UK Heat Transfer Conference 2021
UKHTC2021

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

HEAT TRANSFER ISSUES IN CURRENT AND NEXT GENERATION NUCLEAR REACTORS

Shuisheng He[1,*]

[1]Department of Mechanical Engineering, University of Sheffield, Sheffield M1 3JD, United Kingdom

ABSTRACT

Nuclear energy is recognised to play an important role in mitigating climate change and globe worming in the next few decades. The focus of the nuclear R&D internationally is the development of the advanced nuclear reactors (known as Generation IV reactors) and small modular reactors (SMRs), both of which will provide enhanced safety and reliability as well as commercial competitiveness. Improved thermal hydraulics understanding and analysis tools are required for such new developments. In this paper, we discuss some thermal hydraulics modelling challenges associated with a number of reactor designs while summarising recent research carried out in our research group. We first discuss modelling and simulation tool developments addressing both high-fidelity simulations and multi-scaling modelling. We then provide an overview of studies of liquid metal heat transfer relevant to sodium- and lead-cooled fast reactors, and mixed convection heat transfer encountered in passive cooling of all reactor types in general and supercritical water-cooled reactor in particular. Finally, we discuss some recent work carried out in supporting the safe operation of the Advanced Gas-cooled Reactors (AGR) unique to and currently in operation in the UK.

1 INTRODUCTION AND OVERVIEW

The booming era of the nuclear reactor development and deployment in the late 1950s through to early 1970s was put into a sudden halt due to the disastrous nuclear accidents occurred at Three Mile Island in USA in 1978 and Chernobyl in Ukraine in 1982. Since 2000, however, nuclear energy has been actively pursued as a major contributor to achieving low-carbon emissions to mitigate global warming. The Fukushima Daiichi nuclear disaster occurred in 2011 due to a tsunami following an earthquake again cast significant concerns on the safety of nuclear energy. Nevertheless, there are still significant interests worldwide in nuclear energy as part of the overall energy production portfolio contributing to the decarbonisation of the economy. The new development and deployment have now taken a relatively cautious but a mature approach.

The current R&D activities in nuclear energy include lifetime extension of existing reactors (mostly Generation II reactors), new building of nuclear reactors (Generations III or IIIa) and the design and prototyping of advanced reactors (i.e., the so-call Generation IVs). In order to reduce the initial cost and improve the safety and robustness of reactors, small modular reactor (SMR) technology has been actively researched and developed, which includes both the extended version of Gen III reactors (water-cooled reactors) and those based on the advanced Gen IV reactor concepts. The R&D of new generation reactors is often supported by governments as well as being driven by private sectors, e.g., the NEAMS development in the US and the NIP in the UK [ref]. Internationally, significant collaborations are led by both IAEA, e.g., various Collaborative Research Projects (CRPs), and Generation IV International Forum (GIF) [ref].

The effective cooling of the reactor core is key to the design and operation of nuclear reactors. To this end, the purpose of thermal hydraulics analysis is to ensure that the reactor core is sufficiently cooled and the all components are maintained within their temperature limits.

The advanced nuclear reactors need to be inherently safe and significantly more economical than

*Corresponding author: s.he@sheffield.ac.uk

the present designs, which lead to major scientific challenges. Firstly, most of the advanced designs use ‘new’ coolants to achieve the above objectives, but in comparison with water, our understanding of such fluids at reactor conditions is significantly poorer. A large amount of data would be required for reactor design but technically they are very difficult and expensive to obtain. Secondly, passive cooling is required in the advanced designs, but it is very challenging to predict because of their ‘abnormal’ and varied flow behaviours (including instability). Finally, the ’economical requirement’ necessitates the reduction in conservativeness, which has been very high in existing designs, partly due to the inability to quantify uncertainties in predictions.

Thermal hydraulic calculations for reactor design and safety cases are, traditionally and still the case, based on system or sub-channel codes initially developed in the 1960s/1970s. They are based on lumped/network or 1D approaches making use of a vast amount of separate effect test (SET) and Integral Effect Test (IET) experimental data. Each reactor design would have its codes specifically developed and validated. These approaches are reliable but often highly conservative, and are not designed for dealing with new issues/phenomena encountered. The recently developed computer technology including CFD provides an exciting opportunity, though it also faces many challenges including high computing resources required and the reliability of the models. Consequently, an active direction of development in nuclear thermal hydraulics (NTH) is to modernise the analysis tools and methodologies by effectively combining CFD with the traditional approaches.

Another challenge is associated with the use of ‘new’ media, such as liquid metal, in sodium or lead cooled fast reactors (SFR or LFR) or molten salts in MSR. The Prandtl number of liquid metals is significantly lower than that of water, whereas that of molten salts is higher. All of this invalidates the Reynolds analogy widely employed in turbulence modelling to relate turbulent momentum and heat transfer, and hence leading to inaccurate predictions for such fluids. In addition, the SFRs and LFRs employ large pools of liquid metal in which the reactor core and heat exchangers are submerged. Such designs have many merits but they are also open to potential detrimental effects due to thermal stratification, flow stagnation and thermal striping, for example, which require careful consideration.

Passive cooling is a default requirement for advanced reactor designs to achieve inherent safety in reactor operation. Such systems often include low flowrate, buoyancy driven flows and/or phase change. Improving the reliability and accuracy of predictions of such systems are subject of many recent endeavours.

The purpose of this paper is to review some recent work that has been carried out in our group at the University of Sheffield, through which we analyse a number of the challenges outlined above. We first discuss the development of the modelling and simulation tools and methodologies, including the high fidelity DNS/LES code CHAPSim and the novel coarse-grid CFD, Sub-Channel CFD, together with examples of applications (Section 2). This is followed by an overview of the research on liquid metal heat transfer encountered in SFRs/LFRs in Section 3 and mixed convection under the influences of strong variations of thermal properties and buoyancy in Section 4. The latter is most relevant to supercritical water cooled reactors and more generally to passive cooling employed in many reactor designs. Finally, we discuss some key issues encountered in the existing Advanced Gas-cooled Reactors (AGRs) and our contribution to maintaining the safe operation of such reactors.

2 MODELLING AND SIMULATION TOOL AND METHODOLOGY

2.1 Overview

The principal thermal hydraulic analysis tools for the design and safety cases of nuclear reactors are the system and sub-channel computer codes, which were initially developed in the 1960s/70s when reactors were first developed. The system codes are based on lumped/network approaches, which are used to simulate the entire reactor system/plant, but with each component, including the reactor core, represented with one or few simply connected 'resistors'. The sub-channel codes are aimed at producing more detailed flow/thermal information in individual components. Such components, e.g., a fuel channel, are split into parallel 'sub-channels', and each is represented using one-dimensional approach whereas the interactions between the sub-channels are modelled using simple methods. Each reactor design has such codes specifically developed and input data/correlations calibrated against experimental data for its use. Consequently, the uncertainties of the codes within their application envelopes are known and the predictions are often conservative as far as safety is concerned. A weakness of such approaches is that they are unable to predict conditions beyond the calibration range, making them inappropriate in new reactor designs when new conditions are explored, and needless to say that they would not provide new understanding of local phenomena, which are sometimes needed for design or safety analysis.

Computational fluid dynamics (CFD) provides a new avenue for thermal hydraulic analysis with a capacity to predict detailed phenomena and flexibility for new designs. This method has been widely explored over the last couple of decades especially along with the development of Gen IV reactors. In the context of nuclear, CFD is a very useful tool for the analysis of separate phenomena but it is far too costly for system analysis due to the computing resources required. A notable trend in NTH development is the multi-scaling approach coupling system/sub-channel codes with CFD. Conversely, high-fidelity CFD has been used to provide reliable data for engineering correlation development and better understanding for local phenomena, hence complementing physical experiments. At Sheffield, we carry out research in both of these directions and a brief overview of which is given below.

2.2 High-fidelity simulations

Our group has developed and maintained a DNS solver, CHAPSim, since late 2000s for both isothermal flows [1, 2] and heat transfer with strong variations of properties [3] based on an incompressible flow formulation. The solver uses a structured mesh with a choice of either Cartesian or cylindrical coordinate and has a number of variants, enabling simulations of flows over rough surfaces using IBM, conjugate heat transfer and LES for flows at higher Reynolds numbers. The code is now being developed by the consortium CCP-NTH (https://ccpnth.ac.uk/) as a nuclear community code. The CHAPSim 1.0 release is based on a second-order spatial finite difference discretisation on a staggered mesh arrangement. The third order Runge-Kutta scheme is used for time advancement. These are combined with the fractional-step method and the solution of the Poisson equation for pressure correction to achieve continuity. Currently CHAPSim 2.0 is being developed, which is based on a six-order compact scheme with pencil (two-directional) parallelisation. All simulations reported herein were carried out used CHAPSim 1.0.

