Multi-Scale Modeling of Very High Temperature Reactor Thermal-Fluids Using OpenFOAM

Ivor Clifford*, Kostadin Ivanov, Maria Avramova

The Pennsylvania State University
324 Reber Building, University Park, PA 16803, USA
ivor.clifford@psu.edu; kni1@psu.edu; mna109@psu.edu

March 19, 2011

Coarse mesh CFD, Multi-scale, VHTR, Porous media

Helium-cooled graphite-moderated Very High Temperature nuclear Reactors (VHTRs) are characterized by multiple spatial scales. Computational Fluid Dynamics (CFD) modeling of these reactors requires detailed modeling on all scales (millimeters to tens of meters) to accurately capture the behavior of the reactor. The resulting models are typically very large, requiring significant computational resources. Day-to-day time-dependent accident analysis of VHTRs using detailed CFD models is therefore currently impractical. Coarse mesh CFD methods show promise in this regard since they are computationally less expensive than finite mesh methods. They also offer several advantages over the traditional porous medium approach for the safety analysis of VHTRs, by opening up the possibility for system-wide calculations to be carried out using a consistent set of field equations throughout the calculation, and subsequently the possibility for hybrid coarse/fine mesh or hierarchical multi-scale CFD simulations. To date, a consistent methodology for hierarchical multi-scale CFD has not been developed [1].

In an effort to apply system-wide coarse mesh CFD simulations to VHTRs, a new solver is under development using the OpenFOAM multi-physics toolkit with the aim of modeling heat transfer in the reactor core on all scales. At the finest scale the fluid and solid components are explicitly modeled. As we progressively zoom out, the reactor is modeled as a homogenized two-phase mixture of fluid and stationary solid components, the parameters for which are determined by homogenization of the finer scale solutions.

Initial work has focused on developing a consistent two-phase fluid/stationary solid solver using OpenFOAM (See Figure 1). The modeling equations are based on the multi-phase flow equations of Saurel [2] modified for a fluid and stationary solid mixture. The shear stress tensor and thermal diffusion terms have been reintroduced so that the equations reduce to the single phase flow equations as the fluid fraction approaches one. Specialized pressure and mass flow interpolation schemes have been developed using a conservation of mass at the cell faces, which allows the solver to accurately capture discontinuities in the fluid fraction, pressure and velocity at porous interfaces. Further details of the fluid flow solver are given in [3].

To model the heat transfer between fluid and solid components several features have been developed. Heat transfer between fluid and solid components uses a mapping of values allowing
I. Clifford and K. Ivanov

separate meshes to be used for solid and fluid components. Specialized boundary types provide support for heat transfer from solid to porous regions, as well as from the surface of the solid component of a porous region to a neighboring fluid-filled cavity.

Meshing provides a particular challenge in this work, given the complexity and size of models. One particular problem for prismatic VHTR modeling is a need to model coolant flow in the hexagonal fuel elements as well as leak flows in thin gaps between the fuel elements. The coupling of the very thin gap elements to the remainder of the coarse mesh model without significantly reducing the allowable time step requires the use of hanging nodes on otherwise unstructured polyhedral meshes. Specialized tools are being developed using the SALOME platform [4] to assist in the mesh generation process.

While this initial work shows promise, many questions still need to be addressed. Future research will focus on topics including turbulent mixing and dispersion in porous regions including wall effects, as well as homogenization of fine mesh solutions and methodologies for reconstructing the fine scale solution.

**Figure 1. Coarse mesh CFD solution and mesh for buoyancy driven flow in a VHTR**

**REFERENCES**


