UK Heat Transfer Conference 2021
UKHTC2021

17th UK Heat Transfer Conference (UKHTC2021)
4-6 April 2022, Manchester, UK

DEVELOPMENT AND DEMONSTRATION OF A COARSE-GRID SUB-CHANNEL CFD MODEL FOR THE COOLING OF A 5×5 ROD BUNDLE

Bo Liu[1*], Shuisheng He[1], Charles Moulinec[2], Juan C. Uribe[3]

[1]Department of Mechanical Engineering, Faculty of Engineering, The University of Sheffield, Sheffield S1 3JD, United Kingdom
[2]Sciencific Computing Department, Science and Technology Facilities Council, Daresbury Laboratory, Warrington, WA4 4AD, United Kingdom
[3]EDF Energy R&D UK Centre, Manchester, M13 9PL, United Kingdom

ABSTRACT

Sub-Channel CFD (SubChCFD) is a novel CFD-based coarse-grid sub-channel analysis tool recently developed for thermal hydraulic calculations of nuclear reactors. It combines CFD and the traditional sub-channel approaches, providing intermediate fidelity results at low computing cost with closure modelling for wall friction and heat transfer using empirical correlations. To enhance the capability of the new tool, and allow it to be used for a wider range of flow scenarios, additional modelling functionalities were developed. They include the coupling with the resolved CFD approach and the coupling with the porous media approach. Work carried out in this paper aims to demonstrate the newly developed capabilities of SubChCFD in the modelling of large nuclear fuel bundles with complex internal structures. The NESTOR experiment based on a 5-metre 5×5 rod bundle test facility was selected for the simulations. A coarse-grid baseline model was created to cover the entire heated length with embedded porous media sub-models to describe the spacer girds and account for their effects. A resolved CFD sub-model was created to cover the target span and was coupled with the baseline model, in order to capture the main features of the flow. Through this test, SubChCFD has been demonstrated not only to provide reliable predictions for large-scale flow features but also to produce comparable results to those of conventional CFD for regions of interest with a greatly reduced computational cost.

1. INTRODUCTION

In our recent work [1], a novel coarse-grid Computational Fluid Dynamics (CFD) approach was developed, based on a hybrid technique combining the features of the traditional sub-channel analysis tools and modern CFD. The new method is referred to as Sub-channel CFD (SubChCFD). It is such that the main geometry of the sub-channel structures is resolved using a very coarse mesh, whilst the wall effects are accounted for using well-validated industry-standard correlations usually used in traditional sub-channel analysis codes. This ensures the method to provide results similar to the well-calibrated sub-channel results for straight fuel channels at nominal operating conditions. Later, new functionalities that allow SubChCFD to be coupled with resolved CFD [2] and the porous medium approaches [3] were developed. This greatly enhanced the modelling capability of SubChCFD for it to be used for a wider range of flow scenarios.

The main purpose of this paper is to present new developments and testings of SubChCFD heat transfer capabilities and also to show the applicability of this new tool to complex industrial applications. A test case is selected based on one of the New Experimental Studies of Thermal-Hydraulics of Rod Bundles (NESTOR) series of experiments [4] in which the test facility used closely resembles a real-world PWR fuel bundle and the pressure and temperature cover the normal PWR operating conditions. The experiments provide a variety of high-fidelity measured data, including axial mean velocity distribution, pressure drop and rod wall surface temperature, used for validation. In addition, the EPRI's CFD Round Robin benchmark exercise [4] based on the NESTOR experiments have attracted participants from various organisations who have provided abundant CFD simulation

*Corresponding Author: bo.liu@sheffield.ac.uk

data that can also be used for cross-code comparison with the current simulation to further evaluate the strengths and drawbacks of the new method against conventional CFD tools.

2. OUTLINE OF THE SUBCHCFD METHODOLOGY

A dual mesh system is used, including, namely, (i) a sub-channel filtering mesh which aligns with the mesh used in typical sub-channel codes, enabling the sub-channel-level wall friction and heat transfer calculated using existing engineering correlations, and (ii) a coarse-grid computing mesh, on which a full set of 3-D RANS (Reynolds-averaged Navier-Stokes) governing equations are solved with a near-wall closure method based on calculations of step (i). Details of the development, implementation and validation of the baseline SubChCFD were reported in Liu et al.[1].

