Franklin G. Curtis[1] · **Kivanc Ekici**[1] · **James D. Freels**[2]

Fluid-Structure Interaction Modeling of High-Aspect Ratio Nuclear Fuel Plates Using COMSOL

Abstract The High Flux Isotope Reactor at the Oak Ridge National Lab is in the research stage of converting its fuel from high-enriched uranium to low-enriched uranium. Due to different physical properties of the new fuel and changes to the internal fuel plate design, the current safety basis must be re-evaluated through rigorous computational analyses. One of the areas being explored is the fluid-structure interaction phenomenon due to the interaction of thin fuel plates (50 mils thickness) and the cooling fluid (water). Detailed computational fluid dynamics and fluid-structure interaction simulations have only recently become feasible due to improved numerical algorithms and advancements in computing technology. For many reasons including the already built-in fluid-structure interaction module, COMSOL has been chosen for this complex problem. COMSOL's ability to solve multiphysics problems using a fully-coupled and implicit solution algorithm is crucial in obtaining a stable and accurate solution. Our initial findings show that COMSOL can accurately model such problems due to its ability to closely couple the fluid dynamics and the structural dynamics problems.

Keywords fluid flow, structural dynamics, fluid-structure interactions, FSI, COMSOL, HFIR

1 Introduction

The High Flux Isotope Reactor (HFIR) core is approximately 2 ft. long and 15 in. in diameter. The fuel plates are placed in the core as two concentric fuel elements with the inner fuel element consisting of 171 fuel plates and the outer region consisting of 369 fuel plates. In order to maintain a constant coolant channel thickness, the plates are formed into an involute shape, which are shown in Figure 1 provided below. Both the fuel plates and coolant channels are nominally 0.050 in. in width.

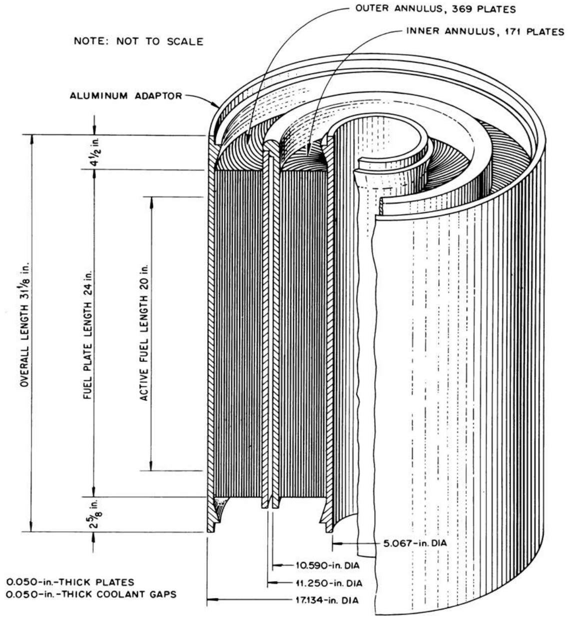

Fig. 1 Cutaway view of the HFIR core.

As the HFIR core undergoes design changes necessary for fuel conversion from highly-enriched uranium (HEU) to low-enriched uranium (LEU), it is necessary to evaluate the effect of these changes on the structural

---

Research funded by the Department of Energy (DOE) Office of Science and the Global Threat Reduction Initiative (GTRI) of the National Nuclear Security Administration (NNSA) http://nnsa.energy.gov/

[1] Mechanical, Aerospace and Biomedical Engineering Dept.
The University of Tennessee
Knoxville, TN 37996

[2] Oak Ridge National Laboratory
1 Bethel Valley Road
P.O. Box 2008
Oak Ridge, TN 37831-6392

E-mail: fcurtis@utk.edu · E-mail: ekici@utk.edu · E-mail: freelsjd@ornl.gov

response of the plates due to the high speed coolant flow between each plate. In the literature to date, significant efforts have been reported on predicting the flow velocity that would cause substantial deflections or even collapse of the reactor fuel plates. Some of the first experiments included preliminary research at the Oak Ridge National Lab (ORNL) for HFIR [1]. Miller [2] developed a theoretical model using potential flow theory and beam theory, and determined the fluid velocity that would lead to plate collapse. This analysis established the parallel reactor fuel plate safety basis known as the Miller Critical Velocity, M_c. This flow velocity was tested using experiments and many found the flow restriction to be conservative, with flows tested at twice the M_c without collapse [3–5].

