High-Resolution FSI Modeling of a High-Aspect Ratio Involute Flow Channel in the HFIR at ORNL

A. I. Elzawawy[1], J. D. Freels[2], F. G. Curtis[3]

[1]Vaughn College of Aeronautics and Technology, East Elmhurst, NY, USA
[2]Oak Ridge National Laboratory, Oak Ridge, TN, USA
[3]Oak Ridge National Laboratory, Oak Ridge, TN, USA; University of Tennessee, Knoxville, TN, USA

Abstract

Introduction: The objective of the present study is to investigate the fluid-structure interaction (FSI) of the high flow rate of cooling water with the fuel-plate structure of the High Flux Isotope Reactor (HFIR). This flow causes the involute-shaped fuel plates to deflect which in turn, changes the flow characteristics. This nonlinear feedback loop between the coolant and the fuel plate is the focus of the present simulation. To capture these dynamics between the coolant flow and the fuel plates, it is advantageous to use the Fluid-Structure Interaction interface in the COMSOL Multiphysics® software. The coupling of the Reynolds-Average Navier-Stokes (RANS) turbulent form of the NS and the structural-mechanics (SM) equations accurately predicts the HFIR plate deflection. These three dimensional (3D) simulations, which are based on both extreme and nominal channel dimensions, require high resolution meshing and ORNL high-capacity computing to obtain reliable results.

Use of COMSOL Multiphysics®: In this work, the COMSOL FSI module is used to simulate the interaction of the fuel plate and the water flow at isothermal fluid conditions. A computer-aided design (CAD) model of the involute channel is constructed entirely using COMSOL CAD-Import Module and Design Module shown in Figure-1. Both the segregated and fully coupled solvers are used in a multi-step solution process.

Results: An initial comparison between the segregated and the fully coupled solver shows similar results for the maximum deflection and its location on the stiff involute plate of the HFIR. This type of performance allows the mesh resolution to be increased while using a more economical segregated solver. The higher-resolution mesh has shown a larger plate deflection that encourages the implementation of further mesh refinement.
Conclusion: The FSI-segregated solver is used to simulate the water flow over a long and thin involute-shaped aluminum plate. A multi-step solution approach is constructed as shown in Figure-2. This strategy has improved the solution convergence rate and physics variables toward a mesh-converged solution. The maximum deflection, as shown by Figure-3, occurs at or close to the leading edge of the fuel plate in all cases. These exploratory and preliminary results indicate a maximum magnitude of the deflection to be approximately 7 mil (0.001 in), which is

approximately 14% of the nominal channel thickness of 50 mil.

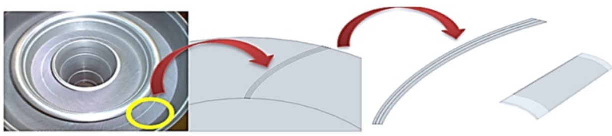
Reference

[1] Adam R., Travis, Simulating High Flux Isotope Reactor Core Thermal-Hydraulics via Interdimensional Model Coupling, Mater Thesis, University of Tennessee, 2014

[2] Vaibhav B., Khane et. al., COMSOL Simulation for Steady State Thermal Hydraulics Analyses of ORNL's High Flux Isotope Reactor, COMSOL Conference, Boston, 2012

[3] Franklin G., Curtis, Fluid-Structure Interaction for Coolant Flow in Research-type Nuclear Reactors, Proceedings of the 2011 COMSOL Conference in Boston, 2011

Figures used in the abstract



Involute Curve equations[1]: $x = r[\cos\theta + \theta * \sin\theta]$, $y = r[\sin\theta - \theta * \cos\theta]$

Figure 1: The investigated 3D COMSOL geometry representation of the HFIR flow channel and involute fuel plate.

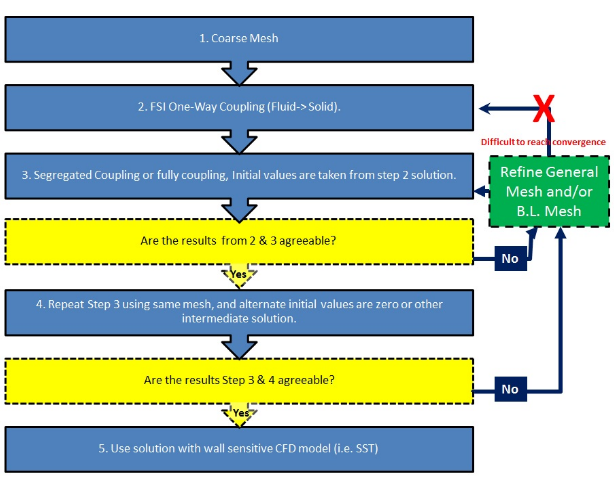


Figure 2: A graphical depiction of the multi-step solution approach toward mesh convergence.

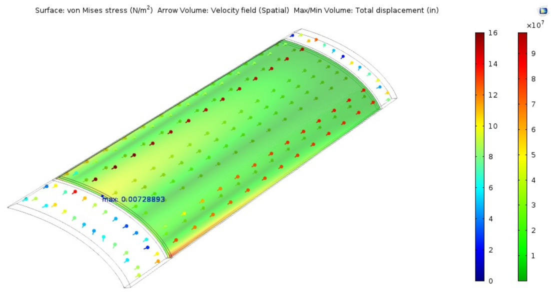


Figure 3: A typical FSI result from this investigation showing the stress of the plate surfaces (color contour), the value and location of maximum deflection in inches, and the colored arrows indicating the fluid flow magnitude and direction.

Figure 4