CHAPSim has been used by a number of research groups for studies of transient turbulence physics over smooth (He et al [2, 4]) and rough walls (Seddighi et al [5]), flow laminarisation (He et al [6]), drag reduction (Takrouri [7]), conjugate heat transfer and heat transfer deterioration in supercritical fluid flows (He et al [8]). Below, we look at the transient flow as an example whereas a discussion on the last topic will be given in section 4.

Transient flow is encountered in the start-up and shut-down of nuclear reactors as well as in many postulated transient fault scenarios. The non-equilibrium turbulence in such flows and its effects on heat transfer are complex and difficult to predict. Recently we have carried out a series of DNS/LES simulations using CHAPSim to improve our understanding of such flows [2, 4, 9]. Figure 1 shows an example in which LES is compared with experiments. The flow is initially (statistically) steady turbulent; at time $t = 0$, the flow is accelerated due to the opening of the valve in the experiment (and equivalent flow acceleration applied in simulation). We have demonstrated that such a typical transient turbulent flow is in fact characterised by the formation and growth of a laminar boundary layer, followed by its instability and transition to turbulence even though the initial flow is already turbulent. Figure 1(a) shows that the friction coefficient during the pre-transition can be well represented by the extended Stokes solution for laminar flow and Figure 1(b) shows that following the increase of the flow rate, elongated steaks are initially formed, which become unstable later leading to formation of turbulence spots, exhibiting typical bypass transition characteristics. Figure 1(c) shows the transitional Reynolds dependence on the freestream turbulence, which provides information on the delay between the increase of flow rate and the arrival of the new turbulence, and potentially improved heat transfer. Such a delay in heat transfer was demonstrated in the experiment of Nakamura et al 2020 [10].

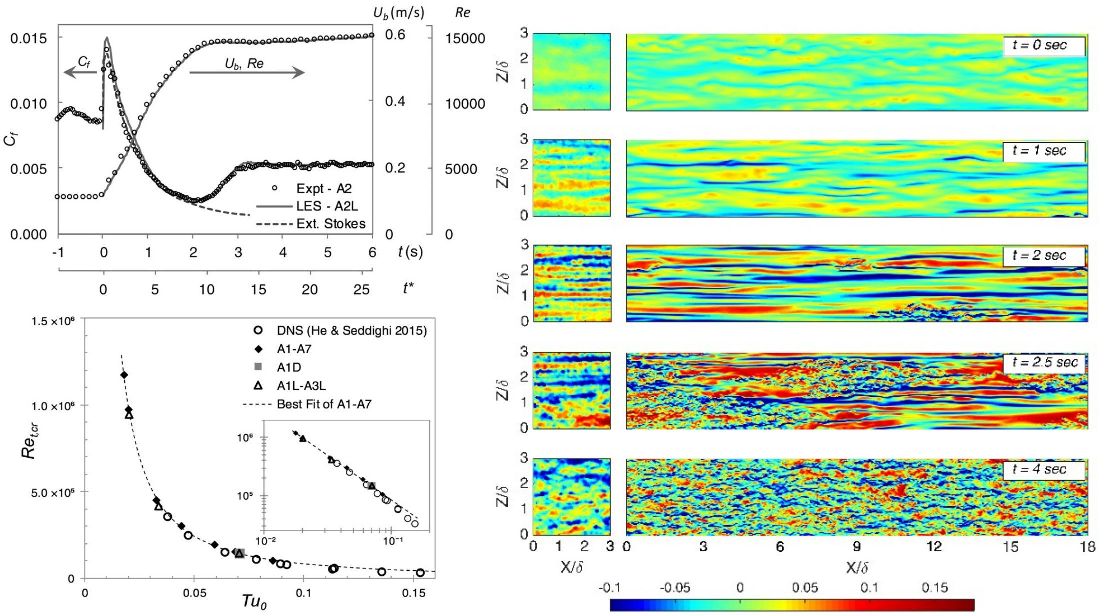


Figure 1: LES/DNS of transient turbulent flow using CHAPSim and comparison with experiment. (a) Top-left: Friction factor, (b) Right: fluctuating velocity on a horizontal plane and (c) Bottom-left: Critical Re against freestream turbulence. This demonstrates that the accelerating turbulent flow is characterised by a laminar boundary layer followed by turbulence transition despite the initial flow is turbulent [9].

2.3 Multi-scale modelling and coarse-grid CFD

At the opposite end of the spectrum to high fidelity simulations, coarse-grid CFD has also attracted significant interest in order to enable CFD to be used for large systems with affordable computer resources. Such methods are aimed at producing the overall/integral quantities (such as friction or heat transfer coefficient) reliably using special modelling methodology to overcome the shortcomings of the use of a coarse-grid mesh. For example, Hu and Fanning [11] used a momentum source term to model the effect of wrap spacers, whereas the model proposed by Viellieber and Class [12] involves first carrying out resolved-CFD simulations for a range of typical conditions and using them to produce distributed body forces to be used in the coarse-grid CFD model enabling the sub-grid flow to be accounted for in the latter. In addition, Roelofs et al [13] adopted the so-called reduced resolution method, in which the mesh is carefully coarsened by neglecting the offset layers near the physical wall. Hanna et al (2020) [14] utilised a surrogate statistical model based on machine learning technology to offset numerical errors arising from grid coarsening.

At Sheffield, we have developed a novel coarse-grid CFD, combining sub-channel method with CFD, which is referred to as Sub-channel CFD (or SubChCFD) [15, 16, 17]. The method is based on iterations between a coarse-grid CFD simulation based on a *computing mesh*, and a pseudo sub-channel solution based on an even coarser *sub-channel mesh* aligned with those used in traditional such-channel. The results from the CFD are integrated to produce sub-channel bulk quantities, which are then used for the solutions at this (sub-channel) level using engineering correlations. These sub-channel solutions are then applied to the CFD through wall treatments, replacing the conventional wall functions. The entire procedure can be implemented on a CFD platform and in our case, is implemented on Code Saturne, an open-source code developed by EDF. By integrating Sub-channel and CFD on a single platform, the method creates a platform for multi-scale modelling of phenomena of different scales.

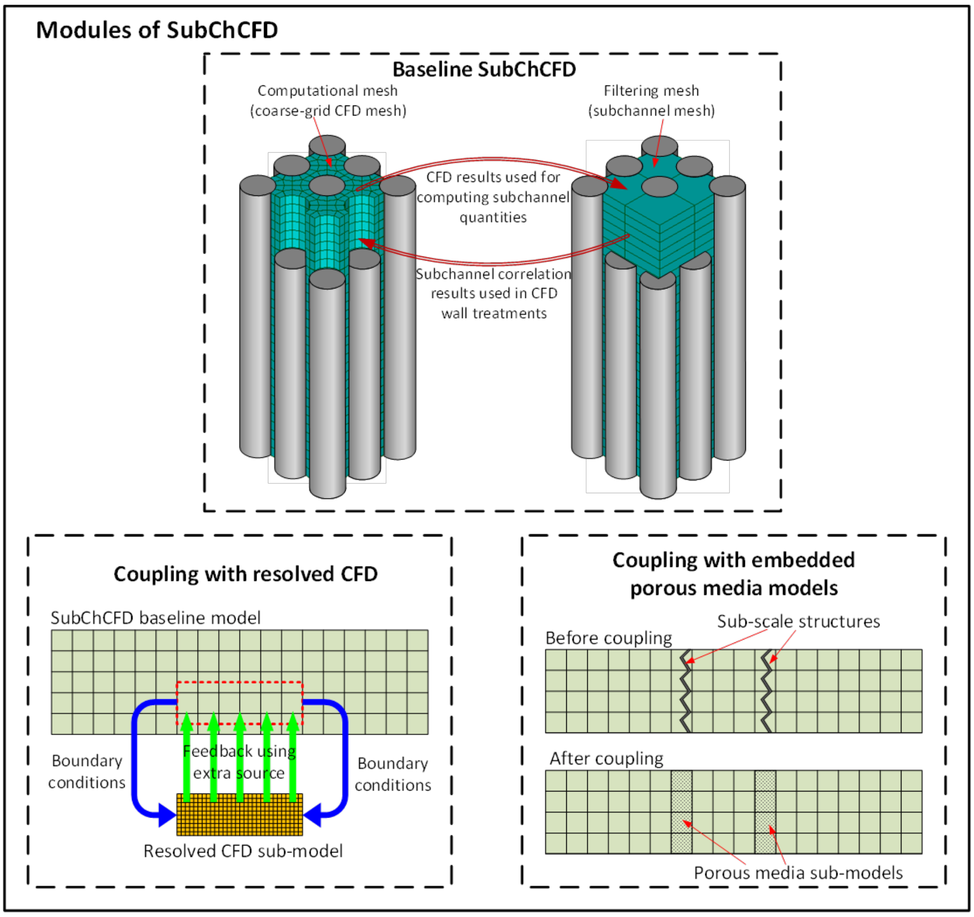


Figure 2: Multi-scaling modelling strategy of SubChCFD [15].