SubChCFD allows resolved CFD sub-models to be nested into the regions of interest but cannot be suitably handled using a coarse mesh, to locally improve prediction. Such a coupling is achieved using a domain-overlapping approach in which the resolved fine-mesh model receives data from the coarse-mesh solution to define its boundary conditions for the coupling interfaces. A Dirichlet-type boundary condition is used for velocity, and accordingly, a homogeneous Neumann condition is used for pressure. Feedback is also allowed from the resolved fine-mesh model to the coarse-mesh SubChCFD model to locally improve the predictions of the latter. This two-way coupling is achieved by adding an extra momentum source term to the coarse-mesh equation as a penalty to force the velocity solution to approach that of the resolved fine-mesh model. A more detailed description of the coupling between SubChCFD and resolved CFD sub-models can be found in Liu et al.[2].

It is also straightforward to couple SubChCFD with the porous medium approach so as to deal with any sub-scale fine structures that are difficult to be resolved using a coarse mesh. In such an approach, a computational domain with an explicit representation of the main sub-channel geometry is still used. Flow regions with sub-scale fine structures are considered as porous media. A spatial averaging operator is then applied to the RANS governing equations over the porous media regions to obtain the macroscopic equations. A momentum source term that appears due to spatial averaging accounts for the resistance forces exerted by the sub-scale fine structures on the fluid. A closure modelling is required to determine this term by relating it to the local macroscopic flow quantities. Details about the implementation and validation of the coupling between SubChCFD and the porous media approach can be found in Liu et al.[3].

Figure 1 shows a sketch of the SubChCFD methodology, including schematics of the baseline model and the coupling modules.

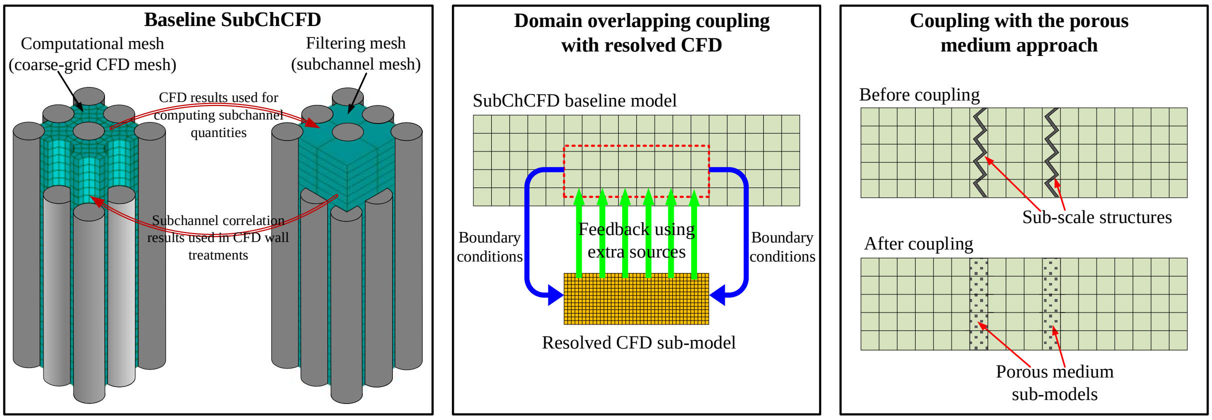

Figure 1: The SubChCFD methodology, including schematics of the baseline model (left), the domain-overlapping coupling with resolved CFD (middle), and the coupling with the embedded porous medium models (right).

3. NUMERICAL MODEL

3.1 Description of the test case

The test section of the NESTOR facility consists of a 5×5 square array of tube rods enclosed in a square bundle casing with Simple Support Grids (SSGs) distributed evenly at axial intervals of 279 mm. The rod outer diameter is 9.50 ± 0.02 mm, and the array pitch and rod-to-wall gap are 12.6 mm and 3.1 mm, respectively. The resulted width of the bundle casing is 61.1 mm. The thickness of the tube is different for the 9 inner rods (0.9 mm) and the 16 peripheral rods (0.675 mm). The total length of the rod bundle is approximately 5 metres, including of a heated length of 3.658 metres. The experiment for the selected test case was performed at a high pressure of 15.6 MPa with an inflow mass flux of 4540 kg/m2/s. Water entered the test section at a temperature of 250.5 °C and was heated up gradually by the tube rods with heating powers of 94.812 kW/rod and 73.902 kW/rod for the 9 inner rods and the 16 peripheral rods, respectively.