As it became apparent that the M_c was too conservative, researchers began to explore other effects to better predict the collapse phenomenon of the fuel plates. The inclusion of friction and the redistribution of flow caused by plate deflections was incorporated into the velocity model by Johansson [6], while Kane [7] explored the effects of deviations from the design thickness of the plates. Time-dependency of the problem was considered in order to capture the instabilities caused by the dynamic movements of the plates [8–10].

In order to verify the theoretical background established, researchers have performed experiments on un-fueled flat plates to better understand the complicated physics that occur. Using an array of flat plates, Stromquist [1] and Doan [11] ran experiments studying the static deflections caused by the coolant flow. Dynamic deflections were explored by Groninger and Kane [3]. In another work, Smissaert [4] found that plate vibrations occur at approximately $2{\cdot}\mathrm{M}_c$. Fuel plate collapse has only been observed in one experiment, which used only two plates [12]. Later on, Li *et al.* [13] set up an experiment that used two plates, which were held fixed at only the four corners. This allowed them to demonstrate large, periodic deflections as an extension of a single plate experiment performed by Liu *et al.* [14].

There are a few experiments that utilized curved plates similar to those found in the HFIR. Ha and Garland [15] used curved plates following a circular arc. Swinson *et al.* [16] ran experiments for the design of the Advanced Neutron Source Reactor (ANSR) at ORNL, and measured deflections of involute fuel plates.

Until recently, the use of computer codes to simulate fluid-structure interactions (FSI) between the fuel plates and the coolant flow has been too computationally expensive. Roth [17] simulated the fluid flow between the fuel plates but was not able to model plate deflections. Kennedy *et al.* [5] have used two separate codes, one for solving the fluid domain and the other one for solving the structural domain. Their approach uses a time-dependent solver to complete the runs. The decoupling of the physics necessary to run two different codes results in an unstable solution process.

The conversion to LEU fuel is not only planned for the HFIR but across all US research reactors and thus experimental and numerical analyses are being performed to aid in the conversion process. A generic test plate experiment is being designed, constructed, and tested at the Oregon State University (OSU) that uses a flat plate, multi-channel design to measure both static and dynamic fuel plate deflections [18]. Single plate experiments are being designed and tested at the University of Missouri (MU) [5]. The goal of the research at ORNL is to establish a simulation technique that is validated against the current safety basis calculations used for the HEU fuel and will allow evaluation and safety analysis for the fuel designs for the LEU fuel. In order to validate the code and establish analysis techniques, the code will be compared to the existing experimental and numerical results available. The main validation effort will be on the current experimental work being performed for the LEU conversion projects currently underway at OSU and MU.

2 Governing Equations

The analysis of the reactor fuel plates utilizes the built-in FSI module available in the COMSOL software. The flow field, consisting of light-water, is considered to be incompressible and is governed by the Navier-Stokes (N-S) equations. The derivations of these equations has been well documented in literature. However, they are presented here for completeness. In differential form, the N-S equations are given by:

$$\rho \frac{D\mathbf{u}_f}{Dt} = -\nabla p + \mu \nabla^2 \mathbf{u}_f + \mathbf{F}_f \tag{1}$$

and

$$\nabla \cdot \mathbf{u}_f = 0 \tag{2}$$

where Eq. (1) represents the conservation of momentum and Eq. (2) represents the conservation of mass. The subscript, f, denotes the fluid domain. The majority of the flows simulated for the problems considered herein are turbulent, and thus the Reynolds Averaged Navier-Stokes (RANS) model is used. The turbulent viscosity is determined using the two equation $k-\epsilon$ closure model.

The structural mechanics is modeled using a linear elastic model with the following governing equations:

$$\mathbf{F}_s = -\nabla \cdot \sigma \tag{3}$$

$$\epsilon = \frac{1}{2}\left[\nabla \mathbf{u}_s + (\nabla \mathbf{u}_s)^2\right] \tag{4}$$

$$\sigma = C : \epsilon \tag{5}$$

where Eq. (3) represents the equation of motion, derived from Newton's Second Law, Eq. (4) represents

Excerpt from the Proceedings of the 2013 COMSOL Conference in Boston

the strain-displacement equation, and Eq. (5) represents the Constitutive equations, derived from the Hooke's Law. The subscript, s, denotes the solid domain. Finally, the fluid-solid interfaces are represented by the following boundary conditions:

$$\mathbf{u}_f = \frac{\partial \mathbf{u}_s}{\partial t} \tag{6}$$

$$\sigma \cdot \mathbf{n} = \mathit{\Gamma} \cdot \mathbf{n} \tag{7}$$

where $\mathit{\Gamma} = -p\mathbf{I} + \mu \left(\nabla \mathbf{u}_f + (\nabla \mathbf{u}_f)^T \right)$.