In comparison with conventional sub-channel codes, SubChCFD can be easily coupled with resolved CFD and porous media CFD. Secondly, SubChCFD, as a 3D solver, has overcome the limitation of the 1D approach used in the conventional method, and it does not require the mixing coefficient between sub-channels, for example. It can accurately resolve the first-order flow behaviour (Bernoulli's effect) in contraction/expansion due to spacers. SubChCFD however inherits an important feature of the conventional sub-channel, that is, the code can be calibrated and tuned for a particular reactor design for nuclear safety analysis. The methodology was first published in [15]. Coupling with conventional CFD to produce locally refined solution, and with porous media model to simplify mesh generation for complex local objects such as spacers were developed and demonstrated in [16, 17] respectively. More recently, the capability of SubChCFD was demonstrated by solving the flow and heat transfer in a fuel channel [18]. Figure 2 shows a sketch illustrating the baseline SubChCFD, and the coupling methods. Figure 3 shows that the overall fuel channel is covered by the baseline model, while the spacers are represented by porous models and a chosen span is represented using resolved CFD. It also shows that the SubChCFD predictions agree well with resolved CFD and the measurements.

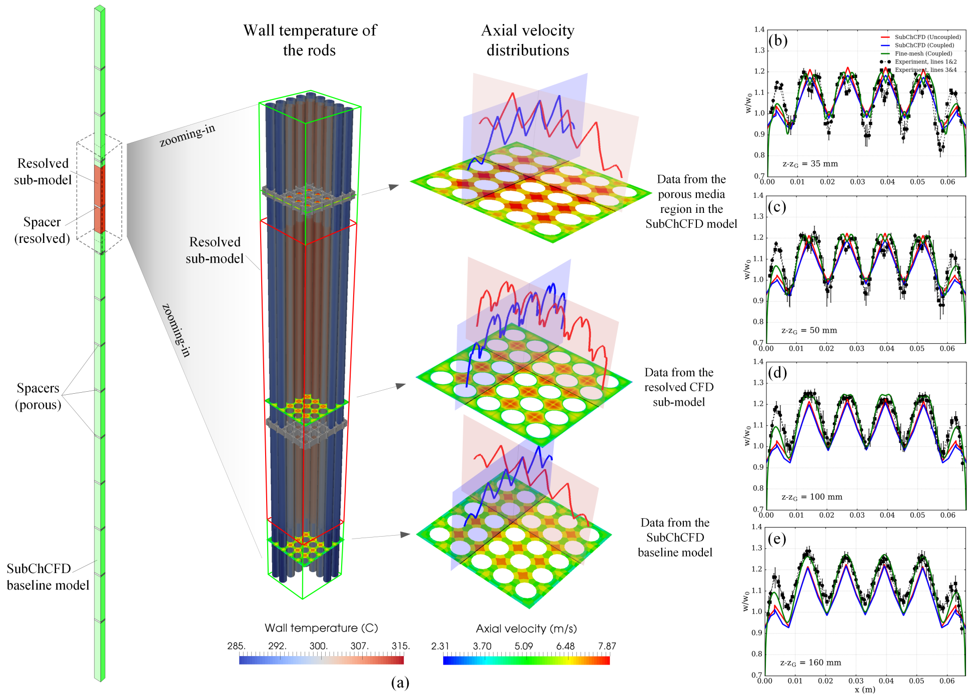


Figure 3: Demonstration of SubChCFD modelling of a fuel channel including coupling porous model for spacers and resolved CFD for one-spacer span, and validation. (a) Left: Overview of the model setup and predictions, and (b)-(e) Right: comparison with experimental data at different heights [15].

3 LIQUID METAL HEAT TRANSFER

Two of the six Gen IV reactors use liquid metal as coolant, that is, the sodium cooled fast reactor (SFR) and the lead-cooled fast reactor (LFR). Together they are referred to as liquid metal fast reactor (LMFR). There are significant similarities between the two designs and hence much research is applied to both reactors. The main thermal hydraulics challenges can be summarised as follows:

- **Liquid metal heat transfer**: An obvious feature is the low Prandtl number (Pr) of liquid metal. The Pr of lead is between 0.01 to 0.1 and that of Sodium between 0.001 to 0.01, whereas the Pr of water is around 1. Turbulence modelling and CFD methods are commonly based on the Reynolds analogy which assumes $Pr \approx 1$, and their applications for liquid metal flow are therefore not always appropriate. A particular challenge is the modelling of turbulence heat flux, which is often modelled using a constant turbulent Prandtl number but this is not sufficient for liquid metal and new models or improved models are needed to be developed [19].
- **Pool thermal dynamics**: The dominant designs of LMFRs are pool types where both the core, the heat exchangers and pumps are submerged in a large pool of liquid coolant, which is at a pressure only slightly above atmosphere. A feature associated with this design is that the flow is highly non-uniform and part of the domain is likely to be inactive. This leads to challenges including potential thermal stratification and flow stagnation in the hot plenum, or flow instability and flow induced vibrations due to the high jet flow from the active core impinging to the above-core structures [20].
- **Core thermal dynamics**: The challenge of the design of the core is to ensure good mixing and effective heat transfer especially considering the small gaps between the fuel pins and various options of wires/spacers used to separate the fuel pins.
- **Cover gas and aerosol dynamics**: There is a region between the free-surface of the liquid pool and the reactor casing, which is filled with an inert gas such as argon to prevent the direct con-

tact between liquid metal and air. The heat loss through this region is significant and needs to be carefully estimated for a number of reasons. The mechanisms are very complex involving radiation, natural circulation, evaporation/condensation and sodium aerosol dynamics. In addition, there may be entrainment of gases into the pool due to uneven free surface causing various negative effects.

An overview of the activities in liquid metal heat transfer at Sheffield is shown in Figure 4, which include studies of cover-gas region aerosol dynamics, hot-plenum pool circulation (E-SCAPE), forced and mixed convection in a three-dimensional flow facility (Tall3D) and flow instability at a T-junction. In the first, we have developed a multi-physics model for the cover gas region, which has been validated using the experimental data collected in Manchester in 1980s'/early 90s' [21]. The model simulates the natural circulation of the argon gas and sodium vapour mixture considering the evaporation/condensation of sodium by solving the momentum, energy and species equations based on RANS formulation. It simulates the formation and growth/diminishing of the sodium aerosol and its interactions with the flow using the general dynamic equation of aerosol. The model also includes radiation using discrete ordinates radiation model. Figure 5(a & b) shows the overall flow structure consists of a primary circulation and a secondary flow. Figure 5(c & d) shows that the model predictions agree well with the experiment data for the heat flux through the rig roof.

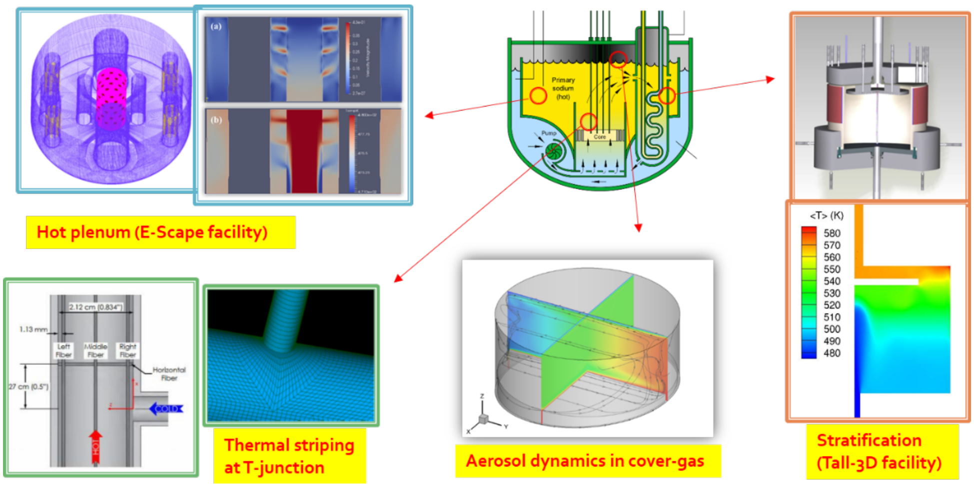


Figure 4: An overview of research in Liquid Metal Fast Reactors (LMFRs).

