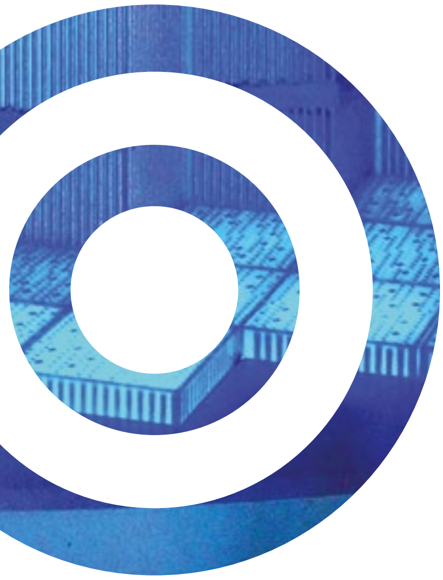
archer2



High-Fidelity Simulations to Improve Performance and Safety of Nuclear Reactors

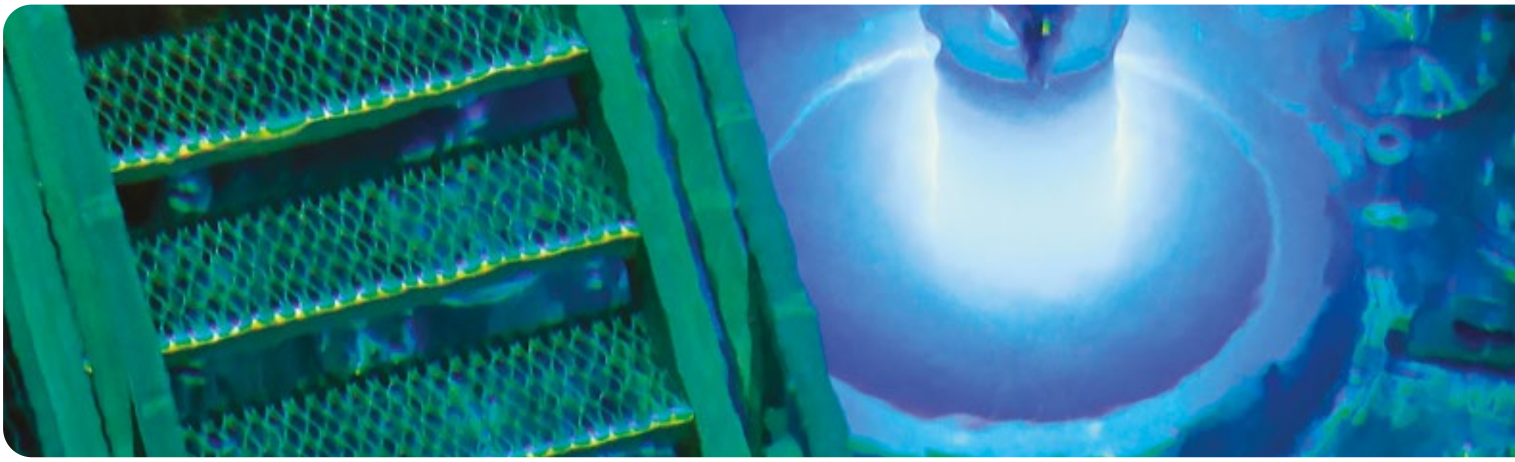
Nuclear power, characterised as a sustainable and environmentally friendly low carbon energy source, stands as a pivotal component of the UK's ambitious Net-Zero strategies for 2050. Central to the use of nuclear power is the industry's paramount commitment to safety. Researchers from the STFC Daresbury Laboratory, the University of Sheffield and EDF R&D have leveraged the computational power of ARCHER2 to carry out high-fidelity Computational Fluid Dynamics simulations of Pressurised Water Reactor fuel assembly under conditions of clad ballooning, mimicking a potential issue with the integrity of a fuel rod. This endeavour aims to gain a deeper understanding of the safety implications of such a scenario, including potential coolant channel blockages, as well as the resulting coolability impairment and heat transfer alteration in decay heat removal.

