Development of CFD models to support LEU Conversion of ORNL's High Flux Isotope Reactor*

Vaibhav Khane[1], Prashant K. Jain[2*], James D. Freels[2]

[1]*143 Schrenk Hall, Missouri University of Science and Technology, Rolla, MO 65409-1230*
[2]*Oak Ridge National Laboratory, 1 Bethel Valley Road, Oak Ridge, TN-37831*
**Email: jainpk@ornl.gov, Phone: (865) 574 6272*

INTRODUCTION

The High Flux Isotope Reactor (HFIR) is a beryllium-reflected, light water cooled, highly enriched uranium (HEU)-fueled research reactor operating at 85 MW power at the Oak Ridge National Laboratory. HFIR is the highest flux reactor-based source of neutrons for research in the United States. Thermal and cold neutrons produced by HFIR are used to study physics, chemistry, materials science, engineering, and biology. The HFIR core assembly resides in an 8 ft diameter pressure vessel which is located in an 18 ft diameter cylindrical pool of water. The HFIR core has two fuel elements, the inner fuel element (IFE) and the outer fuel element (OFE), consisting of 171 and 369 involute fuel plates, respectively, for a total of 540 fuel plates. These involute-shaped fuel plates are uniformly spaced so as to provide equal coolant flow area for each plate within each element [1]. To remove the core heat, a highly turbulent water flow passes through involute-shaped coolant channels from the top to the bottom of the core. A total of 13,000 gal of water passes through the HFIR core every minute. Several physical phenomena—including turbulent flow, conjugate heat transfer, thermal-structure interaction, and fluid-structure interaction—are of significant interest when analyzing the thermal safety of the HFIR core.

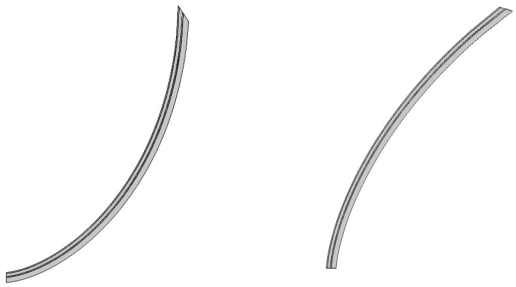
The US Department of Energy's National Nuclear Security Administration (NNSA) is participating in the Global Threat Reduction Initiative to reduce and protect vulnerable nuclear and radiological materials located at civilian sites worldwide. As an integral part of one of NNSA's subprograms, Reduced Enrichment for Research and Test Reactors, HFIR is being converted from the present HEU core to a low enriched uranium (LEU) core with less than 20% of U-235 by weight. Because of HFIR's importance for condensed matter research in the United States, its conversion to a high-density, U-Mo-based, LEU fuel should not significantly impact its existing performance. Furthermore, cost and availability considerations suggest making only minimal changes to the overall HFIR facility. Therefore, the goal of this conversion program is only to substitute LEU for the fuel type in the existing fuel plate design, retaining the same number of fuel plates, with the same physical dimensions, as in the current HFIR HEU core. Because LEU-specific testing and experiments will be limited, COMSOL Multiphysics was chosen to provide the needed simulation capability to validate against the HEU design data and previous calculations, and predict the performance of the proposed LEU fuel for design and safety analyses. To achieve it, advanced COMSOL-based multiphysics simulations, including computational fluid dynamics (CFD), are being developed to capture the turbulent flows and associated heat transfer in fine detail and to improve predictive accuracy [2].

HFIR MODELS

The developed models are being used to assess and quantify the impacts of uncertainties in the LEU fuel performance and associated fabrication tolerances on the overall thermal margin for HFIR. Furthermore, these models will be benchmarked against more detailed 3D COMSOL models currently under development. Several COMSOL-based multiphysics simulations are now being performed to assess the impacts of several parametric variations. Currently the effects of (a) geometric tolerances, (b) uncertainty in heat generation sources, and (c) neutron flux peaking on the effectiveness of core heat removal are being examined.

Because of the azimuthal symmetry of the HFIR fuel elements, only one fuel plate and one coolant channel of each HFIR fuel element (IFE and OFE) are modeled in three dimensions (called 1P1C—one-plate, one-channel models) (Fig. 1). The effects of other plates and channels in each element are modeled through periodic boundary conditions. In each fuel plate, the cladding is present in the top and bottom 2 in. and in two 10 mil (1 mil = 0.001 in.) thick layers sandwiching the 30 mil thick layer of LEU fuel, filler and poison materials (each HFIR fuel plate is 50 mil thick). Fuel meat is assumed to be smeared across the sandwiched region; hence, its radial and axial contouring is not captured explicitly in these models. Effects of the contouring are being accounted for through appropriate variations in volumetric heat sources.

