0.1 / 0.6 | m    |

6.1. The subchannel mesh and setup

The estimation of the $y_{i}^{+}$ is crucial to create the mesh to employ. Since both the wall resolved and the wall function approach have been tested, two meshes are built.

As usual, for the first approach the condition to respect is $y_{i}^{+} < 1$, thus:

$$ y_{i}^{+} = \frac{y_{i} u_{i}}{\mu} < 1 $$

Taking into account the values of the flow at the half of the active length, $y_{i}=2.67\cdot10^{-3}$ mm.

In order to gain a regular and hexahedral mesh the mapped face meshing and the multi-zone method have been used. To put more cells near the rod, the inflation method has been employed, setting $2.67\cdot10^{-3}$ mm as first layer height for 10 layers and a growth equal to 1.43.
The mesh, composed by 240 divisions among the z axis, 19 divisions in the radial direction and 5 in the polar one (22800 elements), is reported in figure 8. As it concerns the wall function approach, the selected value for $y^+$ is $\approx 200$. This will imply that $y=0.5339$ mm.

This choice leads to a mesh with only 2 divisions in the radial direction (while the divisions among the z axis and the polar direction are unvaried), leading to a mesh composed by 4338 nodes and 2400 elements (figure 9).

The two approaches have been solved for constant and cosine heat flux using the realizable k-ε model and solving the momentum and energy equations simultaneously (being lead modeled with temperature dependent quantities). The strategy described previously is then inapplicable. As it concerns the wall function approach, both the enhanced wall function and the new wall function have been tested. The considered expression for the cosine heat flux is:

$$q^* = 1.209 \cdot q_{\text{const}}^* \cdot \cos\left[\frac{\pi (z - 0.4)}{0.9}\right]$$  \hspace{1cm} (11)

where $q_{\text{const}}^* = 697961$ W/m$^2$ and the extrapolated length is 150% of the active length.

The SIMPLE algorithm is used for the pressure-velocity coupling, and for the spatial discretization the least square cell based method is used for gradient, the standard one for pressure and the second order upwind for every other quantity.

![Figure 8. Isometric view of the wall resolved case mesh.](image)

![Figure 9. x-y plane section view of the wall function case mesh.](image)

### 6.2. Results and discussion

The obtained Nusselt numbers for the constant heat flux and the cosine one for the wall resolved (WR) approach, the enhanced wall function (EWF) and the new wall function (NWF) ones are reported in figure 10 and in figure 11 and summarized in table 4 and in table 5.

Firstly, for the constant heat flux case two Nu numbers are obtained. The first one, $\overline{\text{Nu}}$, is averaged among the whole subchannel while the second one is averaged from $z=0.2$ m. The distinction is made seeing as in this case the “inlet effect” might be too sharp, since the fluid finds immediately a high heat flux, leading to unphysical values of Nu. Starting the average from $z=0.2$ m removes this effect. An analysis of a periodic channel with constant heat flux could elucidate this duality, but ANSYS FLUENT v.14 does not allow periodic analysis when the fluid properties are temperature dependent. This is not valid for the cosine heat flux case, since the fluid finds a gradually growing heat flux that avoids the above mentioned behavior.

Looking at these results, one can easily notice that the wall resolved and the new wall function approaches give similar results for Nu, but in this second case the number of iterations and the computational time are reduced. Since the two models of Mikityuk and Ushakov give an average value for Nu, they can be used as reference for any situation. As usual, the CFD obtained Nu number is higher than the experimentally obtainable with the models because of the non-perfect wettability of the wall and because of liquid metals impurities. Giving the Ushakov’s model a higher value for Nu, this is the closer one to the CFD results. Finally, the enhanced wall treatment clearly underestimates the value of Nu, highlighting again how classic approaches are not suitable for liquid metals.
Figure 10. Nu axial profile in the subchannel for constant heat flux.

Figure 11. Nu axial profile in the subchannel for cosine heat flux.

Table 4. Average Nu values with mesh statistics and computational time.

| Method                    | Constant heat flux | Cosine heat flux |
|---------------------------|--------------------|------------------|
|                           | Nu_{tot}           | Nu_{0.2}         | Nodes | Elements | Time [s] | Iterations |
| Wall resolved             | 17.33              | 15.76            | 16.95 | 28920    | 22800    | 390        | 696        |
| Enhanced wall function    | 14.52              | 13.88            | 14.41 | 4338     | 2400     | 25         | 307        |
| New wall function         | 17.03              | 16.72            | 16.97 | 4338     | 2400     | 65         | 293        |
| Mikityuk [23] / Ushakov [24] | 16.13 / 16.91      |                  |       |          |          |            |            |

Table 5. Relative errors on Nu.

| Method                    | Constant heat flux | Cosine heat flux |
|---------------------------|--------------------|------------------|
|                           | Nu_{tot} | \epsilon_{Mik} | \epsilon_{Ush} | Nu_{tot} | \epsilon_{Mik} | \epsilon_{Ush} |
| Wall resolved             | 17.33    | 7.4%            | 2.5%            | 15.76    | -2.3%            | -6.8%            |
| Enhanced wall function    | 14.52    | -10%            | -14.1%          | 14.41    | -17.9%          | -14.8%          |
| New wall function         | 17.03    | 5.6%            | 0.7%            | 16.72    | 3.7%            | 1.1%            |

7. Main issues for the applicability in real cases

Flows involved in real industrial applications are characterized by the variation of density and the other thermo-physical properties with temperature. This implies that the continuity, momentum and energy equations must be solved simultaneously and being every flow parameter variable in space, any user defined function applied (in this case, the thermal wall function) must be as general as possible to account this variability properly.

