Proceedings of the 18th UK Heat Transfer Conference

9-11 September 2024, Birmingham

UKHTC2024-044

NUMERICAL INVESTIGATION OF THE MAINLY AXIAL FLOW IN MIXED CONVECTION REGIME WITHIN TUBE BUNDLES

R. Vicente Cruz[1*], C. Flageul[1], E. Lamballiais[1], V. Duffal[2], E. Le Coupanec[2], S. Benhamadouche[2]

[1]Curiosity Team, Pprime Institute, CNRS – Univ. Poitiers – ISAE/Ensma, France
[2]EDF R&D, Fluid Mechanics, Energy and Environment, France

1. ABSTRACT

Wall-resolved Large-Eddy Simulation (WRLES) is carried-out to investigate the turbulent mixed convection in the upward flow within tube bundles, a configuration representative of a real scenario in a nuclear reactor. A database of first and second-order turbulence statistics is set-up. Interesting physical phenomena triggered by the buoyancy forces in the spatially developing flow are also highlighted.

2. INTRODUCTION

In an accident scenario with shutdown of the primary pumps, the flow in the cooling system of a Pressurised Water Reactor (PWR), typically a tube bundle geometry, is generally characterised by small flow rates with the presence of strong cross flows as well as other non-trivial physical phenomena generated by buoyancy forces and their interaction with the underlying forced convection. Correctly describing these phenomena is naturally challenging for industrial codes, which thus need to be validated for handling these configurations. Recent and significant developments in computer science now make it possible to envisage a high-accuracy simulation approach for some of these flow configurations, providing full access to the flow variables and a more detailed understanding of its physics, for which a lack of reference data is still clearly identified in the literature. This work concerns the high-resolution simulation of the spatially developing flow within tube bundles in such an accidental scenario, for which forced and natural convection effects are comparable (mixed convection). The output data is intended to be used primarily in the validation of EDF R&D's thermal-hydraulics code defined to the sub-channel scale (THYC), while also providing reference data for the literature on a realistic configuration, representative of the real challenges and problems of industrial nuclear applications, close to the configuration studied experimentally by [1].

3. METHODOLOGY

The code Xcompact3d [2] is used to perform wall-resolved Large-Eddy Simulation (LES) of the spatially developing and upward flow within a tube bundle, unsymmetrically heated by the wall heat fluxes $q_{w1} \approx 50\ kW/m^2$ (cold side) and $q_{w2} \approx 100\ kW/m^2$ (hot side), as schematized in Figure 1. Note that this work follows a previous validation of the code Xcompact3d for the simulation of turbulent mixed convection by considering a vertical pipe configuration [3]. The incompressible Navier-Stokes and energy equations are solved in dimensional form with gravitational effects being represented by the buoyancy term G (modelled here using the Boussinesq approximation)

*Corresponding Author: rodrigo.vicente.cruz@univ-poitiers.fr

$$\frac{\partial u_i}{\partial t}+\frac{1}{2}\left(u_j\frac{\partial u_i}{\partial x_j}+\frac{\partial u_i u_j}{\partial x_j}\right)=-\frac{\partial p}{\partial x_i}+\nu\frac{\partial^2 u_i}{\partial x_j x_j}+\delta_{i3}G \quad (1)$$

$$\frac{\partial T}{\partial t}+u_j\frac{\partial T}{\partial x_j}=\alpha\frac{\partial^2 T}{\partial x_j x_j} \quad (2)$$

where the buoyancy term is given by $G=g\beta\left(T-T_{inlet}\right)$, with g the gravity acceleration, β the volumetric thermal expansion coefficient and T_{inlet} the inlet temperature. The inlet velocity conditions are generated by the recycling technique of [4] applied over the length L_r=6D_h (where D_h is the hydraulic diameter), together with an isothermal wall applied in the section $z < L_r$. Preliminary tests have shown that such a length is sufficient to ensure fully developed turbulent inlet conditions. In order to avoid pollution of the inlet conditions by possible thermal effects propagating downwards by diffusion, the heated section of the computational domain starts only at z = 10D_h and has a length of L_h=100D_h, resulting in a total domain length of L_h=110D_h. At the outlet, convection boundary conditions are prescribed for both the velocity and temperature fields.

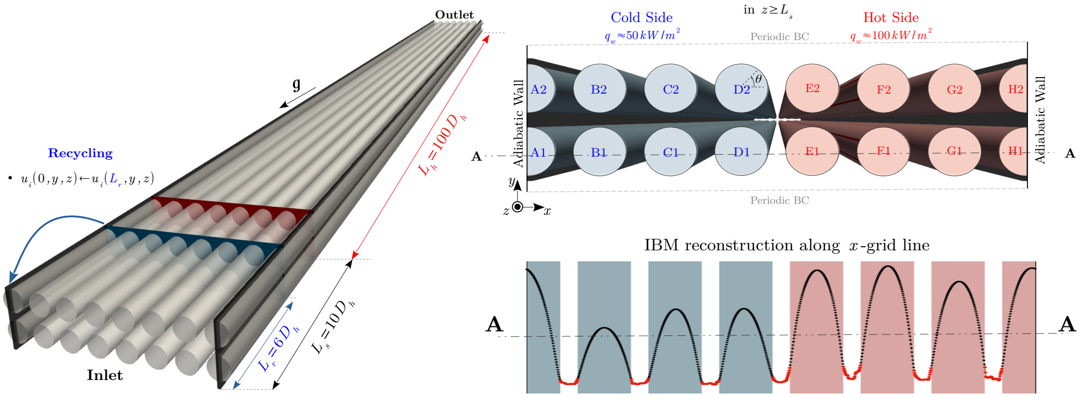

Figure 1: Schematic views of the computational configuration with L_r, L_s and L_h the recycling, safety and heated lengths respectively. The IBM is based on reconstructions of the solution within the solid bodies.

