From: Kenneth Chinembiri, University of Sheffield, 11/6/19
Subject: Free course on modelling fluid flow and heat transfer in a nuclear power plant

Please find below detailed information pertaining to a free training course on the modelling of fluid flow and heat transfer using Computational Fluid Dynamics (CFD). The course will also serve as an introduction to the open-source CFD solver Code_Saturne.

Turbulence and heat transfer applied to HPC-related civil nuclear phenomena - Introduction to Code_Saturne

19-20 June 2019, Manchester

In a nuclear power plant, the prediction of fluid flow and heat transfer is of vital importance for the plant's performance and for safety compliance. This course will focus on the use of CFD for the prediction of fluid flow and heat transfer, including turbulence modelling, near wall modelling and conjugate heat transfer.

The course will run for 2 days comprising a mixture of lectures and tutorials for nuclear internal flows. The open-source HPC software Code_Saturne will be used by the participants to run large scale simulations using the UK national facility ARCHER.

The course is suitable for PhD students and engineers who have limited knowledge in HPC/Nuclear thermal hydraulics but are interested in gaining experience in this area.

The course is organised by the University of Manchester, University of Sheffield, EDF Energy, STFC Daresbury Laboratory and PRACE through EPCC, with the support of the UKFN SIG - Nuclear Thermal Hydraulics.

The course is free to attend, but spaces are limited. Please register your intent to attend through the following link:

https://events.prace-ri.eu/event/865/