SCOPE | SAUDI INTERNATIONAL CONFERENCE ON NUCLEAR POWER ENGINEERING

Proceedings of SCOPE
13-15 Nov.
2023 – KFUPM
Paper 23174

Bare Rod Bundles Fuel Assembly Coolant Flow Analysis through a Hybrid-Based CFD Numerical Simulation

Meri, Yazan[1*], Alhamdi, Abdulwahab[1], Alotaibi, Mishari[1], Alqhtani, Saad[1], Shams, Afaque[1] and Kwiatkowski, Tomasz[2]

[1] King Fahd University of Petroleum & Minerals (KFUPM), Saudi Arabia
[2] National Centre for Nuclear Research (NCBJ), Poland
[3]Interdisciplinary Research Center for Renewable Energy and Power Systems (IRC-REPS), KFUPM, Saudi Arabia

*Corresponding author *email: yazanmeri@hotmail.com*

Abstract –The heat produced in the nuclear fuel rod is dissipated by the coolant running through the channels in the fuel assembly. The flow between fuel assembly rods shows oscillating behaviour, having a noticeable effect on the cooling process. Additionally, the flow effects extend to the fuel assembly causing vibration in its structural system. The design and reliable operation of nuclear systems depend heavily on a comprehensive understanding of flow and temperature in a fuel assembly. In aiming to enhance the nuclear reactor's efficiency, safety and stability, a thorough understanding of fuel assembly coolant is crucial. Therefore, this study analyses the flow between bare rod bundle fuel assembly configuration utilizing advanced computational fluid dynamics (CFD) approaches. In this regard, a hybrid (LES/RANS) turbulence modelling approach has been adopted to study a square lattice bare rod bundle configuration. By minimizing the overall computational cost, the best aspects of Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) are employed. The obtained results are thoroughly compared with the available reference Direct Numerical Simulation (DNS) database of a closely spaced bare rod bundle based on the well-known Hooper experiment. The hybrid methodology is evaluated through a qualitative comparison of the velocity field with the DNS database. Additionally, the prediction of the flow pulsation is analysed numerically. The findings in this work justify the usage of hybrid (RANS/LES) for these types of complex flow configurations and show its reliability.

Keywords: Numerical simulation, Flow analysis, Hybrid simulation, Nuclear reactors, Fuel assemblies

I. INTRODUCTION

Flow inside nuclear reactor cores require special attention to enhance the reactor's efficiency and assure its safety. The core of nuclear reactors is a crucial location to be studied where the nuclear fuel dissipates heat to the cooling fluid. Most fuel element layouts used in nuclear reactors started with rod bundles as their basic configuration. These rod bundles are primarily distinguished by their geometric configurations; for instance, they may have square or triangular distribution. One of the primary design elements of the fuel rod assemblies is rod spacing. The pitch to rod diameter ratio (P/D) primarily serves to specify the distance between the rods, which has a significant effect on the flow.

The coolant flowing through the fuel assembly channels dissipates the heat generated by nuclear fission. Under typical operating conditions of a nuclear reactor, the temperature distribution through the fuel assemblies should, ideally, stay uniform. However, in practice, this does not occur as a result of inter-subchannel mixing. To analyze the situation and predict the lifetime of the fuel rod in terms of structural integrity and mechanical behavior, a solid comprehension of the pulsation-induced vibrations is required. Moreover, localized flow effects like hot spots, thermo-mechanical loads, partial blockage effects, and structure deformation are frequently connected to the flow of the coolant fluid within these sub-channels. The design of reliable nuclear system operations has always depended on the understanding of flow and temperature distribution.

The bare rod bundle configuration's unstable axial flow pulsations and structures have been studied both experimentally and numerically over the past 50 years and are still a subject of research today. Hooper discovered that the P/D ratio has a significant impact on the turbulent flow structure [1]. Hooper and Rehme demonstrated that for a turbulent flow through parallel rod bundles, the azimuthal and axial turbulence intensities in the rod gap region are strongly increased with the rod spacing decreasing [2]. Additionally, they discovered that the mean secondary flow has little effect on the flow parameter.