The Tall3D experiments at KTH were aimed at studying the effect of the 3D flow on the larger system under both steady and transient conditions with LBE as the working fluid [22]. Our study focuses on the 3D flow itself and LES were carried out to study a cooler jet of LBE issuing into a large cavity with warmer fluid under both forced convection as well as mixed convection. Simulations were also carried out with water under otherwise similar conditions to study the effect of low Prandtl number. These simulations provide data for validation of turbulence modelling as well as developing a better understanding of LMHT under such conditions, and hence complement the experiments. Figure 6 shows that the penetration of the negatively buoyancy jet in under mixed convection condition is significantly retarded and the spread is limited to a small region around the jet. The rest of the cavity is largely stagnant with thermal stratification. In comparison, the forced jet impinges on the top plate and causes a strong flow circulation, which in turn changes the characteristic of the jet itself.

The E-Scape facility is a 1/6 scale of the prototype reactor to be developed in Belgium [22]. We have carried out a number of simulations to investigate the complex flow behaviours in this system, including the strong unsteady interactions between the hot and colder streams in the above-core region, the large flow structure and temperature distribution in the hot plenum and the characteristics of the (cold/hot) jets from the barrel holes. It is particularly interesting the that colder fluid appears to occupy the middle height of the hot plenum whereas the top is hot as expected but the lower part of the domain is also occupied by relatively hot fluid due to the flow circulation. This is in contrast to the expectation that the flow may be stratified in less active regions. The hot plenum can be split into flow active and inactive regions, and in the former, turbulent and laminar regions.

The interactions of cold and hotter streams of sodium at T-junctions were studied experimentally at Wisconsin-Madison [22]. Strong thermal fluctuations were found which may lead to thermal fatigue. We have carried out CFD simulations using unsteady RANS to evaluate the capability of such method. It has been found that the model can capture the mean flow distribution reasonable well, but it cannot reproduce the large thermal fluctuations. It appears to be important to use high-fidelity simulations such as LES or DNS in order to reproduce such larger scale unsteady structures [22].

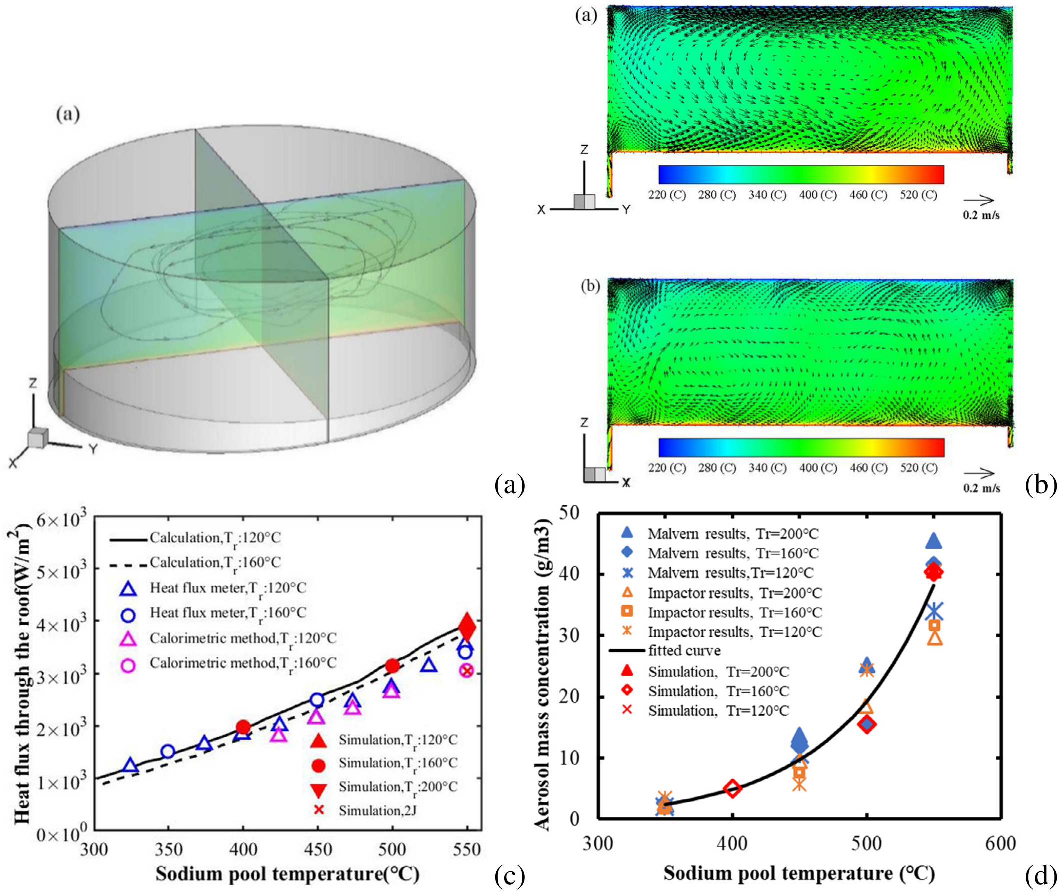


Figure 5: Multi-physics modelling of aerosol dynamics in the cover-gas region above a sodium pool. (a) 3D circulation, (b) Temperature and velocity distributions in two orthogonal planes, (c) Heat loss through roof and (d) aerosol concentration [21].

4 SUPERCRITICAL WATER COOLED REACTORS

Supercritical water cooled reactor (SCWR) is the only light water reactor in Gen IV, which is building on the experience of existing water-cooled nuclear reactors and conventional supercritical thermal power plant. The basic design is a single-loop system with no phase change in the reactor core, making it a relatively simple system. The challenge is however that while there is no phase change as the fluid is heated, its thermal properties may change drastically as the temperature crosses the pseudo-critical temperature and the fluid changes from a liquid-like to a gas-like state. In fact, the density of the water increases

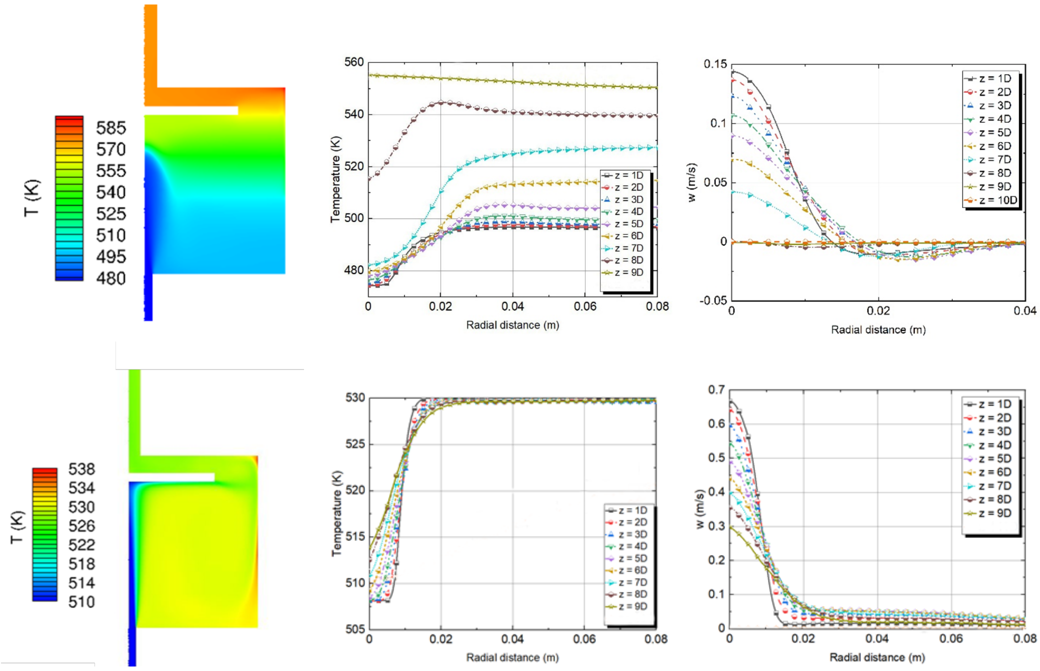


Figure 6: A liquid LBE jet issuing into a cylindrical cavity with a heated wall (Tall3D facility) under mixed (top) and forced convection conditions (bottom) [22]. From left to right, the sub-figures show the temperature contours, radial profiles of temperature and velocities.

around 7 times from the bottom to the top of the reactor core, which is greater than that in a boiling water reactor. Such strong changes in thermal properties (and especially density) may suppress turbulence causing the so-called heat transfer deterioration (DHT). To make things worse, this phenomenon is very challenging to predict with CFD.