3.2 Coupled model system

Figure 2 (top) shows the whole configuration. A coarse-grid SubChCFD baseline model (in green) is created to cover the entire heated length of the rod bundle. The SSGs are modelled using embedded porous medium models (in grey) with a volume porosity of approximately 0.9 across the porous medium sub-domains. To refine the simulation for the target span where the experimental measurements are intensively performed, a resolved sub-model (in red) is created resolving both the rod bundle and the spacer. It is coupled with the coarse-grid baseline model. In the resolved sub-model, heat conduction within the heater tubes is also taken into account to improve prediction.

Meshes used in the coupled model system are first generated in 2-D and then extruded along the axial direction. Figure 2 (bottom) shows the 2-D meshes at some representative locations of the rod bundle. It can be seen that a coarse-grid mesh is used for the SubChCFD baseline model (the mesh does not change across the porous media sub-models where the blockage effect of the spacers are accounted for implicitly using porosities). The coarse-grid mesh consists of 1.4 million cells. The mesh used for the resolved sub-model is block-structured and contains only hexahedral cells. The solid walls of the 9 inner and16 peripheral rods are meshed using 5 and 4 layers of cells in the radial direction, respectively. The circumferential divisions in the solid mesh are consistent with that in the adjacent fluid mesh to ensure conformal interfaces between the two. For simplicity, heat conduction inside the SSG (not expected to have significant effects) is not considered, and the computational domain is not tailored for accounting for that. The total mesh size (including both fluid and solid meshes) is 34.4 million cells (28.9 million cells for fluid and 5.5 million cells for solid).

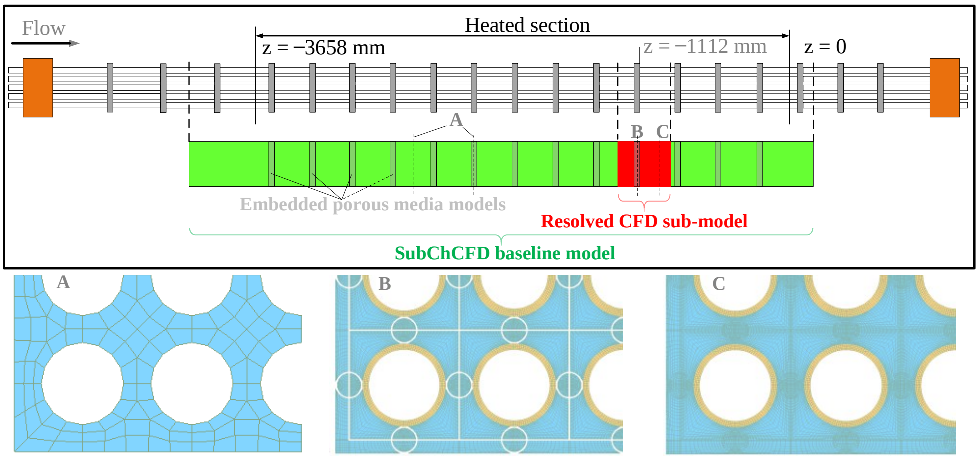

Figure 2: Spatial arrangement of the computational domains in the coupling system (top), and some cross-sectional views of the meshes used (bottom).

4. RESULTS AND DISCUSSION

Before going into a detailed discussion of the simulation results, an overall picture of the flow predicted by the coupled model system is first shown in Figure 3. The left of the figure shows that the resolved sub-model (in red) occupies a small part of the entire geometry of the SubChCFD model system (in green), where the locations of the SSG spacers are also indicated. A magnification of the region of most interest in the model system (i.e. the section enclosed in the dashed box) is shown in the rest of the figure, where some details of the predicted rod wall temperature and axial velocity distributions at some representative elevations are also shown.