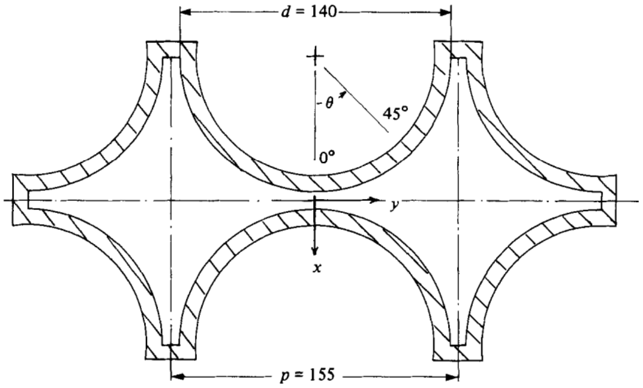

Figure 1. The geometry of a tight lattice rod bundle used in Hooper's hydraulic experiment [2].

A. The Hooper Case

The case in consideration is called the Hooper case. It refers to the specifications of the flow arrangement and Hooper's (1984, 1980) selected hydraulic experiment setup [1]-[3]. The followed experiment was performed in [3], where the setup of the rod bundle in consideration consists of a squared tight lattice configuration that contains a unit of six rods. The pitch (P) amid each rod is taken as a value of 15.5 cm and the radius of each rod (R) is taken as 7 cm. These configurations make a close-spaced rod bundle since the (P/D) ratio is equal to 1.107 (see Figure 1). The length of the unit being assessed is taken to be 9.14 m which corresponds to a factor of 128 hydraulic diameters.

B. Computational Fluid Dynamics (CFD)

CFD is a methodology to analyze fluid dynamics using numerical solution methods. Fluid dynamics is concerned with the physical principles of flows in the form of partial differential equations. CFD solvers convert these principles into algebraic equations and effectively solve these equations numerically. CFD's studies develop fluid-related design processes, making them less expensive and faster than conventional testing. Furthermore, in real-life tests, only a limited number of values are assessed at a time, but in a CFD study, all necessary quantities are measured at once and with excellent time and spatial resolution. Because CFD analyses approximate a real physical solution, it is crucial to emphasize that these CFD analyses cannot entirely exclude physical testing procedures. Experiments should still be undertaken for verification purposes.

The CFD methodology used in the present study is called hybrid LES-RANS. The hybrid LES-RANS method is an approach to solve fluid dynamics problems with increased accuracy and reduced cost. It combines Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) methods at regions where they best behave. The hybrid method had been initiated to solve several flow types that appear in the same region. To best understand this methodology, a comprehension of the principles, abilities and limitations of the LES and RANS methods should be developed.

LES is a simulation approach that solves large-scale eddies and models small ones. It shows a high accuracy compared to RANS and an incomparably lower computational cost to the Direct Numerical Simulations (DNS) in separated flows. However, it has a limitation in near-wall boundary regions where the eddies scale becomes smaller. It is a computational limitation as small eddies require finer meshing, which requires massive computational resources. Alternatively, RANS is constructed based on solving the Reynolds decomposition, which decomposes the flow variable into mean and fluctuating quantities. The term Reynolds Stress Tensor arises upon the application of the decomposition on the Navier-Stokes equation. The number of equations never suffice to solve as more unknown terms arise. This issue is known as the closure problem, and it is solved (closed) by a modelling methodology. The level of this modelling is classified based on the number of differential equations included. The 2-equation model has proven its efficiency in solving near-wall attached flows.

So, the hybrid method uses RANS to solve the boundary region and LES to solve the separated flow region. In between these regions, a region called the grey area exists [4]. The grey area poses a challenge to continually solve the flow. It is either solved or reduced by the different hybrid methods which are classified into zonal and non-zonal approaches. In zonal methods, flow is solved in separated regions, however, the non-zonal method solutions assure gradual transfer between RANS and LES.

C. Case description

There is several of research that has been done on the Hooper case such as [4]–[6]. However, this research seeks to provide a simulation for the flow between rods in nuclear reactors utilizing the advantages of an accurate and affordable method. The hybrid (LES/RANS) turbulence modelling approach has been adopted to study flow behaviour in a square lattice bare rod bundle configuration. By minimizing the overall computational cost, the best aspects of LES and RANS are employed. The obtained results are thoroughly compared with the available reference Direct Numerical Simulation (DNS) database of a closely spaced bare rod bundle, which is based on the well-known Hooper experiment.