Whether ANSYS FLUENT is used to deal with these cases, some issues may arise due some limits that are entailed in the code. Firstly, all the flow’s equations must be solved at the same time and then the step by step strategy proposed for the presented channel flow case is not applicable. Secondly, the value of $u_\tau$ is not accessible through any cell macros in ANSYS FLUENT. A way to overcome this limit might be to preliminary compute this quantity’s value in some sections of the domain of a real configuration (i.e. a subchannel) and then to impose a constant value in a thermal UDF.

Thus, a solution strategy to solve real cases through ANSYS FLUENT is the simultaneous solving of all the flow equations using directly the thermal wall function with a constant value of $u_\tau$, but this approach has two limitations: a possible inaccuracy of the velocity field due the improper wall function used for the velocity and the imposition of a constant value for $u_\tau$ in the whole domain instead of the local one. Studies addressed to analyze the effect of these limits using this CFD software have then to be carried to validate the wall function approach in liquid metals for industrial applications.
8. Conclusions
The inadequacy of the Reynolds analogy with the classic turbulent Prandtl number concept for liquid metals has been confirmed and other expressions, such as the Reynolds’ and Kays’ ones have been assessed through wall resolved RANS calculations. Simulations have been carried in a channel flow for several Re cases at Pr=0.025. Thanks to its dependence from local values, the Kays’ expression is more accurate and seems more eligible for calculations in complex geometries. Afterwards a wall function approach has been investigated, testing a wall function law for the temperature field, developed by Bricteux et al. [5]. Both this law and the step by step methodology proposed in this study for its implementation in the solver gave rewarding results using ANSYS FLUENT comparing to DNS and LES analyses available in the literature. The comparison of the obtained Nusselt numbers with the reference ones and those coming from other available models, highlighted how the Lubarsky and Kaufman correlation seems to be the most accurate for the considered cases.

Moreover, a real case study has been carried out and from the comparison of the Nusselt numbers obtained it has been highlighted how the wall resolved and the new wall function approaches give results in good agreement with the reference ones, but with a drastically reduced computational time in the second case. Finally the limits of the adopted strategy to implement the new thermal wall function in real cases are discussed, showing a possible way to overcome them.

References
[1] Grötzbach G 2013 Nucl. Eng. Des. 264 41-55
[2] Reynolds A 1975 Int. J. Heat Mass. Tran. 18 1055-69
[3] Kays W 1994 J. Heat Trans-T. ASME 2 (116) 284-95
[4] Weigand B, Ferguson J and Crawford M 1997 Int. J. Heat Mass. Tran. 17 (40) 4191-6
[5] Bricteux L, Duponcheel M, Winckelmans G, Tiselj I and Bartosiewicz Y 2012 Nucl. Eng. Des. 246 91-7
[6] Bricteux L, Duponcheel M, Manconi M and Bartosiewicz Y 2012 in Proc. 6th European Thermal Sciences Conference (Eurotherm 2012) (Poitiers, France, 4-7 September 2012)
[7] Jisha M and Rieke H 1979 Int. J. Heat Mass. Tran. 22 (11) 1547-55
[8] Aoki S 1963 Bull. Tokyo Inst. Tech. 54 63-73
[9] Dwyer O 1966 Atom. Energy Rev. 1 (4) 3-92
[10] Cheng X and Tak N 2006 Nucl. Eng. Des. 236 385-93
[11] Hasan B 2007 vol X no 1 (Nahrain University, College of Eng. Journal (NUCEJ)) pp 53-64
[12] Kawamura H, Abe H and Matsuo Y 1999 Int. J. Heat Fluid Fl. III (20) 196-207
[13] Abe H, Kawamura H and Matsuo Y 2004 Int. J. Heat Fluid Fl. III (25) 404-19
[14] Moser R, Kim J and Monsour N 1999 Phys. Fluids IV (11) 943-5
[15] Tiselj I, Berar A and Cizelj L 2011 "DNS database of temperature fluctuations in channel flows for liquid metals," Technical report, THINS project, "Josef Stefan" Institute.
[16] Kawamura H http://murasun.me.noda.tus.ac.jp/turbulence/poi/poi.html#poi395 [Online]
[17] ANSYS 2009 "ANSYS FLUENT 12.0 Theory Guide".
[18] Bricteux L, Duponcheel M, Manconi M, Winckelmans G and Bartosiewicz Y 2014 Int. J. Heat Mass. Tran. 75 470-82
[19] Launder B and Spalding D 1974 Comput. Method. Appl. M. 3 269-89
[20] Lienhardt J H I and Lienhardt J H V 2008 A heat transfer textbook (Cambridge, MA, USA, Phlogiston Press)
[21] Lyon R 1952 Liquid Metals Handbook (Washington: A.E.C. and Dept of the Navy)
[22] Lubarsky B and Kaufman S J 1955 NACA Technical note 3366
[23] Mikityuk K 2009 Nucl. Eng. Des. 239 680-7
[24] Ushakov P, Zhukov A and Matyukhin M 1977 High Temperature 5 (15) 868-73
[25] Ponciroli R, Bigoni A, Cammi A, Lorenzi S and Luzzi L 2014 Prog. Nucl. Energ. 71 15-29