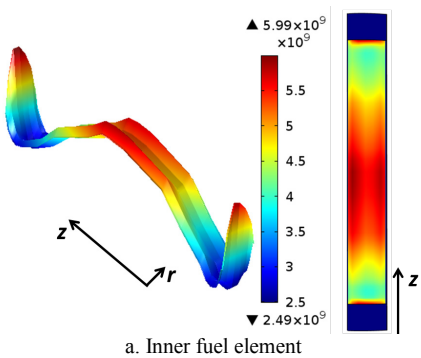
*This manuscript has been authored by UT-Battelle, LLC, under Contract No. DE-AC05-00OR22725 with the US Department of Energy. The US government retains and the publisher, by accepting the article for publication, acknowledges that the US government retains a nonexclusive, paid-up, irrevocable, worldwide license to publish or reproduce the published form of this manuscript, or allow others to do so, for US government purposes.



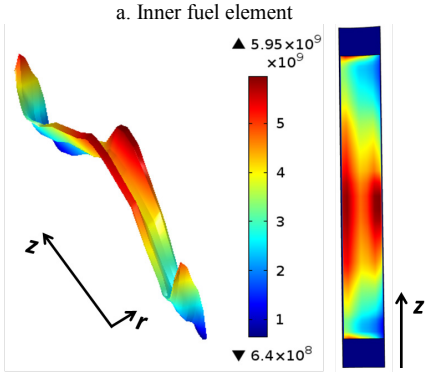
a. Inner fuel element b. Outer fuel element

Fig. 1. Cross-sectional views of HFIR's one-plate, one-channel models.

The volumetric heat source for the smeared fuel region is calculated from separate MCNP calculations for the axially non-contoured LEU fuel design at the beginning of cycle (Fig. 2).



a. Inner fuel element



b. Outer fuel element

Fig. 2. Power density profiles (in W/m^3) for 100 MW LEU at the beginning of cycle.

A standard, non-isothermal, k-ε turbulence model was used in the CFD models to simulate the turbulent flow of the coolant. Various boundary conditions assigned in the model are shown in Fig. 3.

The steady state inlet velocity and outlet pressure boundary conditions used in these CFD models were based on data from HFIR's design and operational experience (lower, more conservative pressures will be applied in the future for bounding safety analyses). Periodic heat conduction was applied on the convex and concave sides of the 1P1C model to capture the effect of other adjacent plates and coolant channels. A mapped mesh was generated based on the two-dimensional involute geometry of the fuel plate and channel and swept through the entire length of the fuel element. A boundary layer mesh for the coolant channel adjacent to the wall was also utilized in order to capture the steeper gradients there. A 10-element-thick boundary layer was used in the simulations with the first layer thickness equal to 0.005 mm (wall lift off in viscous units y^+~11.06). In addition to the boundary layers, 10 elements were used to further subdivide the coolant channel thickness and 60 elements in the direction of the involute span. Mesh refinement studies suggested this to be an optimum mesh providing a mesh-independent solution. COMSOL's direct PARDISO solver was used to solve partial differential equations associated with this simulation study. A relative-error convergence criterion of 10^{-3} was chosen for all the simulation variables [velocity components (u, v, w), pressure (P), temperature (T), and turbulence parameters (k and ε)].

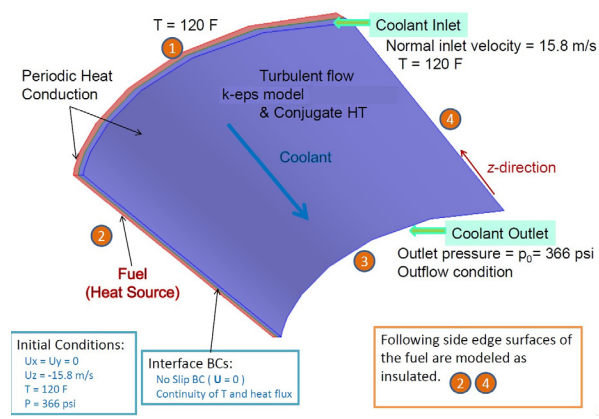


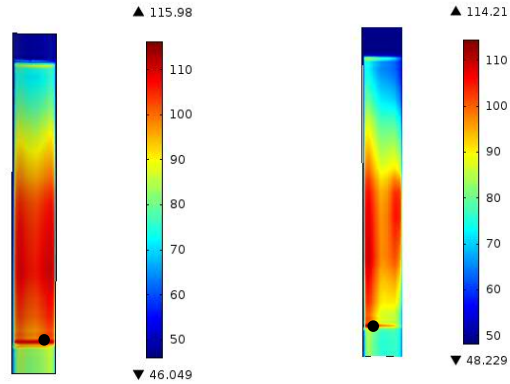
Fig. 3. Boundary and initial conditions for the one-plate, one-channel COMSOL model.