II. Numerical Methodology

A. Flow Configuration

The computational domain is made up of two subchannels in the squarely packed rod bundle, as depicted in Figure 1. Table I shows the dimensions that are used for the geometry. This configuration makes a closely spaced rod bundle since the (P/D) ratio is equal to 1.107. The periodic length of the unit being assessed is taken to be 2.285 m as per [6], where a comprehensive study of the geometry length has been performed.

Table I. Geometry dimensions

Symbol	Definition	Value
R	Rod radius	7 cm
P	Rod pitch	15.5 cm
D_h	Hydraulic diameter	0.0714 cm

B. Boundary Conditions

The boundary conditions are taken as the same boundary conditions for the numerical Hooper case in [7],[8]. The inlet mass flow rate imposed corresponds to Re = 22 600. Periodic boundary condition was set in the streamwise direction. Also, a constant heat flux of 0.1 W/m2 has been imposed on rods.

C. Mesh Generation

The mesh for the given computational domain is generated using the ANSYS Meshing software [9]. Figure 2 shows a mesh cross-section demonstrating a completely hexahedral mesh throughout the flow domain, which gives excellent cell quality for the bulk solution. A mesh with stretched layer structure is employed in the near-wall region, which is critical for capturing the near-wall gradients. In this case, a two-step method is employed to generate the mesh, namely:

Step 1: 2D (in x-y plane) mesh for the cross-section is generated.

Step 2: This 2D mesh is then uniformly extruded in the streamwise direction allowing for a high-quality mesh.

The overall mesh consists of 38 million computational cells with non-dimensional sizes Δy+ (wall-normal direction) = 0.0005 (near the wall), and

0.001 (in the bulk). The boundary mesh is structured of 20 layers with a stretching ratio of 1.15 in the near-wall region. The first layer cell size was computed to maintain the average y+ value below 1 (y+ denotes the normalized distance from the nearest wall in wall units). The mesh quality is checked through its orthogonality, skewness and aspect ratio.

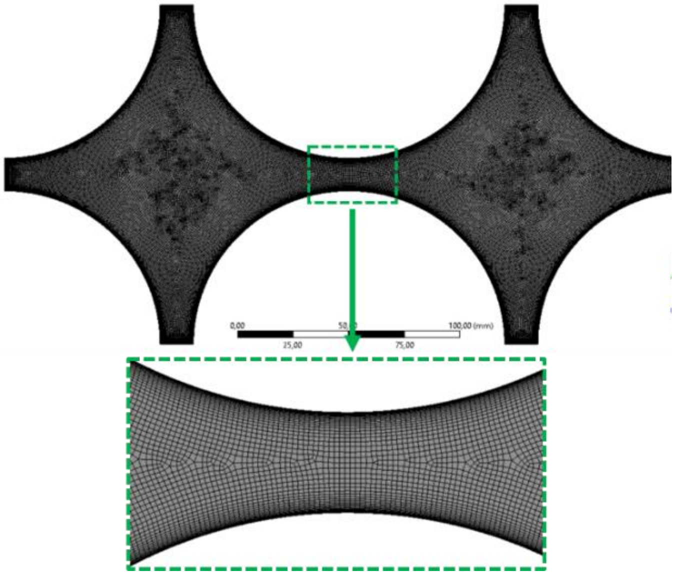

Figure 2 2D (in the x-y plane) mesh for the cross-section. The zoom mesh generated in the region between two adjected rod bundles is presented.

D. Turbulence Modelling

The commercially available Ansys Fluent Version-2022R1 [10] software was used to carry out all the numerical simulations discussed in this paper. ANSYS Fluent is the industry-leading fluid simulation software known for its advanced physics modelling capabilities and industry-leading accuracy. It provides a cutting-edge, approachable user interface that simplifies the CFD process from pre- through post-processing inside a single window workflow.

The set turbulence modelling for this project is the developed model of the hybrid approach detached eddy simulation (DES). Unsteady RANS and LES are combined in a DES mixture in an effort to solve near-wall regions utilizing the RANS methodology and the rest of the flow with the LES. The improved version of the delayed detached eddy simulation (IDDES), which is the most recent DES formulation, is chosen for the current flow configuration.