Science and Technology Facilities Council

University of Sheffield

EDF

epcc



www.archer2.ac.uk

As a low carbon energy source, nuclear power provides a reliable and consistent supply of electricity without directly producing greenhouse gas emissions. Pressurised Water Reactors (PWRs) are currently the most widely used nuclear reactor design. Their core comprises an array (or bundle) of fuel rods, separated by small channels, where the coolant flows. This design provides effective heat transfer from the fission fuel to the coolant (see figure 1). Ensuring the safety of Nuclear Power Plants (NPPs) is of paramount importance to the harvesting of nuclear energy. However, it remains a constant challenge throughout the construction, deployment, and operation of NPPs. The analysis of hypothetical accidents within reactor design is a fundamental approach to ensuring the safety of nuclear reactor systems. One such critical scenario is the Loss of Coolant Accident (LOCA), which poses significant challenges to the safe operation of the reactors. In the event of a large break LOCA, rapid depressurisation may occur alongside local overheating, causing the pressure within the fuel rods to increase and resulting in clad “swelling” or “ballooning”. This phenomenon directly impacts on the coolant channels, causing local narrowing or complete blockage, and consequently, fuel relocation and/or escalation of power and temperature in the ballooned region, posing substantial safety risks.

Computational Fluid Dynamics (CFD), which is capable of providing sufficient resolution and flexibility to capture the complex 3-dimensional physics of the flow, proves ideal for an in-depth examination of the flow redistribution near ballooning and the resultant negative impact on heat transfer. However, most of the existing CFD studies are limited to using lower-order/fidelity approaches or are restricted to small-scale rod bundle configurations due to the enormous computing power typically required for detailed simulations. The inherent uncertainties of the lower-order/fidelity approaches bring difficulties in accurately predicting complex flows, particularly in scenarios involving flow separations, swirling and strong anisotropy. In contrast, high-fidelity approaches such as Large-Eddy Simulation (LES) would help capture most of the physics involved and provide a better understanding of the ballooning effects. LES of a large-scale full fuel assembly requires a very large mesh to numerically solve the governing equations, thus resulting in an extremely high demand for computational resources. Thanks to gaining access to a machine of the size of the ARCHER2 system, performing a fully resolved LES of the clad ballooning phenomena in a large-scale fuel assembly has become feasible.