The research on heat transfer to fluids at supercritical pressure was actively pursued initially in the 1960s/70s in support of the development of the conventional supercritical power plant but was very much paused until the turn of the new millennium, when interest has revived. Extensive experimental and computational activities have been developed since due to a number of 'new' found applications such as nuclear energy (for the primary or secondary loop), low-grades heat power cycles, and the cooling of rocket engines for example. Our early studies focused on the development and assessment of modelling approaches using conventional RANS CFD for the prediction of heat transfer behaviours under strong influences of buoyancy. We assessed the performances of turbulence models in predicting the DHT against experimental data [24], and identified that the significant influence of flow acceleration due to heating (rather than boundary) to be the primary cause for laminarisation and DHT in a small diameter tube [25]. By comparing with the early DNS data, we analysed the key turbulence model parameters and how they are related to the model performance in predicting such flows [26]. We found that whereas the direct effect (modelling of the buoyancy production) is not important in predicting the flow laminarisation and DHT, it may be the key reason that nearly all turbulence models are unable to predict the recovery under the influence of strong buoyancy. More recently, we investigated into the fluid-to-fluid scaling to facilitate the use of data based on CO2 or other surrogate fluids to reduce experimental challenges with water [27]. We studied flow in a horizontal pipe and identified the staged development [28]. In particular, we noted that whereas the flow in heated supercritical pressure is known to develop continuously, they can reach a pseudo-developed stage where the heat transfer behaviours do not change downstream.

More recently, our focus has been on using direct numerical simulation (DNS) to develop fundamen-

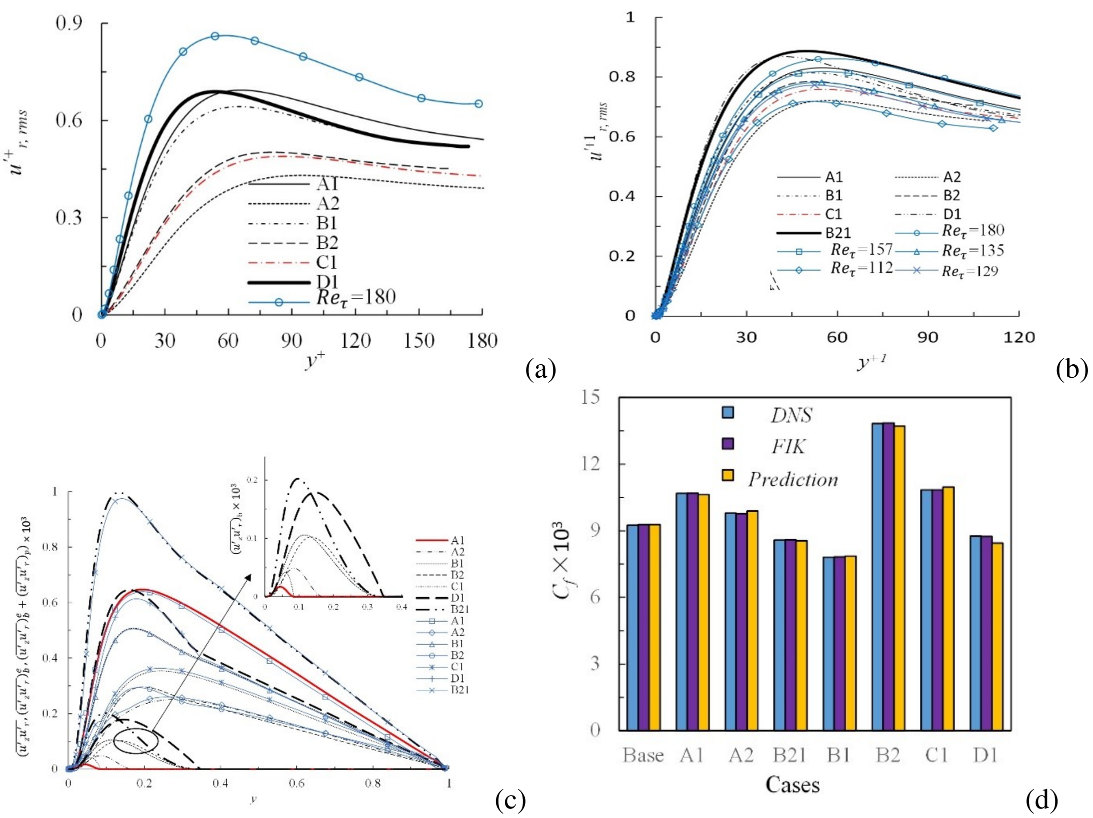


Figure 7: The Apparent Reynolds number theory for laminarisation due to streamwise body forces [6]. (a) Turbulence (v') reduction in conventional scaling, (b) turbulence compares well with corresponding reference flows in apparent friction scaling, (c) prediction of turbulence shear stress agrees well with DNS and (d) friction factor. A1, A2,... are DNS cases with different profiles/densities of non-uniform body force imposed.

tal understanding of the mechanisms of laminarisation and DHT. Initially we focused on buoyancy alone and isolate it from all other effects using a prescribed body force with a linear or step change profile to emulate the buoyancy in an isothermal flow setting (the latter may occur when the pseudo-critical temperature happens to be in the thermal boundary layer). A useful outcome of that study was the proposal of the concept of Apparent Reynolds Number (ARN) used to explain the flow laminarisation. Conventionally the flow is said to be partially or fully laminarised under the influenced by buoyancy or other effect when compared with a reference flow of equal flow rate without such effects. In contrast, He et al [6] established that the key turbulence characteristics remain largely unchanged when they are compared with those of a new reference flow based on equal (equivalent) pressure gradient (EPG). This pressure gradient was used to define the apparent Reynolds number for the flow. The complex non-equilibrium turbulence in laminarising flows has been found to be readily predicted using this simple EPG reference flow and the body-force profile.

Figures 7(a & b) shows that the turbulence is reduced in various body-force influenced flows in conventional presentation, but it remains largely the same when normalised using the apparent friction velocity. Figures 7(c & d) shows that the turbulent shear stress and the friction factor can be reasonably predicted by the ARN theory using only the EPG reference flow data and the body-force profiles.

The ARN theory was then applied to analyse DNS data of a heated air flow in a vertical pipe subjected to strong buoyancy (Marensi et al [23]). The DNS data show that for a given Reynolds number, with the increase of the buoyancy (heating) the flow may become laminarised or turn into a convective shear flow (in the recovery region) (Figure 8a). The ARN theory produces two lines shown in the graph and suggests

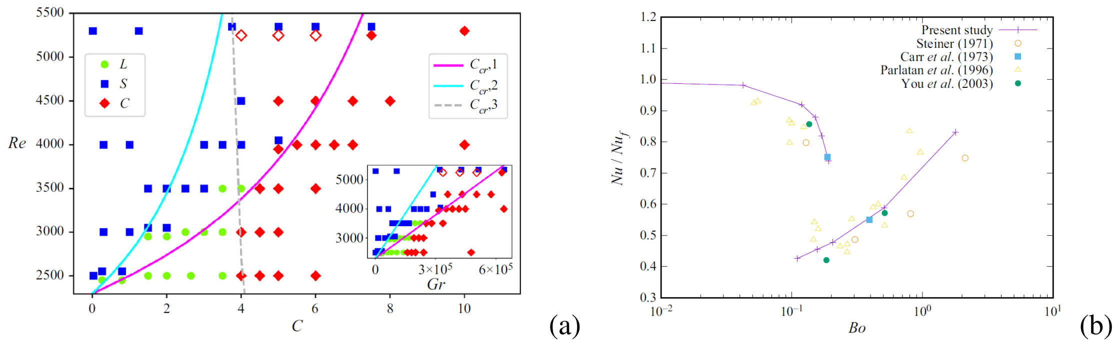


Figure 8: Transition map for shear-driven (S), convective-driven (C) turbulent and laminarised (L) flows due to buoyancy. Markers: DNS, blue and purple lines: boundaries prediction by Apparent Reynolds number theory. (a) Regions on Re-C, where C=Gr/16Re, (b) Nusselt number ratio versus buoyancy parameter (Bo) [23].

that below the pink line, there will be no wall shear state (i.e., the flow will be in either fully laminarised or in the recovery region) and on the left of the blue line, the flow will not be fully laminarised. In between the two lines, the flow may be in any states dependent on if the simulation (or experiment) starts from a laminar or turbulent flow. It can be seen from the figure that the above hypothesis is in an excellent agreement with the DNS data. Some of the simulations are replotted in figure 8(b) in the format that is often used in the literature, where the Nusselt number ratio is shown against the buoyancy parameter. The bi-state nature predicted by the ARN theory is clearly evident in this figure, which can potentially explain the large spread of experimental data often shown in this region. Indeed, one should be prepared for both eventualities to happen in designing an engineering system.