---

3 Methods

The ultimate goal of using COMSOL's FSI capability is to simulate both the LEU and HEU fuel plates for static and dynamic deflections. The HFIR core consists of 540 involute shaped fuel plates that are 50 mils thick with a coolant channel of the same thickness between each plate. No experiments have been performed on the fueled HFIR plates to examine the deflections caused by the high-speed coolant flow. Because of this, it is necessary to validate COMSOL using other experiments. Presently, a large, multi-plate FSI facility is being constructed and calibrated at OSU [18] and it is expected that the data generated from this experiment will be used to validate COMSOL. This data is not yet available but MU has provided deflection data for a single plate case for code validation [5].

3.1 Experimental Background

MU's experiments consist of a single flat, aluminum plate inside of a Perspex channel. Two different plate thicknesses are used in their experiment, 40 mils and 32 mils. The plate was offset in the coolant channel with one channel 80 mils and the other 100 mils thick; this was done to ensure that the plate deflected in a predictable manner. The plate and channel were 4.342 in. wide. The plate was 25.5 in. long with an inlet section of 7.5 in. and an outlet section on 3 in. The schematic of the flow test is provided in Figure 2. The plate deflections were measured using a laser instrument using the Perspex wall as a means to see into the experimental flow channels. This method differs from most other flat plate deflection experiments which use strain gages for measurements. The numerical model in COMSOL was designed to follow the setup used in the experiment.

3.2 Numerical Procedure

The high aspect ratio plates used in research reactors have proven to be difficult to solve numerically. Typically, in a loosely coupled approach, the fluid domain is solved first and the pressure information is then fed into the structural solver. The process is repeated until some pre-defined convergence criterion is achieved. This approach has been used with limited success by a few groups using a transient solver [5]. At the earlier stages of this research, this method was also attempted with COMSOL using the segregated solver feature. However, it has proven to be unstable in both attempts and the fully-coupled solver has been utilized in COMSOL in order to improve stability to great success.

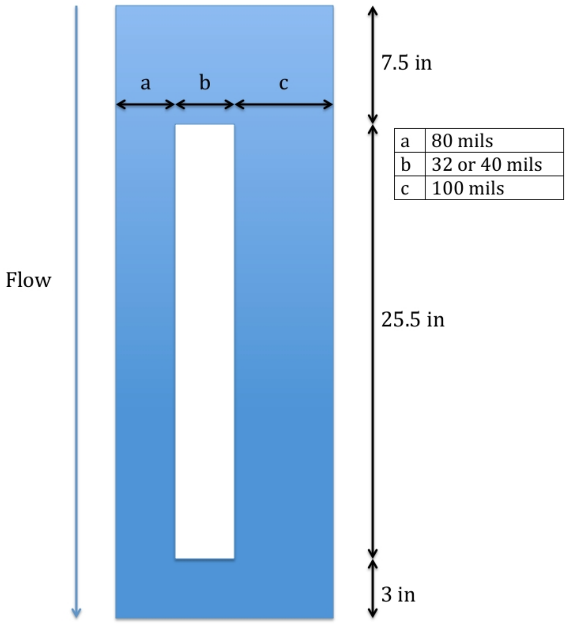

Fig. 2 The experiential and numerical setup of the MU flat plate tests.