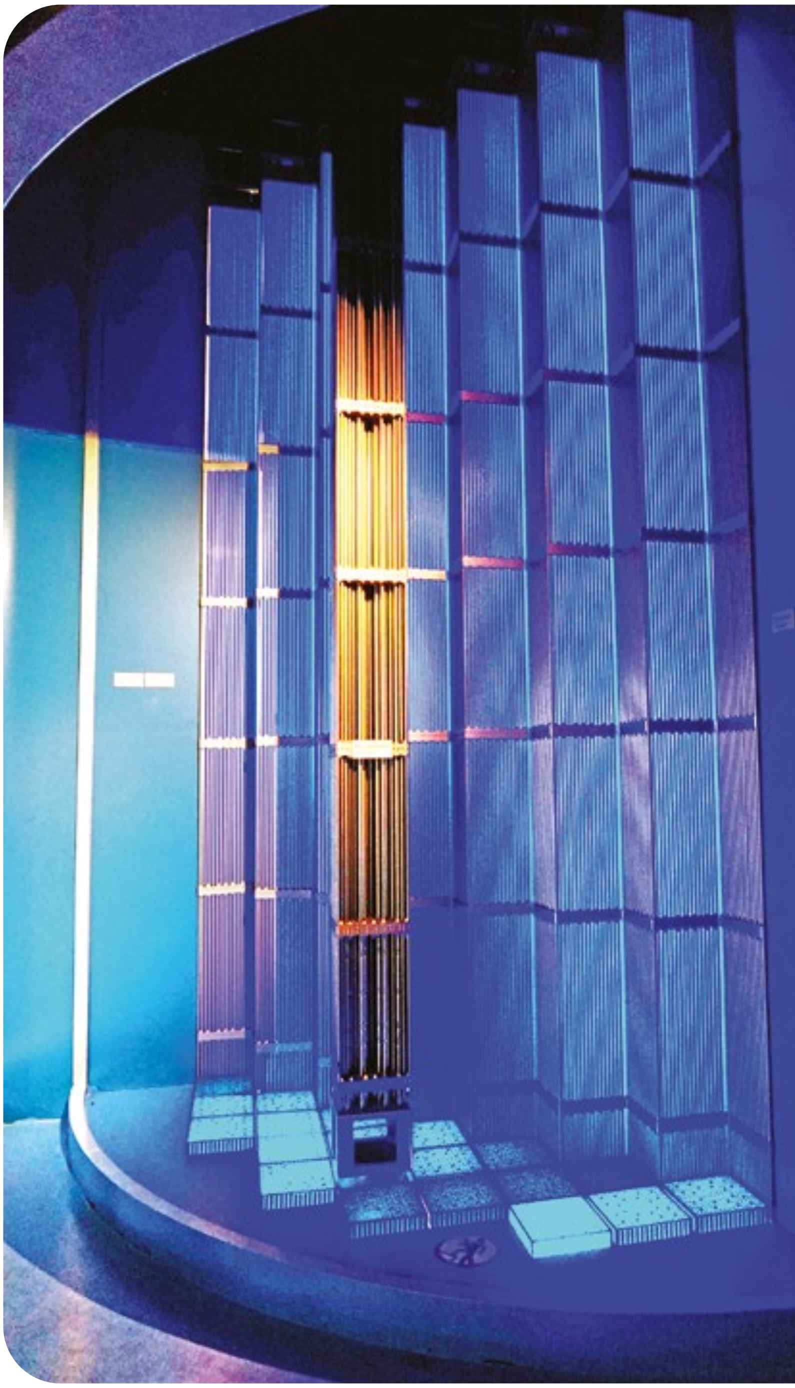


Figure 1: Core design of a typical PWR (credit to Wikimedia).

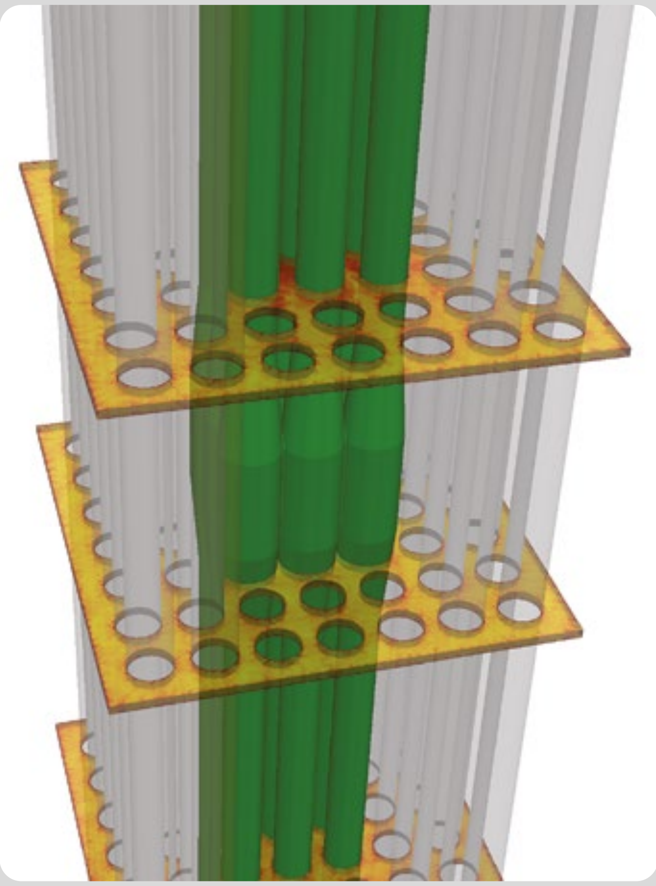


Figure 2: Configuration of a 7x7 PWR fuel bundle with ballooning at the nine central rods (the ballooned rods are highlighted in green). The velocity contours predicted by Large-Eddy Simulation are also shown for some horizontal planes at several axial locations near the ballooning.