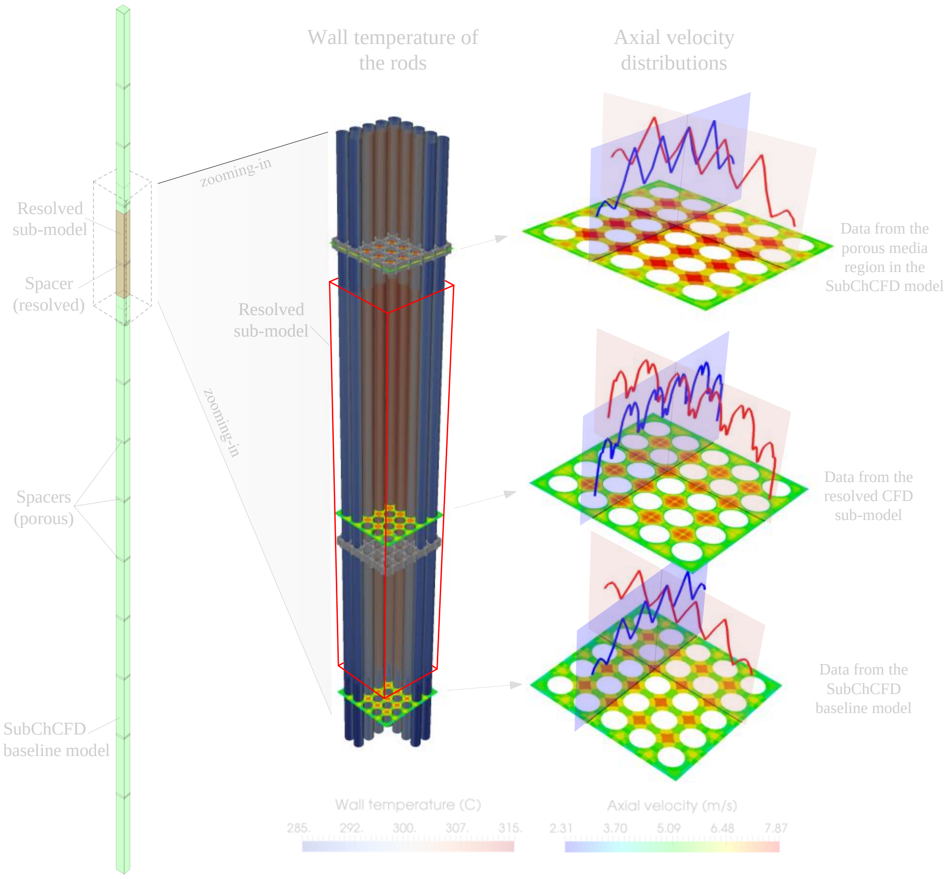

Figure 3: Overall flow pattern predicted by the coupled model system

4.1 Flow field

The SSG size is small and its design is simple. However, its effects on the flow are expected to be significant. The axial velocity profile is expected to be highly distorted, where the flow passes through the SSGs. This phenomenon cannot be captured by a SubChCFD simulation by definition. One of the aims of coupling SubChCFD with a resolved sub-model is to capture some details of the grid effects on the flow downstream. Since the velocity measurements were not performed for the simulated test case, measured data from an isothermal experiment performed in the same rig (i.e. the MANIVEL experiment in [4]) was used to assess the simulation. Figure 4 shows the axial velocity profiles normalised by cross-sectional averaged values at several axial locations downstream of the SSG. The distortion of the axial velocity profile caused by the SSG is captured as expected by the resolved sub-model in the coupled simulation, although the peak values in the edge sub-channel are under-predicted by around 10%. This may be due to either the turbulence model used or the discrepancy of flow conditions between the thermal case simulated and the isothermal case used as experimental data, or both combined.