E. Delayed Detached Eddy Simulation.

The sensitivity of older DES models to Grid Induced Separation (GIS), where well-intentioned grid refining techniques may actually lower the quality of an LES simulation and yield findings less accurate than conventional RANS simulations on coarser grids, was one of its weaknesses [11]. Instead of the more common cube root of the grid volume, the maximum three-dimensional grid spacing is employed as the grid length scale in this approach, which avoid the issue discussed. according to Spalart, extending the RANS region by detecting boundary layers, as opposed to the LES/RANS switching over-simply being a function of the wall distance or grid size alone, is one of the ways to lessen the GIS impact in the Delayed DES (DDES) [11]. According to Fröhlich and von Terzi, the resulting adjusted length scale is represented by the formula below[12].

$$\tilde{d} = d - f_d \ \max\ (0; d - \ C_{DES}\, \Delta)$$

where Δ is the sub-grid filter, d is the RANS length scale, and CDES (= 0.65) is the DES constant. The function *fd* is intended to identify and postpone the onset of LES near attached boundary layers where RANS modelling is preferable.

F. Improved Delayed Detached Eddy Simulation.

A further variation of the DDES, the improved delayed detached eddy simulation (IDDES) of Shur et al. (2008) combines the DDES with wall modelling LES (WMLES) and blends the applied RANS and LES length scales with blending functions. According to Shur, a separate and essential element of IDDES is a new definition of the sub grid length-scale, which solely considers the grid spacing and explicitly accounts for wall-distance dependence [12]. The sub grid scale is described as follows:

$$\Delta = \min\{\max[C_w y, C_w \Delta_{max}, \Delta_{wn}]\, , \Delta_{max}\}$$

where, y is the wall distance, Cw is a constant, Δmax is the maximum local grid-spacing and Δwn is the normal direction grid step in the wall. The near wall region of this sub-grid filter has a smaller Δ than that of traditional sub-grid filters. Δ equals the greatest local grid spacing as viewed from a far distance from the wall. As a result, the tension close to the wall is reduced while the stress further from the wall is increased. As was already indicated, IDDES has two branches: DDES and WMLES. Only when there is no turbulent content in the inflow conditions does the DDES branch of the model become active. The WMLES branch, on the other hand, is only meant to be operational when the inflow conditions are unstable and impose some turbulent content. when combining the WMLES and DDES branches Now, the length scale for IDDES may be calculated as:

$$\tilde{d}_{hyb} = \tilde{f}_d(1+f_e)I_{RANS} + (1-\tilde{f}_d)I_{LES}$$

The blending function, *fd*, is defined as *fd* = max((1-fdt), fB), is a shielding. For details on how the two branches were added to produce the above equation

G. Flow Parameters

The rod has a 14 cm diameter (D) and a 15.5 cm pitch (P) between the two rods. As a result, the example under consideration has a pitch-to-diameter ratio (P/D) of 1.107, defining it as a close-spaced rod bundle. The test section's streamwise length is 2.285 m. The chosen configuration has a bulk Reynolds number of 22600 as done in [7].

Table II. Summary of the simulation flow parameters

Symbol	**Definition**	**Value**
V_b	Bulk velocity	1 m/s
q	Heat flux on the rods	0.1 W/m^2
T	Temperature	100 C

III. Results and discussion

The results obtained from the simulation using the Hybrid IDDES method are presented in this section. To ensure the results are accurate, they are compared with corresponding DNS results done in [7], [8]. The validation of the results is done in two aspects: the first one is a qualitative aspect where the results are compared visually, and the second one is a quantitative aspect where the results are compared in terms of certain parameters and exact numerical results obtained from the simulation.

A. Qualitative Comparison

The data collected for the qualitative comparison are the cross-section of the rod bundle and the axial streamwise behaviour. In the presented paper, the instantaneous velocity magnitude is selected to observe the qualitative prediction of the IDDES hybrid model.