In this study, a square-lattice 7x7 rod bundle configuration, with trapezoid-shaped blockages positioned at the nine central rods, is used to mimic the ballooning phenomenon that could occur in a large break LOCA (see figure 2). An open-source multi-purpose CFD package, code_saturne[1], is used for the CFD modelling. code_saturne has demonstrated excellent scalability on ARCHER2, enabling the use of up to 512 nodes (65,536 CPU cores) – or almost 10% of the full ARCHER2 system – for the numerical simulations outlined in this study, which ensures a fast production of the simulation results.

In comparison to the lower-order/fidelity methodologies, LES is better at capturing turbulence effects, particularly in the immediate vicinity downstream of the blockage, thereby producing more accurate predictions of temperature distribution within these regions. The LES performed in this work provides the very first fully resolved high-fidelity simulation of a large-scale PWR fuel assembly with locally ballooned rods. The resultant dataset contributes to deepening the understanding of the multi-physical phenomena of coolant flow and heat transfer and thermo-mechanical behaviours of the fuel during the ballooning process. Acquiring insights into such a critical condition further helps promote the advancement of novel and accurate diagnostic tools for supporting reactor design and safety assessments within the nuclear industry. Furthermore, results of the high-fidelity simulations are to be gathered into a database and shared within the UK and international research communities, as reference data, to help evaluate the performance and facilitate improvements of the existing lower-order/fidelity simulation tools within the nuclear thermal hydraulics field of expertise.