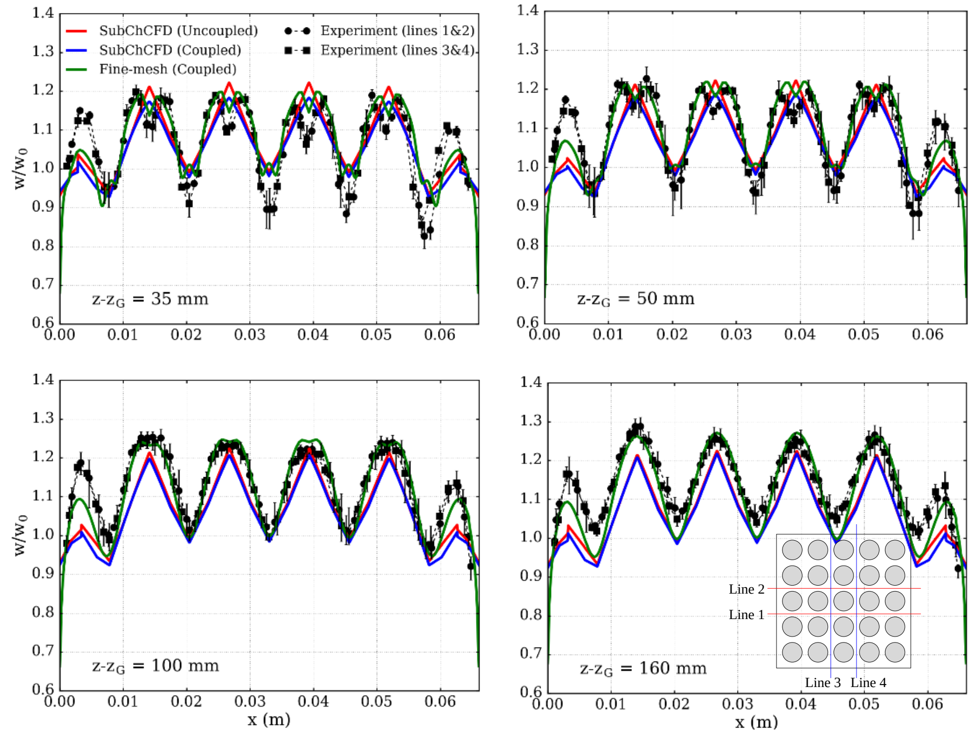

Figure 4: Mean axial velocity profiles along the centre line of the sub-channels located between the second and the third rank (Line 2) of the rods at axial locations of 35 mm, 50 mm, 100mm and 160 mm downstream of the SSG. The experimental data are plotted separately for Lines 1&2 and 3&4 and error bars are also shown.

4.2 Thermal field

Figure 5 shows the simulation results of the circumferentially averaged rod wall outer-surface temperature over the central rod (i.e. the one in the middle of the rod bundle). Interestingly, the coarse mesh results agree better with the experimental data than the fine mesh result does in terms of temperature magnitude. They are even better than most of the full-domain resolved CFD results. However, the coarse-mesh results do not show a convex shape as most of the CFD results do in the target grid span, although such a feature is very insignificant. This result is reasonable as by definition the coarse-grid model is not expected to resolve the detailed effects of the SSG on the flow and heat transfer downstream. Conversely, the fine-mesh result of the coupled simulation agrees better with most of the reference CFD results as well as the experimental data in this regard, even though the overall temperature magnitude is over-predicted by about 3 °C.

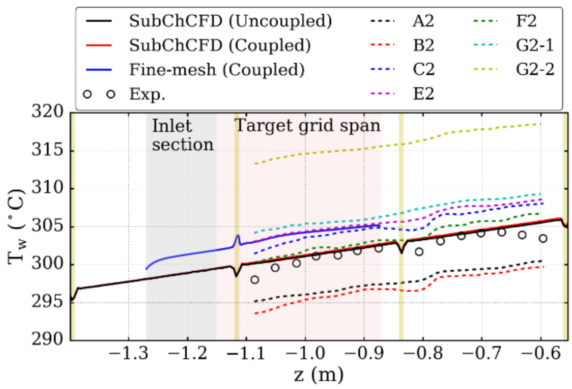

Figure 5: Axial evolution of the circumferentially averaged rod wall outer-surface temperature of the central rod in the target grid span. Resolved CFD results represented by A2, B2, …, G2-2 are provided by the participants of EPRI's Round Robin benchmark exercise [4].