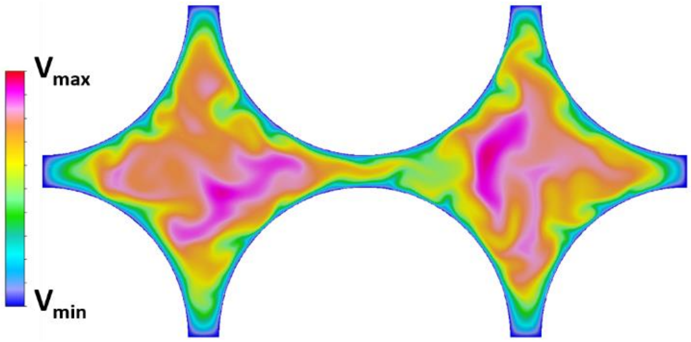

Figure 3a. XY-plane velocity field capture of hybrid IDDES simulation.

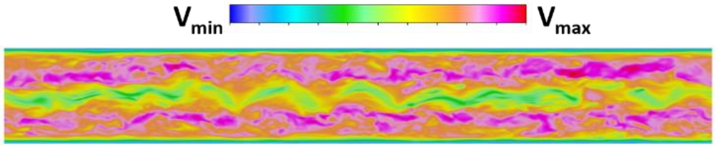

Figure 3b YZ-plane velocity field capture of hybrid IDDES simulation.

B. Velocity Field Qualitative Comparison: Mean and RMS

The instantaneous velocity field is shown in Figure 3. Figure 3a shows the cross-section flow behaviour of the domain where Figure 3b shows the axial flow behaviour. Comparing the contours of the DNS instantaneous velocity obtained by [8] in Figure 4, the turbulent eddies are well captured in both the main and secondary flow. The hybrid method predicts the velocity profile and captures both the turbulence and the pulsation shown in the results in Figure 4. It produced multiscale vortexes in the secondary flow zone that reached small sizes, as appeared in Figure 3a. Alongside, a sinusoidal pattern can be observed on the flow field specifically when considering the streamwise capture shown in Figure 3b. According to the findings in [14], this is an expected characteristic for bare rod bundles with a low P/D ratio. The hybrid simulation accurately represented the flow characteristics and captured fine details as compared to the DNS results in [8].

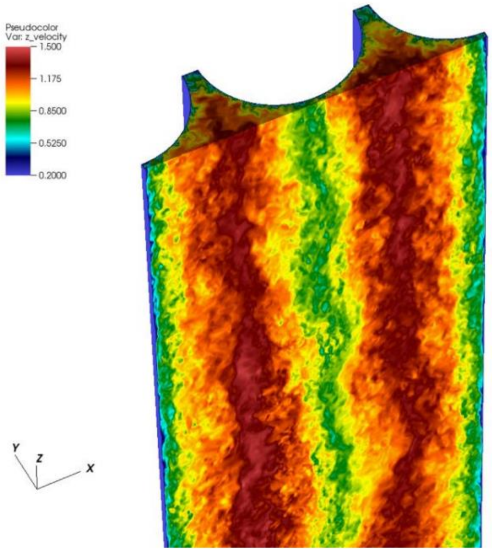

Figure 4. Instantaneous velocity field slice visualization of the DNS simulation [8].

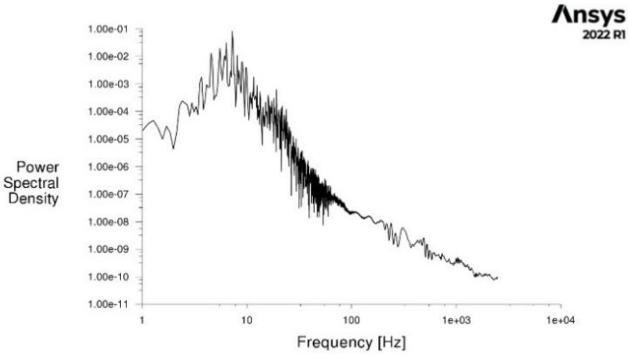

Figure 5. PSD plot for hybrid simulation

C. Power spectral density comparison

The power spectral density (PSD) plot against the frequency of mass flow rate oscillations for the hybrid simulation is shown in Figure 5. It is helpful to analyze the models capabilities predicting the flow pulsation. In the considered case, PSD graph shows a similar behaviour compared to the DNS findings in Figure 6 done by [7]. This implies that the numerical methodology is able to predict the overall flow pulsation in the fuel assemblies flow.

