11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

LARGE EDDY SIMULATION OF FLOW INSIDE THE LOW PRESSURE VESSEL OF AN ADVANCED GAS-COOLED REACTOR

Charles Moulinec[1,*], Juan Uribe[2] and David R. Emerson[3]

[1] STFC Daresbury Laboratory, charles.moulinec@stfc.ac.uk
[2] EDF Energy R&D UK Centre, juan.uribe@edfenergy.com
[3] STFC Daresbury Laboratory, david.emerson@stfc.ac.uk

Key words: *Advanced Gas-cooled Reactor, LES, Conjugate Heat Transfer, Code Coupling.*

The most common nuclear reactors used to generate electricity in the UK are the Advanced Gas-cooled reactors. Their life time extension is currently under investigation, in order to face the UK increasing energy consumption demand. Due to the complexity to monitor flow and heat transfer inside the reactors while in operation, the phenomena are not fully understood. HPC for CFD is a clear candidate to help in this matter. A quick description of how a reactor works is given first, before focusing on the simulations in an important part of the reactor, the vessel underneath the hot box dome.

High-pressure carbon dioxide gas at a relatively low temperature (of approximately $T_1 = 290^o$C) enters the reactor by fuel channels located at the bottom of the graphite core. The gas is heated as it flows upwards past the fuel and is discharged into the hot box at a temperature of approximately $T_2 = 650^o$C. It then circulates down through the boilers, and finally back to the bottom of the core.

The hot box is isolated from the graphite core by the hot box dome, which enables another flow path, the aim of which being to cool down the graphite core. The dome is insulated from the T_2 gas by insulation packs fitted above the dome. The main objective of the simulation is to be able to predict the steel temperatures which determine the performance of the reactor.

This work focuses on the region underneath the steel dome. The flow pattern inside the under-dome region is very complex since it traverses a series of tubes (fuel guide tubes, induction tubes and control rods) with different diameters and heat sources. The gas enters at the outer radius of the dome and moves through a tube forest towards outlets which are situated at the core boundary and at entrances to induction tubes which are located close to the dome surface to enhance cooling of the dome. The fuel guide tubes are arranged in a square pattern whereas the control rod tubes and induction tubes are placed in such a way that the only real geometric symmetry is achieved for half of the dome.

A first study has been made by Uribe *et al.* [1] when they computed an LES of the flow in a sector made of $1/8^{th}$ of the whole dome, imposing an artificial symmetry of the tube distribution. However, it appeared that the flow at the centre of the dome was not properly computed because of the periodic boundary conditions preventing from correctly predicting the flow pattern. The current work simulates the flow in the whole dome.

Two codes are coupled to compute the gas flow and temperature under the dome and the temperature in the solid body, which is made of the steel dome and the insulation packs. The codes used are Code_Saturne [2] and Syrthes [3], both open-source software primarily developed by EDF R&D. The flow is computed using the Smagorinsky Large-Eddy Simulation model [4]. The mesh used for the fluid part is made of 45 million cells, whereas the mesh for the solid part has 1.6 million elements. The coupled simulations are run on 16,384 cores of the Hartree Centre IBM Blue Gene/Q located at Daresbury Laboratory, and 32 of them are used by Syrthes.

Results of the unsteady LES will be presented together with the effects of the fluid flow on the steel dome temperatures. The results in the full domain will be compared to those obtained previously in the $1/8^{th}$ sector, which failed to capture the correct vortex evolution near the centre of the dome.

ACKNOWLEDGEMENTS

The authors would like thank the UK Engineering and Physical Sciences Research Council (EPSRC) for their support of Collaborative Computational Project 12 (CCP12) and the Hartree Centre for using their supercomputers.

REFERENCES

[1] J. Uribe, R. Howard, and M. Rabbitt. Fluid-solid coupling in advanced gas-cooled reactor thermohydraulics. In Turbulence Heat and Mass Transfer 7, Palermo, Italy, 2012.

[2] http://code-saturne.org/cms/

[3] http://researchers.edf.com/software/syrthes-44340.html

[4] J. Smagorinsky. General circulation experiments with the primitive equations: I The basic equations. Monthly Weather Review, vol. 91 pp. 99-164, 1963.