Figure 6 shows the detailed circumferential distributions of the rod wall outer-surface temperature over the central rod at the four elevations where experimental measurements are available. Despite an over-prediction in magnitude, the fine-mesh results of the coupled simulation are consistent with most of the full-span CFD results (especially those produced using isotropic turbulence models) in terms of circumferential variations. A maximum of about 3 °C wall temperature variation along the circumferential direction of the rod is predicted. The rod wall outer-surface temperature predicted by the coarse-grid model are highly dependent on how the reference temperature is selected. In the current simulation, the temperature of the wall adjacent mesh cell is selected as a reference, leading to very similar predictions of the wall temperature distribution to those of resolved CFD approaches.

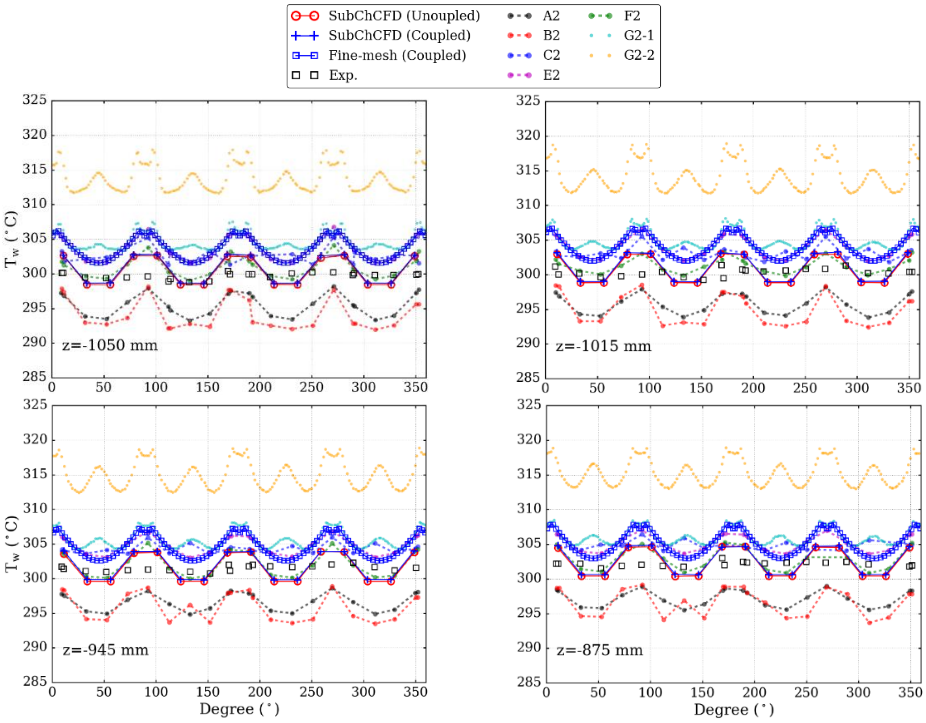

Figure 6: Circumferential distributions of the rod wall outer-surface temperature over the central rod at axial elevations of z = -1050 mm, -1015 mm, -945 mm and -875 mm, based on the coordinates shown in Figure 2.

5. CONCLUSIONS

SubChCFD has been demonstrated to show good predictability and flexibility in modelling large nuclear reactor components with complex internal structures. With the newly developed heat transfer and coupling capabilities, it is able to produce comparable predictions to those of conventional CFD approaches for regions of interest, while requires much less computing resources.
The authors would like to acknowledge the support received through BEIS's Digital Reactor Design research program (TRN 1659/10/2018) and EPSRC's CCP for Nuclear Thermal Hydraulics (No. EP/T026685/1).

REFERENCES

[1] B. Liu, S. He, C. Moulinec & J. Uribe, Sub-channel CFD for nuclear fuel bundles. *Nucl. Eng. Des.*, **355** (2019) 110318.

[2] B. Liu, S. He, C. Moulinec & J. Uribe, A Coupling Approach between Resolved and Coarse-grid Sub-channel CFD. *Nucl. Eng. Des.*, **377** (2021) 111124.

[3] B. Liu, S. He, C. Moulinec & J. Uribe, Coupled Porous Media Approaches in Sub-channel CFD. *Nucl. Eng. Des.*, **377** (2021) 111159.

[4] EPRI, Computational Fluid Dynamics Benchmark of High Fidelity Rod Bundle Experiments Industry Round Robin Phase 1 - Rod bundle with simple supporting grids. *Technical Report*, Palo Alto, CA, (2014).