One of the disadvantages of running a fully-coupled solver is the increased memory requirements needed to run a study to a mesh converged solution. With the increasing availability of large memory machines, this issue has become less important. Although the fully-coupled solver is more stable and large problems are no longer impossible due to memory, run-time is still an issue. In order to obtain a stable solution quickly, a three step process has been established and followed for all of the numerical runs performed. The procedure is given as follows:

Excerpt from the Proceedings of the 2013 COMSOL Conference in Boston

1. Obtain a course mesh solution using the one-way coupled FSI solver[a]
2. Use the course mesh solution obtained in the previous step as an initial condition for the fully-coupled solver
3. Obtain a mesh-converged solution by increasing the mesh density using the previous solution of the fully-coupled solver as the initial condition

The first step has proven to be crucial in establishing a good initial condition for the fully-coupled solver. The one-way coupling allows for the solution of the flow field, which is then used to determine the boundary conditions for the linear elastic model. Because the deflections are relatively small, the undeformed flow solution and deformed structural solution are then fed into the fully coupled solver in Step 2. This technique has allowed for the convergence of fully-coupled solutions in approximately 3-4 hours on a 12 core machine with 96 GB of memory. Without the first step, the solutions would either diverge very quickly or would take around 2 orders of magnitude longer to solve.

All of the cases performed for this paper were solved using the steady-state solver because the deflections observed in the MU experiments were all run at steady-state conditions. The higher inlet velocities (8-9 m/s) simulated using COMSOL initially needed artificial dissipation in order to converge with stability. This would provide deflections orders of magnitude smaller than those reported by Kennedy *et al.*. Therefore, a work around was found by implementing the pseudo-time stepping scheme. This technique, which essentially adds a temporal relaxation factor while still maintaining the benefits of the steady-state solver, is used widely in CFD to stabilize solvers. In order to retain the benefits of the pseudo-time stepping scheme, we enforced a limitation on the Courant–Friedrichs-Lewy (CFL) number used in COMSOL. During the solution process, the CFL number was held at 10 or below and this greatly improved the stability of the solver at higher inlet velocities. Steady-state solutions can be verified by allowing the CFL number to increase to 10^4 or higher.

As stated earlier, the computational domain was created to match the published experimental setup as closely as possible. The simulation was run in three-dimensions and a two-dimensional representation is presented in Figure 3. All of the boundaries, except the inlet and outlet were specified as wall boundaries for the wetted area; the plate is held fixed along the channel walls. The inlet boundary condition was set to be a velocity inlet and the outlet boundary condition was set as a zero-pressure boundary.

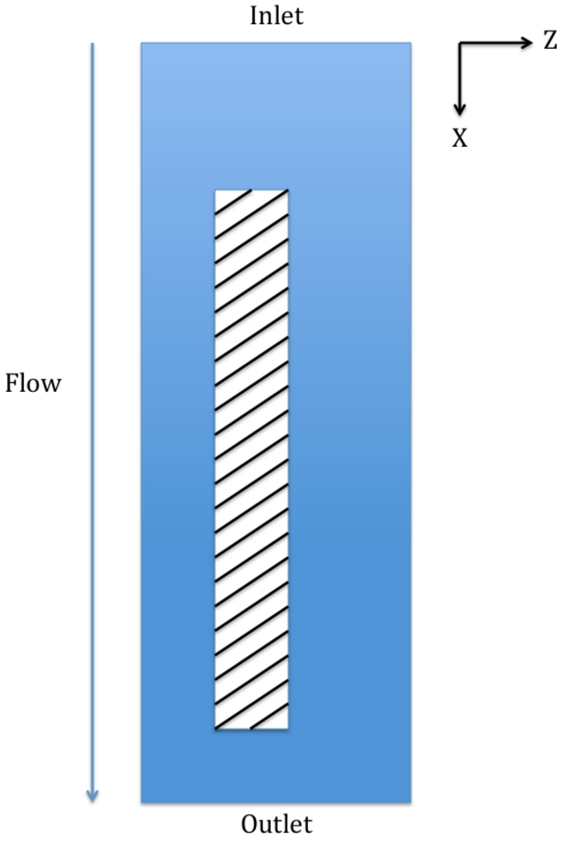

Fig. 3 The computational domain used for the COMSOL models. The Y-direction is into the paper.

4 Results

The results presented in this paper are for the case of the 40 mil thick aluminum plate without leading and trailing edge combs. All of the cases were perfomed using COMSOL v4.3a on a 12 core machine with 96GB RAM. The report provided by Kennedy *et al.* [5] presents the deflection for the leading edge of the plate using the laser measurement device.

A mesh convergence study has been performed for the present analysis. The results for the leading edge deflection for an inlet velocity of 6 m/s are presented in Figure 4. The final mesh consisted of 16 cuts in the span-wise direction and the boundary layer meshing feature was used. (The general nomenclature for these plates puts the span-wise direction perpendicular to the flow direction, that is along the width of the plate.) The total number of elements for the finest grid was 494,496. All of the subsequent simulations had similar mesh convergence characteristics resulting in a mesh converged solution.

