Experimental and CFD Studies of Coolant Flow Mixing Within Scaled Models of the Upper and Lower Plenum of a Prismatic Core VHTR

PI: Yassin A. Hassan – Texas A&M University

Collaborators: Richard R. Schultz- Idaho National Laboratory

Program: Mission Supporting Reactor Concepts RD&D

ABSTRACT

Background

The goal of this proposal is to investigate the fundamental physical phenomena associated with internal coolant flow in a prismatic core VHTR vessel during normal operation and under accident scenarios. Previous studies have revealed the importance of complex jet/plume flows in each plenum, with potential to generate recirculation zones that can lead to formation of hot spots within the lower plenum. It is therefore of interest to ensure that adequate mixing is promoted, but the complexity of the internal flow fields (characterized by structures spanning multiple orders of magnitude in time and length scales) makes rational design challenging. These difficulties are further compounded by limited availability of data for validation of predictive models.

Objectives

Here we propose to overcome these limitations by uniquely combining state-of-the-art experimental and computational capabilities of the project team. Experimentally, a geometrically scaled test facility incorporating a faithful 3D representation of the prismatic core VHTR vessel upper and lower plenum will be constructed to overcome difficulties encountered in previous attempts to capture the complex flow field using configurations incorporating a symmetry plane. This facility will be supplemented by the use of innovative high-speed high-resolution imaging capabilities that enable the multiscales of fluid motion (velocity and temperature) to be probed with unprecedented spatial and temporal resolution. Computationally, we will employ the use of high performance cluster computing to simulate the flows using advanced computational fluid dynamics (CFD) techniques to capture the velocity and temperature fields, both globally and locally in recirculation zones. Current modeling approaches typically rely on multi-scale averaging processes that impose serious constraints when unsteady phenomena must be captured. This limits the ability of commercial CFD codes to capture unsteady turbulence multiscales encountered here. We will deal with these complexities by employing large eddy simulation (LES) methods that enable large scales of motion to be directly calculated while finer scales are characterized by a physics-based model.

A distinguishing feature of this work will be the unique ability to perform direct cross-validation between experiment and simulation, enabling more accurate and rational prediction of the coolant flow field characteristics than is currently possible.