In our latest study, the ARN theory was applied to an even more challenging problem of a flow of CO2 at supercritical pressure under strong heating (He et al [8]). The flow exhibits complex behaviours under the influences of a number of factors including variable viscosity, flow acceleration due to heated expansion and buoyancy. The flow and thermal fields also develop rapidly spatially from the start of heating leading to strong convection/inertial effects (Figure 9(a)). The application of the ARN theory has led to the proposal of a unified explanation for all these effects (Figure 10), in which each effect is represented by a pseudo-body force. Figures 9(b & c) show the contribution of each effect to the flow distortion. It can be seen that the effect of the variation of viscosity is as strong as that of the buoyancy during most part of the laminarising stage (say up to t=10). It is also interesting to see that during this stage, inertia exhibits a very significant impact on the flow which compensates partly some of the other effects. Figure 9(d) shows that the suppression of turbulence ($\overline{u'v'}$) during flow laminarisation can be well predicted using the ARN theory even for such a complex flow.

5 ADVANCED GAS COOLED REACTORS (AGRS)

Up to 2020, twelve of the thirteen rectors that are currently in operation in the UK are AGRs. Such reactors are unique to the UK. They are significantly larger in size than the water reactors due to the use of gas (CO2) as the primary coolant. Its efficiency is around 40%, which is significantly higher than those of the PWR/BWR, which are typically just above 30%. The AGRs use graphite as moderator in the form of graphite bricks, the stack of which forms the core structure. Currently all AGR reactors have passed their design lives and their safe operations are justified by the so-called lifetime extension safety cases. A major threat to AGRs is the cracking of the graphite bricks which may eventually cause the dysfunction of the cooling channels and force the reactors to retire.

Due to the cracking and relative movement of the brick layers, the fuel channels may be distorted, and the main annual flow passage between the brick wall and the fuel stringers may be significantly

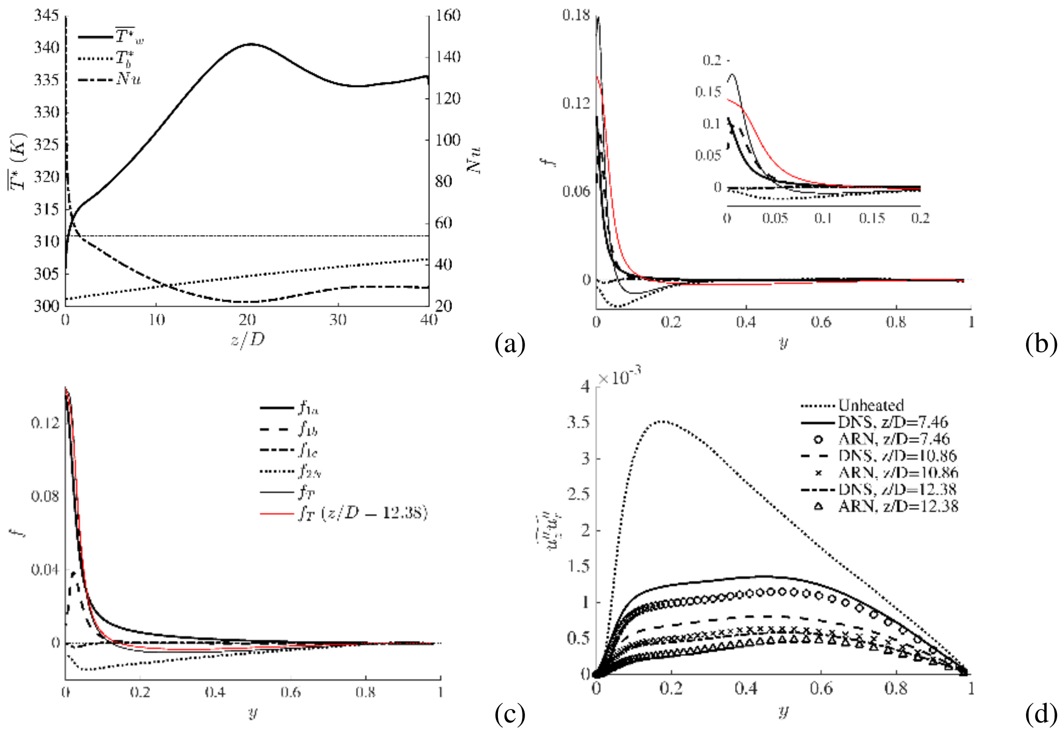


Figure 9: (a) Variations of temperature and Nusselt number in a heated vertical pipe at supercritical pressure, (b) & (c) Representation of buoyancy (f_{1a}), variable viscosity (f_{1b}), radial density profile (f_{1c}), and inertia (f_{2N}) using pseudo body forces at z/D=7.46 and 12.38 and (d) Prediction of turbulent shear stress using the ARN theory [8].

distorted, resulting in an eccentric flow passage. The cooling of the brick is mostly provided by this flow and hence the eccentricity may cause local hot spots in the graphite bricks further speeding up the core damage. At Sheffield, we have studied the effect of the eccentricity for a number of reactors, taking into consideration of the real reactor geometries, boundary conditions and thermal properties of the irradiated graphite and CO2. It has been found that in most cases, the maximum increase in graphite temperature is less than 30 °C and their direct effect on the integrity of the brick is moderate.

The second area of our research is fuel-route thermal hydraulics analysis and tools development. After removed from the reactor, the spent fuel elements are moved to Irradiated Fuel Buffer Store to allow the decay heat to reduce to a suitable level before being dismantled in the Irradiated Fuel Dismantling Facility (IFDF), and finally stored in the cooling pond. This fuel handling process is the time during which there is a much higher possibility for accidents to occur than in normal operation. The industry-standard thermal analysis tool is based on highly conservative 1D analyses. For example, if the forced cooling is unavailable for any reason, the cooling is assumed to be dependent on conduction and radiation only and the natural circulation is completely neglected leading to strong conservative estimation. Recently we have developed a 3D CFD based model, FREEDOM, for both intact and damaged fuel bundles with carbon/fuel oxidisation as well as the decay heat considered. The fuel rods are considered using inhomogeneous porous model in the fluid model, which is coupled with a rods-resolved thermal conduction/radiation model. The model is carefully validated against experimental data and is being integrated into the EDF fuel route analysis tool portfolio [31].

To develop fundamental understanding of the cooling of a fuel element under postulated accidental conditions when external cooling is unavailable, we carried out LES simulations of natural circulation in a quarter- and a full-height fuel element [29, 30]. Even though natural circulation in simple geometries