As mentioned earlier, the high-aspect ratio of the plates creates a problem for a resolved boundary layer mesh. The geometry results in a solution that does

---

[a] One-way Coupling was introduced in COMSOL v4.3a

Excerpt from the Proceedings of the 2013 COMSOL Conference in Boston

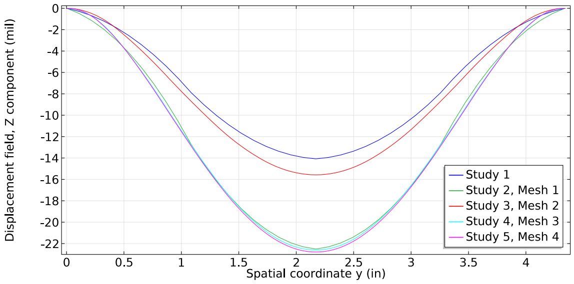

Fig. 4 Mesh convergence study for the inlet velocity of 6 m/s.

not have ideal y^+ values according to the literature. For the $k-\epsilon$ turbulence model, the ideal y^+ values are in the range of 30-100 when compared to available experimental data [19]. COMSOL suggests the value of y^+ to be 11.06 for the $k-\epsilon$ turbulence model on all boundaries and this was found to be the value along the length of the plate. In order to fall within the experimentally accepted range, only two mesh points could be used to raise the y^+ values above 30. Two mesh points in the flow field would not be sufficient to effectively capture the boundary layer flow so that it became necessary to use the finer mesh. The final mesh was created by using a free mesh on the side wall and then sweeping that mesh in the span-wise direction. The mesh at the leading edge is provided in Figure 5.

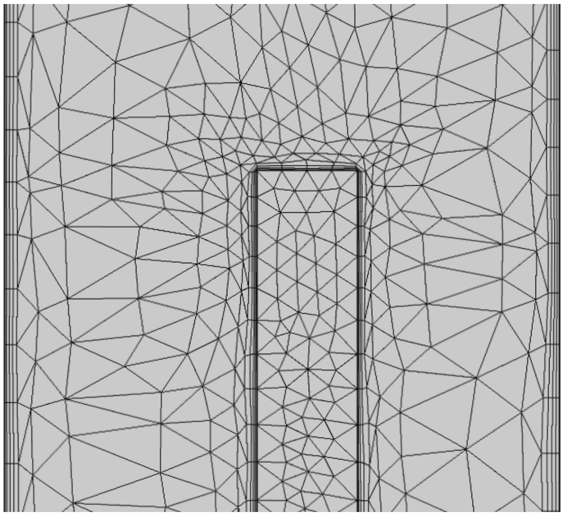

Fig. 5 The mesh at the leading edge as seen from the side wall of the plate.

Each case was run independently using the procedure described in the Methods section. Because the plates did not have leading edge combs, the largest deflections occurred at the leading edge of the plate in the mid-span location. In order to understand the effects of this deflection on the flow field, a contour plot, located at the leading edge and in the middle of the span of the plate, is presented in Figure 6.

