INL/EXT-06-11780

Computational Flow Predictions for the Lower Plenum of a High-Temperature, Gas-Cooled Reactor

Donna Post Guillen

November 2006

The INL is a U.S. Department of Energy National Laboratory
operated by Battelle Energy Alliance

INL/EXT-06-11780

Computational Flow Predictions for the Lower Plenum of a High-Temperature, Gas-Cooled Reactor

Donna Post Guillen

November 2006

Idaho National Laboratory
Idaho Falls, Idaho 83415

Prepared for the
U.S. Department of Energy
Office of Nuclear Energy
Under DOE Idaho Operations Office
Contract DE-AC07-05ID14517

Computational Flow Predictions for the Lower Plenum of a High-Temperature, Gas-Cooled Reactor

Donna Post Guillen

Idaho National Laboratory, P.O. Box 1625, Idaho Falls, ID 83415

INTRODUCTION

Advanced gas-cooled reactors offer the potential advantage of higher efficiency and enhanced safety over present day nuclear reactors. Accurate simulation models of these Generation IV reactors are necessary for design and licensing. One design under consideration by the Very High Temperature Reactor (VHTR) program is a modular, prismatic gas-cooled reactor. In this reactor, the lower plenum region may experience locally high temperatures that can adversely impact the plant's structural integrity. Since existing system analysis codes cannot capture the complex flow effects occurring in the lower plenum, computational fluid dynamics (CFD) codes are being employed to model these flows [1]. The goal of the present study is to validate the CFD calculations using experimental data.

PROBLEM DESCRIPTION

In the reactor core helium coolant flows downward over the fuel rods and enters the lower plenum, where it turns 90º and flows past circular, cylindrical support columns to the outlet duct. The outlet duct channels the hot gases to an intermediate heat exchanger, where the heat is transferred for electric power generation, hydrogen production or process heat utilization [2]. Designers are concerned whether hot jets impinging on the insulation layer on the floor of the core lower plenum, the graphite support posts, or the metallic components of the outlet duct can produce temperatures that exceed material capabilities.

Analysis of flow through the lower plenum of the prismatic reactor design must be able to handle complex geometries along with a wide range of operating temperatures, leading to significant variations of gas thermodynamic properties with possible buoyancy effects during normal and reduced power operations and loss-of-flow scenarios. The shortcoming of system analysis codes are their inability to accurately model 3D flow phenomena where turbulent mixing is driven by viscous and momentum force phenomena, including jet entrainment, eddy shedding, and wall shear. Due to the complexity of the flow channels in the lower plenum, CFD solutions validated by experimental data are required to model the turbulent mixing process.

EXPERIMENTS

Experimental data for a representative section of the lower plenum are being obtained using the world's largest Matched-Index-of-Refraction (MIR) flow system located at the Idaho National Laboratory. MIR uses optical techniques, including laser Doppler velocimetry (LDV) and particle image velocimetry (PIV) to obtain non-intrusive flow measurements. The data will be part of a benchmark database used to assess CFD predictions of the velocity and turbulence fields [3].

A flow test model was constructed using quartz columns and side walls, since quartz has the same index of refraction as the mineral oil used as the working fluid of the MIR system. The model consists of eight inlet jet ports above a symmetrical arrangement of five cylindrical columns along the centerline and ten half-columns along the two parallel side walls. The columns extend the full height of the model. A wedge-shaped element at one end simulates the hexagonal support block for the outer reflector.

During the experiments, the temperature of the mineral oil is precisely controlled and maintained at 23.3 °C. Unheated MIR experiments provide data for the baseline case of negligible buoyancy and constant fluid properties. Once such flows can be computed with confidence, the conjugate heat transfer problem including thermal mixing will be considered.

COMPUTATIONAL FLUID DYNAMIC RESULTS

The predictive capabilities of a commercial CFD code, FLUENT version 6.2.16, were assessed for the VHTR. The flow test model geometry was reproduced using GAMBIT and a computational mesh consisting of nearly 400,000 hexahedral cells was constructed. The mesh includes 4 ports for the inlet jets, numbered consecutively from right to left.

Results from the MIR experiments were used to generate boundary profiles for inlet jets 3 and 4 at the jet/plenum interface. Average x-, y-, and z-velocity profiles and the turbulent kinetic energy profile were specified at these locations. Inlet jets 1-2 and 5-8 were not used for this set of experiments.

FLUENT was run to convergence using the segregated solver with the k-ε turbulence model. Figure 1 depicts the computed velocity vectors. The red circles on the top of the figure depict the inlet jets and the rectangular region attached to the left side of the model represents the outlet duct. The flow enters the modeled section of the lower plenum through inlet jets 3 and 4 with a maximum velocity of 3 m/s. Recirculation regions form in front of the hexagonal support block and just downstream of the jets along the upper surface of the test section. CFD calculations were compared to the experimental data for the same geometry and flow conditions.

ACKNOWLEDGMENT

The MIR experimental data was provided by Hugh McIlroy of the Idaho National Laboratory. This manuscript has been authored by Battelle Energy Alliance, LLC under Contract No. DE-AC07-05ID14517 with the U.S. Department of Energy. References herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the U.S. Government, any agency thereof, or any company affiliated with Idaho National Laboratory.

REFERENCES

1. Very High Temperature Reactor – Design Methods Development and Validation Research & Development Program Plan," Idaho National Laboratory Report INL/EXT-04-02293, September 2005.

2. "Next Generation Nuclear Plant Project Preliminary Project Management Plan," Idaho National Laboratory Report INL/EXT-05-00952, November 2005.

3. Condie, K.G., McCreery, G.E., McIlroy, H.M., Jr., and McEligot, D.M., "Development of an Experiment for Measuring Flow Phenomena Occurring in a Lower Plenum for VHTR CFD Assessment," Idaho National Laboratory Report INL/EXT-05-00603, September 2005.

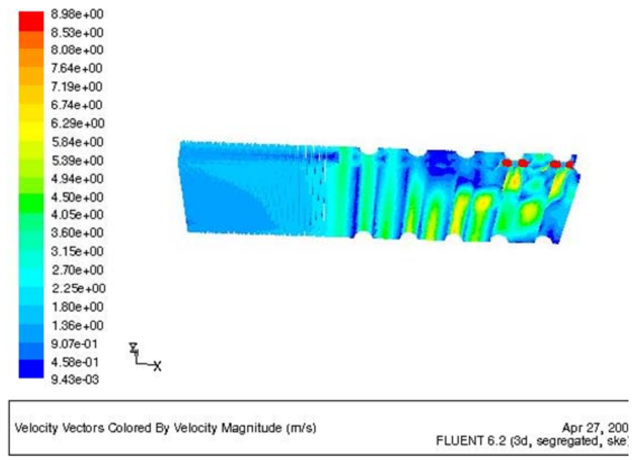

Figure 1. Computed velocity vectors.