A design-basis thermal-hydraulic analysis of the 1P1C model for the inner as well as the outer fuel element was carried out at 100% FP at the beginning of cycle; results are provided in the next section. Note that the limiting control setting for the HFIR safety system is 1.3 times the normal operating power level [3,4]. Hence, thermal hydraulic analyses of the 1P1C model at 130% FP operation are also needed to estimate available thermal margins. This case was also simulated in COMSOL, and the results obtained are reported in the next section.

RESULTS

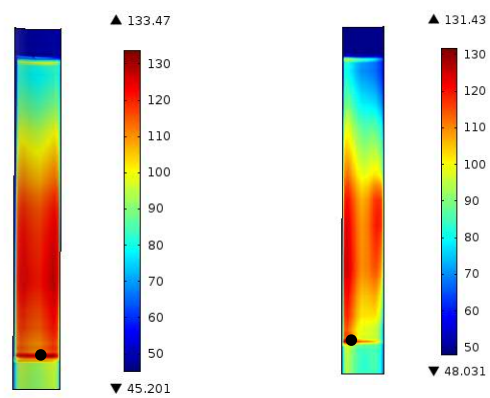
COMSOL surface temperature results obtained for 100% FP operation and 130% FP operation are presented in Figs. 4 and 5. The maximum clad surface temperature and its axial location on the plate, local pressure at that

location, corresponding saturation temperature T_{sat}, margin to Onset of Nucleate Boiling (ONB), net pressure drop, and average coolant exit temperature for the above two cases are tabulated in Tables I through III.



a. Inner fuel element b. Outer fuel element

Fig. 4. Surface temperature plots (in °C) at 100% FP (black 'dot' shows the location of peak clad temperature).



a. Inner fuel element b. Outer fuel element

Fig. 5. Surface temperature plots (in °C) at 130% FP (black 'dot' shows the location of peak clad temperature).

The maximum clad surface temperature for the IFE was found to be higher than that of the OFE. (Table I).

Table I. Peak Clad Surface Temperature and its Location

Cases	Max. Clad Surface Temp. (in °C) A		Axial Location of Max. Clad Surface Temp. (in inches from inlet)	
	IFE	OFE	IFE	OFE
100% FP	115.9	114.2	21.78	22
130% FP	133.4	131.4	21.8	21.8

The margin to ONB was calculated for each case by calculating local pressure at the point of maximum clad surface temperature, and finding the corresponding saturation temperature (T_{sat}). A superheat of about 5–10°C above the saturation temperature of water is required for ONB [3] to occur, which was taken into account when calculating the margins (Table II).

Table II. Saturation Temperature at Local Pressure and Margin to ONB

Cases	Local Pressure at Respective Axial Locations (in psig)		T_{sat} at Local Pressure B (in °C)		Margin to ONB $B+C^*-A$ (in °C)	
	IFE	OFE	IFE	OFE	IFE	OFE
100% FP	373	372.6	227.5	227.5	116.4	118.2
130% FP	372.9	373.1	227.5	227.5	99	100.9

*It is assumed that C = 5°C of superheat is required for ONB.

The average coolant exit temperature was found to be higher in the IFE than in the OFE for both cases (Table III). This is expected because of the IFE producing more heat per plate than the OFE. The net pressure drop predicted for the IFE and the OFE are in good agreement with each other.

Table III. Average Coolant Exit Temperature and Net Pressure Drop

Cases	Avg. Coolant Exit Temp. (in °C)		Overall Pressure Drop (ΔP, in psi)	
	IFE	OFE	IFE	OFE
100%F.P.	80.4	76.4	92.2	89.3
130%F.P.	89.6	84.5	90.9	88.2

Note that this is an ongoing project, and different designs of the LEU fuel are currently being analyzed; therefore, these results may change depending upon the chosen fuel design and associated heat source variation. Furthermore, work is underway to estimate the effects of geometric tolerances and buildup of oxide layer on the thermal safety of HFIR. The models presented here can be used to identify bounding limits on the uncertainties associated with various factors and quantify their effects on thermal margins.

ACKNOWLEDGEMENTS

The authors would like to acknowledge financial support provided for this work by the US Department of Energy National Nuclear Security Administration's Reduced Enrichment for Research and Test Reactors program.

REFERENCES

1. *High Flux Isotope Reactor (HFIR) User Guide,* NSUF revision 1, ORNL, October 2011.
2. J. D. FREELS and P. K. JAIN, "Multiphysics Simulations of the Complex 3D Geometry of the High Flux Isotope Reactor Fuel Elements Using COMSOL," COMSOL Conference, Boston, October 2011.
3. H. A. McLAIN, "HFIR Fuel Element Steady-State Heat Transfer Analysis," Revised Version, ORNL/TM-1904, Oak Ridge National Laboratory (December 1967).
4. *HFIR Safety Analysis Report*, ORNL/HFIR/ SAR/ 2344/Rev. 8, Oak Ridge National Laboratory, June 2011.