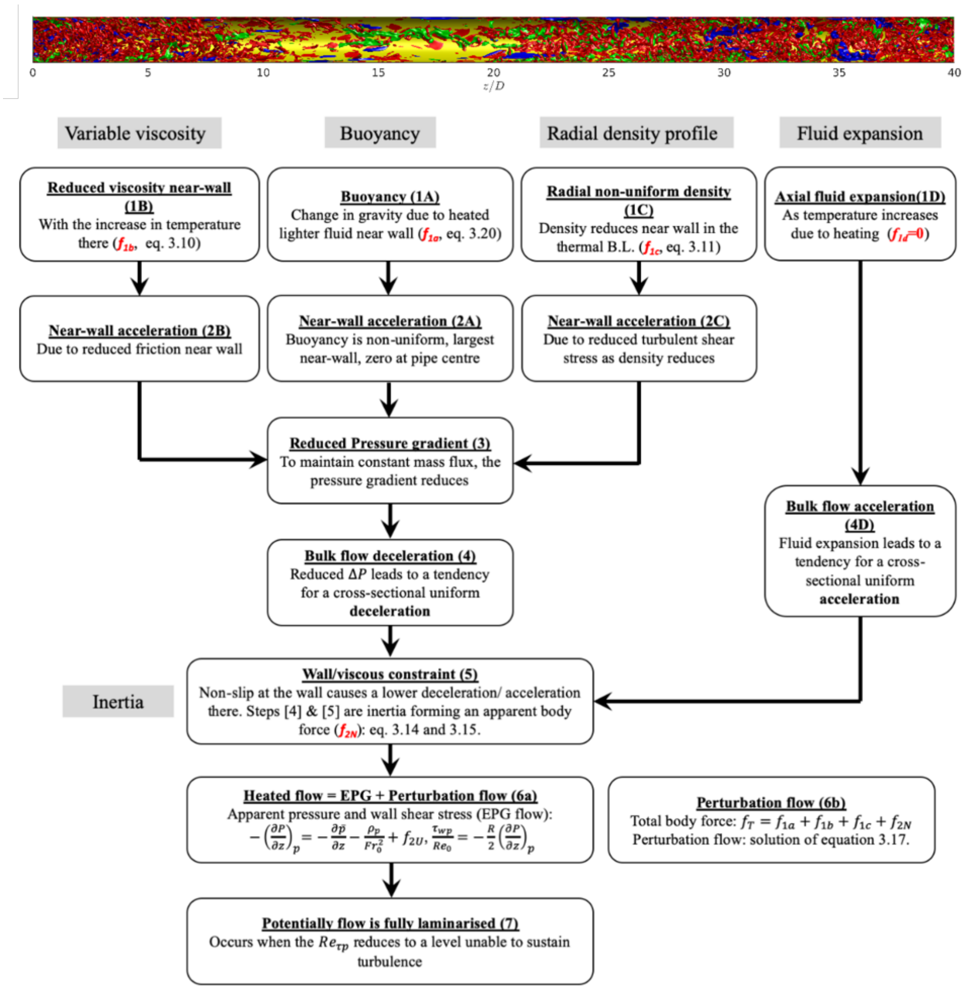


Figure 10: A unified explanation for flow laminarisation in a heated flow. Buoyancy, thermal expansion, variable properties and inertia are all treated as (pseudo) body forces and their effects are explained using the apparent Reynolds number theory [8].

such as rectangular or annular cavities are well researched, data on rod bundles are rare. It has been found in our study that in the shorter channel, most of the domain is filled with stagnation and stratified fluid; the rod is cooled by a boundary layer developing along their surfaces, and the influences from neighbouring rods are negligible (Figure 11(a)). In contrast, the flow between the heated rods in the central region of the longer channel is akin to a heated pipe/channel after a short entrance length (Figure 11(b)). As a result, the well-established heat transfer correlations for pipes are expected to be applicable with small modifications. On the other hand, the flow over the cooled containment wall resembles the flow in a simple, rectangular, asymmetrically heated/cooled cavity, and the heat transfer to the containment is closely represented by a natural convection correlation, being initially laminar, then transition, and finally turbulent (Figure 12).

6 CONCLUSIONS

Recent thermal hydraulics research is very often motivated by the development of advanced and/or small modular reactors, though there are significant activities associated with the life-time extension of current reactors and nuclear new build. The challenges in advanced reactor designs are often associated with the use of 'uncommon' fluids as the primary loop coolant, i.e., liquid metal, molten salts, or water

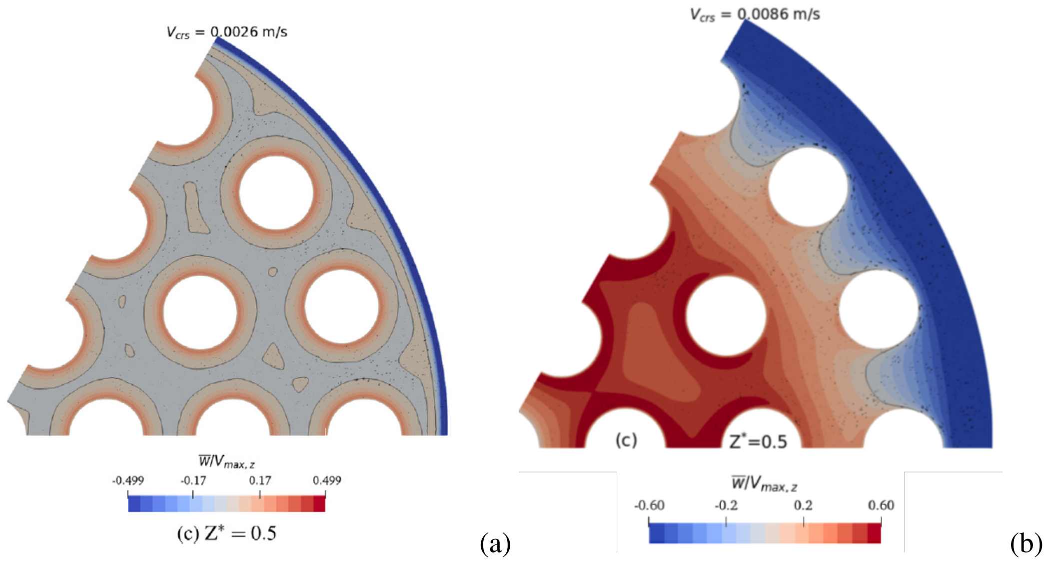
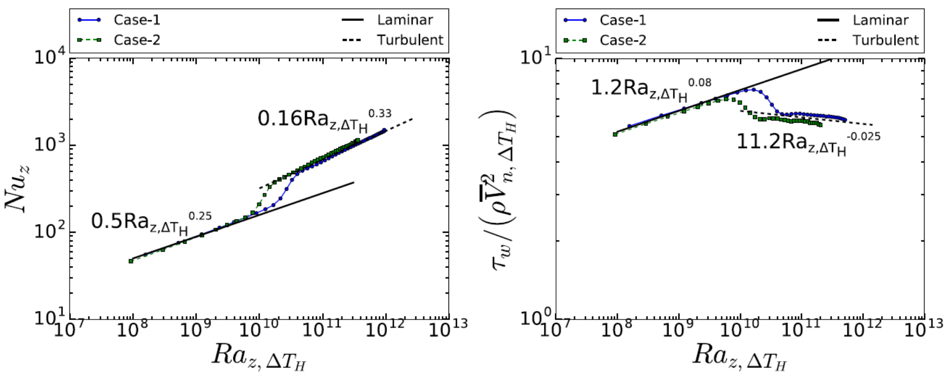


Figure 11: Vertical velocity distribution in a (a) 1/4th- and (b) full-height AGR fuel bundle. In the former, the fuel pins are cooled by a boundary around them with a stagnant fluid in most part of the channel. The flow in the latter case occupies the whole channel as in a 'normal' channel flow [29, 30].

at supercritical pressure. New data, understanding and analysis tools are required to address the specific characteristics of such reactors due to their specific designs and/or thermal-physical properties. In addition, the recent development of computer technologies and numerical methods have presented both a challenge and opportunity to modernise the thermal hydraulics tools and analysis. On the one hand, there are significant activities in developing multi-physics, multi-scaling modelling to leverage the traditional system/sub-channel methods, and on the other hand, effects have been made to develop 'numerical' experimental capabilities using high-fidelity simulations based on High Performance Computing (HPC) to generate new understanding and data to complement physical experiments.

In this paper, we have reviewed the research carried out in our group in the field of nuclear reactor thermal hydraulics. In the area of simulation and modelling tools, we have developed a DNS code CHAPSim which has been used by a number research groups for the study of non-equilibrium turbulence that may be encountered in reactor systems including unsteady turbulence, flow over rough surfaces, conjugate heat transfer, mixed convection and heat transfer deterioration. For the latter, we have made significant endeavours in improving our understanding of flow laminarisation due to buoyancy and variations of thermal properties. We have directed much of our efforts towards supercritical water-cooled reactors but the findings are also relevant to passive cooling in other reactor designs. We have established a new perspective for flow laminaration based on the concept of apparent Reynolds number, which enables the prediction of such phenomena significantly more easily with the knowledge of 'equilibrium' turbulence only. Additionally, we have developed a novel coarse-grid CFD combining sub-channel and CFD methods, which facilitates multi-scale modelling for large nuclear systems. In the area of liquid-metal fast reactors, we have carried out numerical simulations for the studies of stagnation and stratification in large pools of liquid metal (in the hot plenum or a cavity), thermal stripping due to interactions and mixing of streams with different temperatures, and sodium aerosol dynamics in the area above a liquid pool. To support the operation and safety cases for the AGRs currently in operation in the UK, we have supported work towards understanding the effects of a distorted reactor core due to the cracking of the graphite bricks on the fuel and reactor core life; and analysed the flow physics and cooling effectiveness of natural circulation in an enclosed fuel bundle under the condition of a lost coolant accident and developed a 3D simulation tool for engineering analysis for potential off-design conditions



(a) Nusselt number (b) Friction factor

Fig. 21. Nusselt number and friction factor plotted against the Rayleigh and Grashof number, respectively at the containment wall.