1 https://www.code-saturne.org/cms/web/

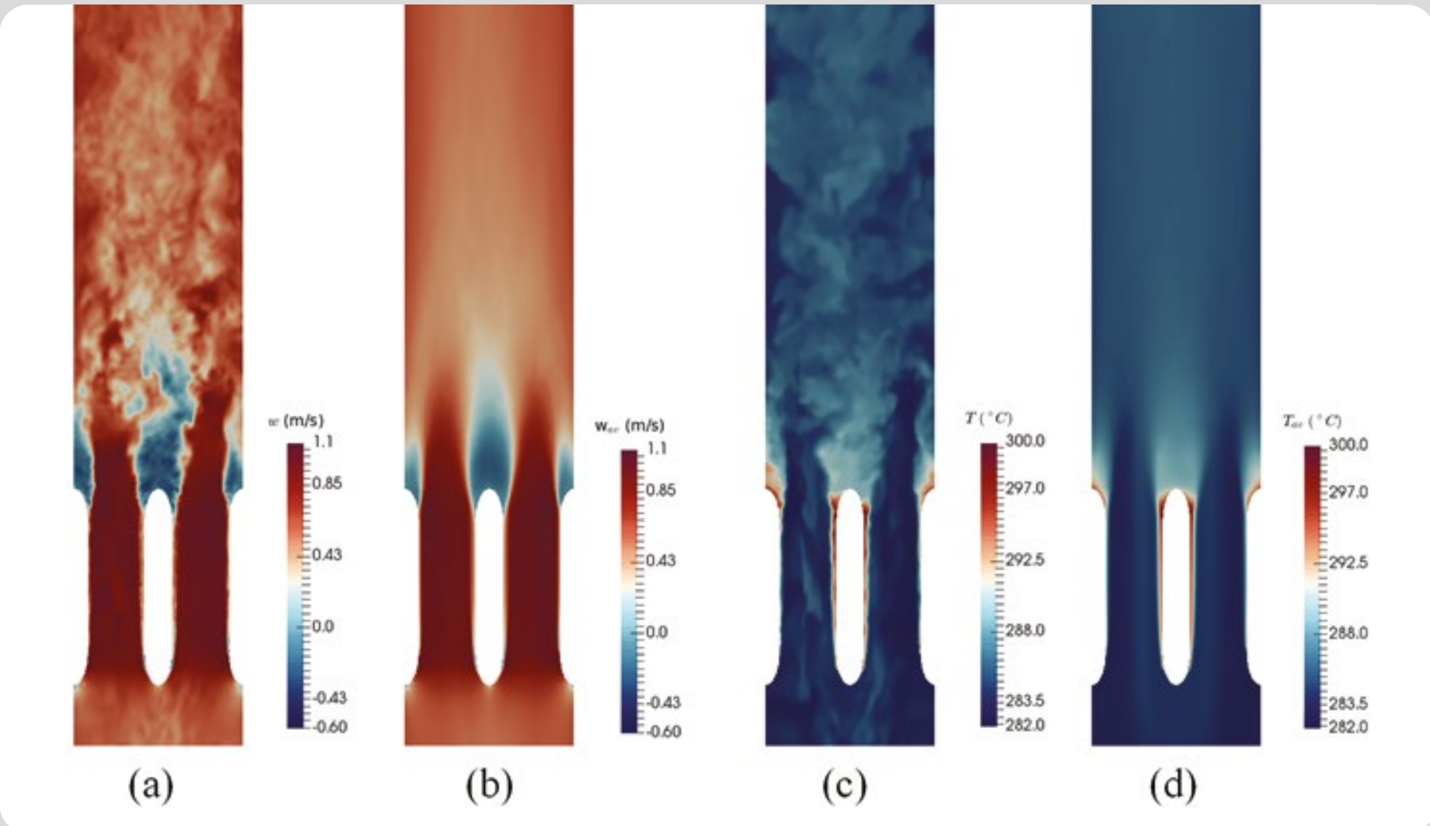
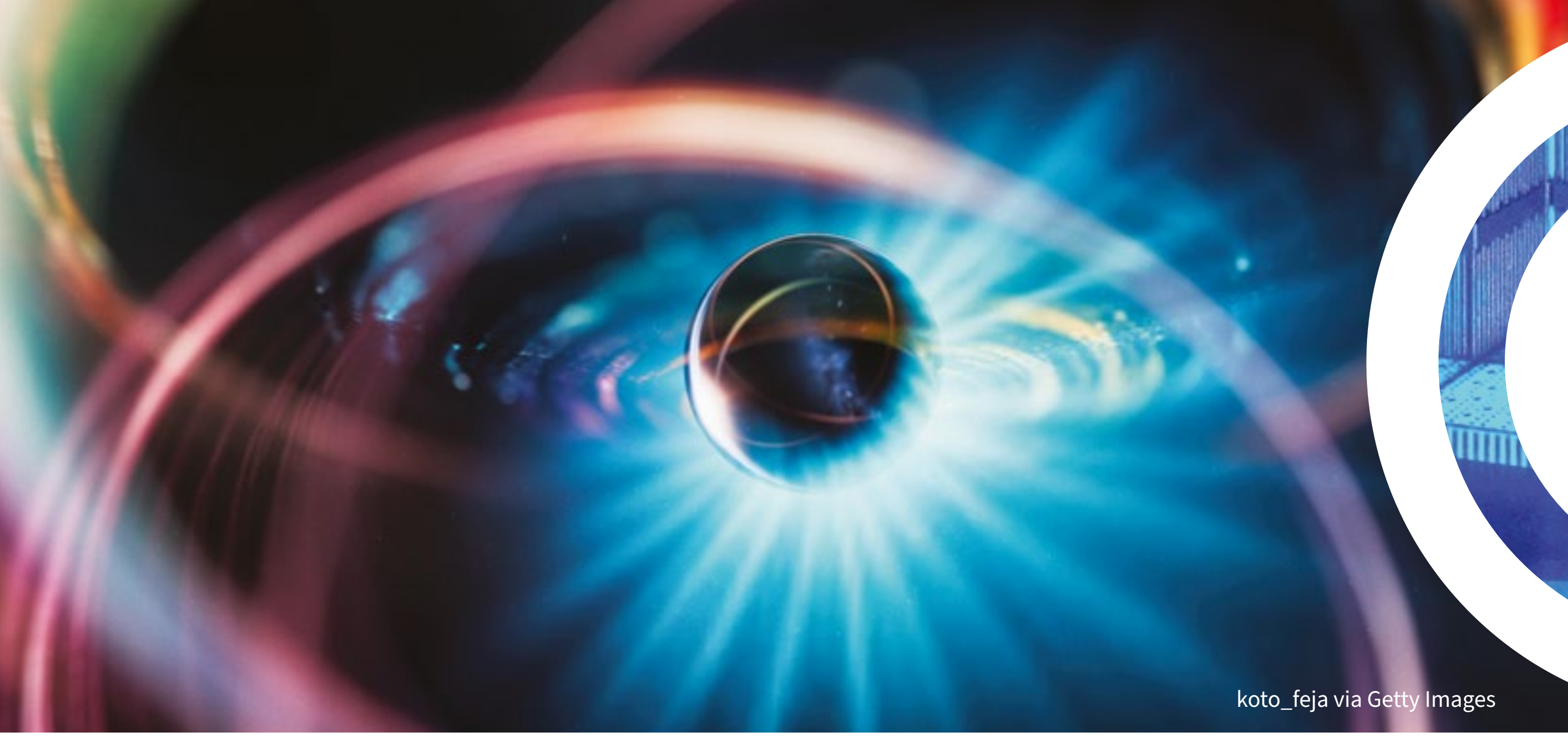


