

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

**ANS 2006 Winter Meeting**

Donna Post Guillen

November 2006

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



This is a preprint of a paper intended for publication in a journal or proceedings. Since changes may be made before publication, this preprint should not be cited or reproduced without permission of the author. This document was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, or any of their employees, makes any warranty, expressed or implied, or assumes any legal liability or responsibility for any third party's use, or the results of such use, of any information, apparatus, product or process disclosed in this report, or represents that its use by such third party would not infringe privately owned rights. The views expressed in this paper are not necessarily those of the United States Government or the sponsoring agency.

# 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 prismatic reactor design under consideration here, helium coolant flows downward over the fuel rods and enters the lower plenum, where it turns 90° 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 local temperatures that exceed material limits.

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 predictions validated by experimental data are required to model the turbulent mixing process.

## EXPERIMENTS

Experimental data for a representative section of the lower plenum are needed to enable validation of computational models. Benchmark data are being obtained using the world's largest Matched-Index-of-Refractive (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]. Results from the MIR experiments will be used to generate boundary profiles for the inlet jets. This data will include average and x-, y-, and z-velocities and turbulent kinetic energies over the inlet surface.

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. Figure 1 depicts the flow test model.



**Figure 1.** MIR test model.

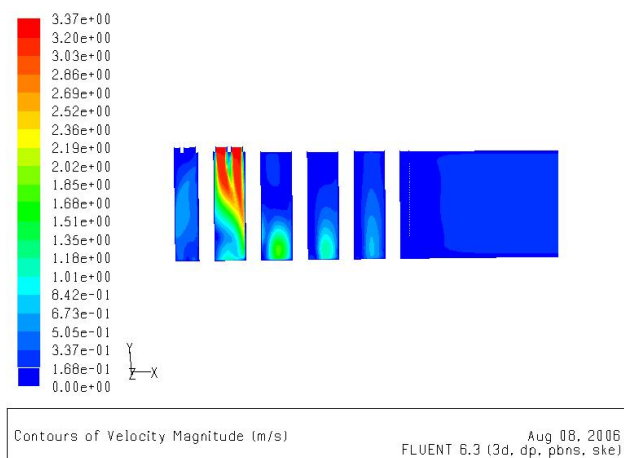
During the experiments, the temperature of the mineral oil is precisely controlled and maintained at 23.3 °C. Unheated MIR experiments provide data for the baseline case of negligible buoyancy and constant fluid properties. Scaling studies have been performed to ensure that the scaled model with mineral oil flow under isothermal conditions duplicates the pertinent non-dimensional parameters [4]. A systematic approach to validating computational predictions of turbulent mixing in the lower plenum is being implemented, wherein separate effects are examined before attempting to validate a much more complex, coupled problem using CFD. Once such flows can be computed with confidence, integrated effects will be examined.

## COMPUTATIONAL FLUID DYNAMIC RESULTS

The predictive capabilities of the commercial CFD code FLUENT were assessed for the VHTR. The flow test model geometry was reproduced and a computational mesh consisting of nearly 840,000 tetrahedral cells was constructed using Gridgen software. The tetrahedral cells were converted to approximately 690,000 polyhedral cells using FLUENT version 6.3.17. The computational model includes 4 ports for the inlet jets, numbered consecutively from right to left. Inlet jets 5-8 will not be used in the experiments.

Since the MIR data was not yet available, the FLUENT model assumed a constant velocity of 3.2 m/s of mineral oil flowing into jets 3 and 4. Since preliminary computations revealed backflow at the exit plane of the model, the computational model incorporates a 300 mm long rectangular shaped outlet duct downstream of the model exit. The feature was added to ensure that there was no backflow at the outlet, which was specified as a constant pressure boundary. The presence of backflow was observed in the laboratory via the upstream movement of air bubbles injected at the outlet of the test model.

Figure 2 depicts the computed velocity through the mid-plane of the model. Stagnation regions form in front of the hexagonal support block and just downstream of the jets along the upper surface of the test section. Quantitative comparisons of the CFD calculations to the experimental data will be performed once the experimental data is available.



**Figure 2.** Computed velocity through mid-plane of model.

## ACKNOWLEDGMENT

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.
4. McEligot, D.M. and McCreery, G.E., “Scaling Studies and Conceptual Experiment Designs for NGNP CFD Assessment,” Idaho National Laboratory Report INL/EXT-04-02502, 2004.