Figure 12: Nusselt number and friction fator plotted against Rayleigh and Grashof number at the containent wall of a full-height fuel bundle [30].

in the fuel route.

ACKNOWLEDGEMENTS

The work reported herein have been carried out by numerous PhD students and research associates in collaboration with a number of research groups worldwide. I would specially like to acknowledge the contributions of Mehdi Seddighi (MS) and Wei Wang (WW) in developing the code CHAPSim and studying transient flow and supercritical pressure flow, respectively; Bo Liu in developing Sub-channel CFD; MS, Sam Gorji, Akshat Mathur on study of transient flows; Kui He, MS on turbulence laminarisation; Xiaoxue Huang, Ashish Saxena, Matthew Falcone on LMHT; WW, Jundi He, Muhsin Mohd Amin, Yu Duan (YD), Junjie Yan, Ran Tian on heat transfer to fluids at supercritical pressure and YD, Cosimo Trinca and Kenneth Chinembiri on AGR related research. I am in-debt to my PhD supervisor and long-term mentor Professor JD Jackson and long-term collaborators Walter Ambrosini and Peixue Jiang and their colleagues Andrea Pucciarelli and Ruina Xu.

The work reviewed in the paper has been funded by a number of organisations and funding bodies. We acknowledge the recent funding that directly funded the reported work: EPSRC funding through the Indo-UK initiative (EP/K007777/1,EP/M018733/1), NEUP/EPSRC collaboration (EP/T002395/1, EP/T002417/1), and Computational Collaborative Projects (EP/T026685/1); UK Government BEIS Nuclear Innovation Program (TRN 1210/09/2016(3) & TRN 1659/10/2018); EU H2020 (Grant Agreement ID: 945234); UK Turbulence Consortium (EP/R029326/1); IAEA Collaborative Research Project (CRP); and EDF through a number of PhD studentships, research and consultancy contracts.

REFERENCES

[1] M. Seddighi. Study of turbulence and wall shear stress in unsteady flow over smooth and rough wall surfaces. *University of Aberdeen (Ph. D. thesis)*.

[2] S. He & M. Seddighi. Turbulence in transient channel flow. *Journal of Fluid Mechanics*, **715** (2013) 60–102.

[3] W. Wang & S. He. Mechanisms of buoyancy effect on heat transfer in a horizontal flow. In *Proceedings of the 7th International Symposium on Supercritical Water-Cooled Reactors (ISSCWR-7)*, pp. 15–18 (2015).

[4] S. He & M. Seddighi. Transition of transient channel flow after a change in Reynolds number. *Journal of Fluid Mechanics*, **764** (2015) 395–427.

[5] M. Seddighi, S. He, D. Pokrajac, T. O'Donoghue, & A. Vardy. Turbulence in a transient channel flow with a wall of pyramid roughness. *Journal of Fluid Mechanics*, **781** (2015) 226–260.

[6] S. He, K. He, & M. Seddighi. Laminarisation of flow at low Reynolds number due to streamwise body force. *Journal of Fluid mechanics*, **809** (2016) 31–71.

[7] K. Takrouri. *Study of Turbulence and Drag Reduction for Flow Over Backswimmer Textured Surface*. Liverpool John Moores University (United Kingdom) (2021).

[8] J. He, R. Tian, P. Jiang, & S. He. Turbulence in a heated pipe at supercritical pressure. *Journal of Fluid Mechanics*, **920**.

[9] A. Mathur, S. Gorji, S. He, M. Seddighi, A. Vardy, T. O'Donoghue, & D. Pokrajac. Temporal acceleration of a turbulent channel flow. *Journal of Fluid Mechanics*, **835** (2018) 471–490.

[10] H. Nakamura, R. Saito, & S. Yamada. Delay in response of turbulent heat transfer against acceleration or deceleration of flow in a pipe. *International Journal of Heat and Fluid Flow*, **85** (2020) 108661.

[11] R. Hu & T. H. Fanning. A momentum source model for wire-wrapped rod bundles—Concept, validation, and application. *Nuclear Engineering and Design*, **262** (2013) 371–389.

[12] M. Viellieber & A. G. Class. Anisotropic porosity formulation of the coarse-grid-CFD (CGCFD). In *International Conference on Nuclear Engineering*, volume 44984, pp. 473–483. American Society of Mechanical Engineers (2012).

[13] F. Roelofs, V. Gopala, L. Chandra, M. Viellieber, & A. Class. Simulating fuel assemblies with low resolution CFD approaches. *Nuclear engineering and design*, **250** (2012) 548–559.

[14] B. N. Hanna, N. T. Dinh, R. W. Youngblood, & I. A. Bolotnov. Machine-learning based error prediction approach for coarse-grid Computational Fluid Dynamics (CG-CFD). *Progress in Nuclear Energy*, **118** (2020) 103140.

[15] B. Liu, S. He, C. Moulinec, & J. Uribe. Sub-channel CFD for nuclear fuel bundles. *Nuclear Engineering and Design*, **355** (2019) 110318.

[16] B. Liu, S. He, C. Moulinec, & J. Uribe. A coupling approach between resolved and coarse-grid sub-channel CFD. *Nuclear Engineering and Design*, **377** (2021) 111124.

[17] B. Liu, S. He, C. Moulinec, & J. Uribe. Coupled porous media approaches in sub-channel CFD. *Nuclear Engineering and Design*, **377** (2021) 111159.

[18] B. Liu, S. He, C. Moulinec, & J. Uribe. A multiscale model of a rod bundle using subchannel CFD. *Submitted for publication*.

[19] E. Merzari, P. Fischer, H. Yuan, K. Van Tichelen, S. Keijers, J. De Ridder, J. Degroote, J. Vierendeels, H. Doolaard, V. Gopala, *et al.* Benchmark exercise for fluid flow simulations in a liquid metal fast reactor fuel assembly. *Nuclear Engineering and Design*, **298** (2016) 218–228.

[20] D. Laurence, H. Iacovides, T. Craft, A. Cioncolini, C. Moulinec, S. He, & B. Liu. Critical Review of State-of-the-Art Thermal Hydraulic Prediction Capability. Frazer-Nash Consultancy. *FNC 53798/46733R Issue 1*.

[21] X. Huang & S. He. Numerical modelling of cover gas thermal hydraulics in Sodium-cooled Fast Reactors. *Nuclear Engineering and Design*, **355** (2019) 110347.

[22] X. Huang & S. He. Liquid Metal Thermal Hydraulics. Frazer-Nash Consultancy. *FNC 60148/51220R*.

[23] E. Marensi, S. He, & A. P. Willis. Suppression of turbulence and travelling waves in a vertical heated pipe. *Journal of Fluid Mechanics*, **919**.

[24] S. He, W. Kim, & J. Jackson. A computational study of convective heat transfer to carbon dioxide at a pressure just above the critical value. *Applied Thermal Engineering*, **28** (2008) 1662–1675.

[25] S. He, P.-X. Jiang, Y.-J. Xu, R.-F. Shi, W. Kim, & J. Jackson. A computational study of convection heat transfer to CO2 at supercritical pressures in a vertical mini tube. *International journal of thermal sciences*, **44** (2005) 521–530.

[26] S. He, W. Kim, & J. Bae. Assessment of performance of turbulence models in predicting supercritical pressure heat transfer in a vertical tube. *International Journal of Heat and Mass Transfer*, **51** (2008) 4659–4675.

[27] A. Pucciarelli, S. He, & W. Ambrosini. A successful local fluid-to-fluid similarity theory for heat transfer to supercritical pressure fluids: merits and limitations. *International Journal of Heat and Mass Transfer*, **157** (2020) 119754.

[28] R. Tian, S. He, M. Wei, & L. Shi. The staged development of a horizontal pipe flow at supercritical pressure. *International Journal of Heat and Mass Transfer*, **168** (2021) 120841.

[29] K. Chinembiri, S. He, J. Li, & C. Trinca. Natural circulation in a short-enclosed rod bundle. *International Journal of Heat and Mass Transfer*, **179** (2021) 121674.

[30] K. Chinembiri, S. He, & J. Li. Natural circulation in an enclosed rod bundle of large aspect ratio. *Applied Thermal Engineering*, **188** (2021) 116534.

[31] K. Chinembiri, S. He, C. Trinica, & J. Li. DEVELOPMENT AND DEMONSTRATION OF A POROUS MODEL FOR ADVANCED GAS-COOLED REACTOR (AGR) FUEL BUNDLES. In *The 19th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-19), Brussels, Belgium, March 6 - 11, 2022* (2022).