Figure 3: Snapshots of Large-Eddy Simulation results on the middle vertical plane of the central two sub-channels: (a) instantaneous velocity, (b) time-averaged velocity, (c) instantaneous temperature and (d) time-averaged temperature.


koto_feja via Getty Images

Acknowledgement:

ARCHER2 Pioneer Project e693

EPSRC Collaborative Computational Project for Nuclear Thermal Hydraulics (CCP-NTH EP/T026685/1)

Reference:

Bo Liu, Charles Moulinec, Wei Wang, Shuisheng He & Juan Uribe. A Numerical Investigation of Heat Transfer and Flow Redistribution in a 7x7 Ballooned Rod Bundle, in *Proceedings of 17th International Heat Transfer Conference*, Cape Town, South Africa, 2023. DOI: 10.1615/IHTC17.330-100

URL: **https://www.ihtcdigitallibrary.com/conferences/ihtc17,75da1d301b705895,7ef65ef324bfa7e5.html**

Authors:

Charles Moulinec (STFC Daresbury Laboratory)
Bo Liu (STFC Daresbury Laboratory)
Wei Wang (STFC Daresbury Laboratory)
Shuisheng He (University of Sheffield)
Juan Uribe (EDF R&D)

Contact Details:

PI: Charles Moulinec **charles.moulinec@stfc.ac.uk**
Bo Liu **bo.liu@stfc.ac.uk**

About ARCHER2

ARCHER2 is the UK's National Supercomputing Service, a world class advanced computing resource for UK researchers. ARCHER2 is provided by UKRI, EPCC, HPE and the University of Edinburgh. ARCHER2 is the latest in a series of National Supercomputing Services provided to UK researchers.

More ARCHER2 case studies can be found at:
https://www.archer2.ac.uk/research/case-studies/

This project was an **ARCHER2 Pioneer Project**, awarded by UKRI to conduct computationally intensive modelling, simulation and calculations to deliver ambitious and pioneering projects.