IV. Conclusion

Understanding the flow through the fuel assembly properly is essential considering the reliability of the nuclear reactor. So, in this study, an effort has been made to examine these aspects using CFD methods. In this context, a square lattice bare rod bundle configuration has been studied using a hybrid turbulence modelling approach. The best features of RANS and LES are used to reduce the overall computing cost. The acquired results are carefully compared with the reference DNS database that is currently available [6] and are based on the well-known Hooper experiment for a bundle of tightly packed bar rods. The chosen configuration has a (P/D) of 1.107 and a bulk (Re) of 22 600. It is found that the hybrid simulation was able to predict the flow behaviour. Turbulence eddies were captured at small scale with good accuracy compared to the DNS findings. Additionally, the flow pulsation showed a matching behaviour as illustrated in the PSD assessment. Therefore, the results demonstrated the ability of IDDES model to capture the general flow characteristics.

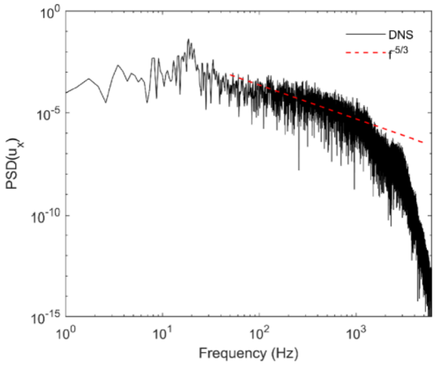

Figure 6. PSD plot for simulation done in [7].

References

1. J. D. Hooper, "Developed single phase turbulent flow through a square-pitch rod cluster," Nuclear Engineering and Design, vol. 60, no. 3, pp. 365–379, (1980).

2. J. D. Hooper and K. Rehme, "Large-scale structural effects in developed turbulent flow through closely-spaced rod arrays," J Fluid Mech, vol. 145, no. 1, p. 305, (1984).

3. J. D. Hooper and D. H. Wood, "Fully developed rod bundle flow over a large range of Reynolds number," Nuclear Engineering and Design, vol. 83, no. 1, pp. 31–46, (1984).

4. S. Panguluri, D. Reasor, and R. LeBeau, "Investigation of grey area construction on the performance of detached eddy simulation," 18th AIAA Computational Fluid Dynamics Conference, (2007).

5. T. Kwiatkowski and A. Shams, "Towards the accurate prediction of axial flow and heat transfer in a tightly spaced bare rod bundle configuration," Nuclear Engineering and Design, vol. 403, p. 112119, (2023).

6. A. Shams and T. Kwiatkowski, "Towards the Direct Numerical Simulation of a closely-spaced bare rod bundle," Ann Nucl Energy, vol. 121, pp. 146–161, (2018).

7. J. K. Lai, G. Busco, E. Merzari, and Y. A. Hassan, "Direct Numerical Simulation of the Flow in a Bare Rod Bundle at Different Prandtl Numbers," J Heat Transfer, vol. 141, no. 12, (2019).

8. J. K. Lai, E. Merzari, Y. A. Hassan, and A. Obabko, "Validation and development of DNS database for low Prandtl numbers in Rod Bundle," Volume 2: Computational Fluid Dynamics, (2019).

9. "Ansys meshing | mesh generation and analysis for FEA, CFD." [Online]. Available: https://www.ansys.com/products/ [Accessed: 14-Feb-2023].

10. "Ansys Fluent | Fluid Simulation Software." [Online]. Available: https://www.ansys.com/products/fluids/ansys-fluent. [Accessed: 20-Nov-2022].

11. P. R. Spalart, "Detached-Eddy Simulation," Annu Rev Fluid Mech, vol. 41, no. 1, pp. 181–202, (2009).

12. M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin, "A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities," Int J Heat Fluid Flow, vol. 29, no. 6, pp. 1638–1649, (2008).

13. A. Shams, F. Roelofs, E. M. J. Komen, and E. Baglietto, "Numerical simulations of a pebble bed configuration using hybrid (RANS–LES) methods," Nuclear Engineering and Design, vol. 261, pp. 201–211, (2013).

14. S. Tavoularis, "Rod bundle vortex networks, gap vortex streets, and gap instability: A nomenclature and some comments on available methodologies," Nuclear Engineering and Design, (2011).