The computational domain of dimensions $L_x \times L_y \times L_z$ is discretized with a regular Cartesian mesh of $n_x \times n_y \times n_z$ = 1080 × 300 × 3751 nodes regularly distributed. The tube bundle geometry is represented by a customised immersed boundary method (IBM) [5], designed to provide accuracy in the near-wall region, where velocity and temperature boundary conditions are ensured by reconstructions of the physical fluid solution into the solid zones (A_i, B_i, C_i, D_i, E_i, F_i, G_i, H_i), the principle is illustrated in Figure 1 . The working fluid has a Prandtl number slightly above unity and the inlet Reynolds number Re_0 is around 7500. A wall-resolved LES database of first and second-order one-point statistics is set-up.

4. RESULTS

Figure 2 provides some preliminary results of the present study. The predictions of the mean viscous friction at the tube surfaces show that, within the entry region $z \lesssim 20Dh$, the designed mesh ensures the DNS accuracy for both cold and hot sides, based on observations from previous investigations [3]. Upstream however $z \gtrsim 20Dh$, due to the acceleration/deceleration experienced in the cold/hot sides respectively and the laminarisation/turbulence redevelopment effects triggered by the spatially growing buoyancy force [3], the present mesh resolution falls rather into the wall-resolved LES category and thus, we shall adopt this terminology to address the present database. Also in Figure 2, a visualisation of a typical output data used for validation of EDF's THYC code is presented. As the spatial resolution of the latter is defined such that the finest mesh corresponds to a sub-channel in the tube bundle, present velocity/temperature fields are averaged within the corresponding subdomains. These are only preliminary results and additional calculations are underway. The wealth of information provided by this data still needs to be exploited in more detail, including, for example, the change in the turbulence dynamics across the spatial laminarisation/redevelopment processes which manifest differently on the hot and cold sides, the characteristics of the thermal mixing layer, the presence of transverse flows and low frequencies, etc. Such analyses will be presented in greater detail at the conference.

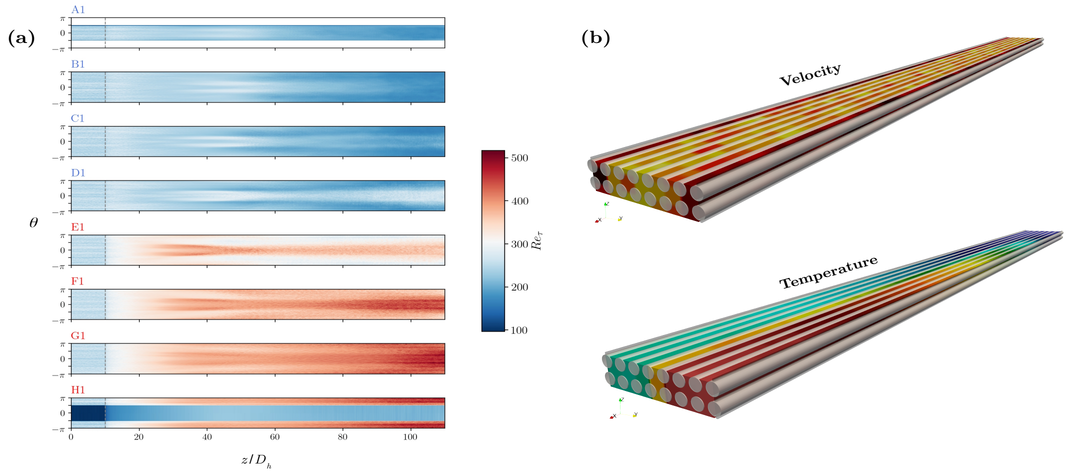

Figure 2: (a) Prediction of the friction Reynolds number computed at the tube surfaces (frame of reference as in Figure 1). (b) Visualization of velocity and temperature averaged within the sub-channel.

5. CONCLUSIONS

This study presents preliminary results of the highly-accurate numerical simulation of the turbulent mixed convection within tube bundles. The present numerical strategy associates massive parallelism, high-order schemes and a customised IBM to produce a database of first and second-order statistics. Further and more detailed results shall be presented at the conference.

ACKNOWLEDGMENT

This work was granted access to the HPC resources of TGCC/CINES under the allocations A0132A07624 / A0152A07624 made by GENCI.

REFERENCES

[1] Bates, J. M., & Khan, E. U. (1980). *Investigation of combined free and forced convection in a 2 x 6 rod bundle during controlled flow transients* (PNL-3135, 7042813; p. PNL-3135, 7042813).

[2] Bartholomew, P., Deskos, G., Frantz, R. A. S., Schuch, F. N., Lamballais, E., & Laizet, S. (2020). Xcompact3D: An open-source framework for solving turbulence problems on a Cartesian mesh. SoftwareX, 12, 100550.

[3] Vicente Cruz, R., Flageul, C., Lamballais, E., Duffal, V., Le Coupanec, E., & Benhamadouche, S. (2023). High-Fidelity Simulation of Turbulent Mixed Convection in Pipe Flow. Proceeding of 10th International Symposium on Turbulence, Heat and Mass Transfer, THMT'23.

[4] Lamballais, E. (2014). Direct numerical simulation of a turbulent flow in a rotating channel with a sudden expansion. *Journal of Fluid Mechanics*, *745*, 92–131.

[5] Vicente Cruz, R., & Lamballais, E. (2023). A versatile immersed boundary method for high-fidelity simulation of Conjugate Heat Transfer. *Journal of Computational Physics*, *488*, 112182.