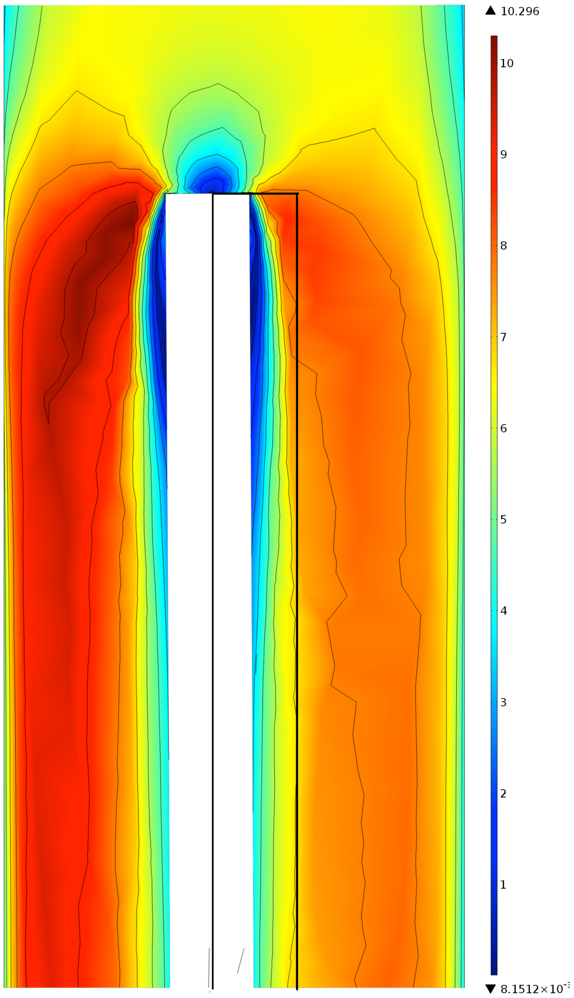

Fig. 6 Plot of the velocity field for the leading edge of the plate for the 6 m/s case. The cut is located in the center of the plate in the span-wise direction and the legend is in m/s.

As the plate begins to deflect toward the wall boundary, the channel velocity rapidly increases and careful inspection reveals some flow separation around the leading edge corners of the plate. The original plate location can be seen in the plot as the rectangular black line to the right of the white plate. The maximum deflection for this case is computed to be approximately 23 mils.

The linear elastic model used for the analysis is able to capture more than one mode shape. Multiple mode shapes can be seen in the experimental data provided by Smissaert [4] and others. A three-dimensional plot of the deflection for the 9 m/s case shows that multiple structural modes are present in the solution as shown

Excerpt from the Proceedings of the 2013 COMSOL Conference in Boston

in Figure 7 . Of particular interest is the second large deflection near the end of the plate.

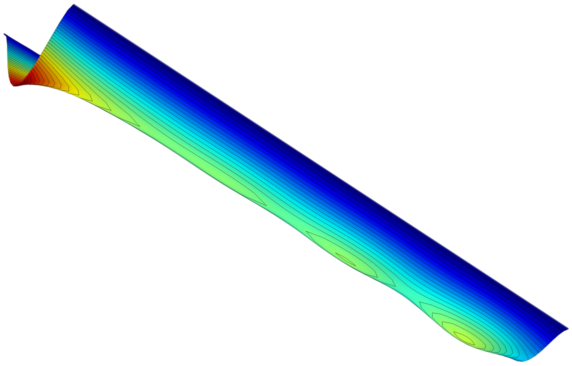

Fig. 7 Total deflection for the 9 m/s case.

The maximum leading edge deflection occurs at the center of the span-wise direction. This deflection was extracted from each case and is compared to the experimental results in Figure 8.

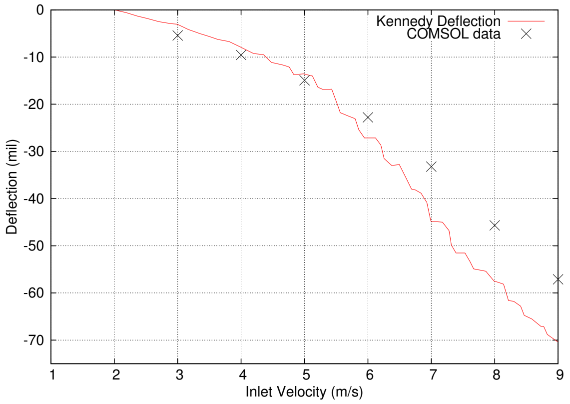

Fig. 8 Results for the comparison of the 40 mil thick plate.

As can be seen in Figure 8, the COMSOL code was able to match the experiment quite well. The code is conservative, that is, it predicts a slightly larger deflection than the experiment until an inlet flow velocity of approximately 5 m/s. This corresponds to the calculated M_c of approximately 4.5 m/s. This, according to Miller, is when the pressure difference between the top and bottom of the plate exceeds the restoration forces of the plate.

5 Conclusion

Accurate computation of the flow through high-aspect ratio plates typically used in research-type reactors such as HFIR requires the use of fully-coupled (fluid and structural) solvers in order to maintain stability in the solution process. A decoupling of the physics leads to an unstable and often incorrect solution. With the increasing availability of large shared memory machines, it is now possible to use a fully-coupled and implicit iteration solver to solve these problems using COMSOL. The choice of the numerical solution techniques play a vital role in convergence characteristics of the solver. The inclusion of the pseudo-time stepping technique has greatly increased the likely-hood of convergence for such complicated problems. By matching the experimental data provided by MU, COMSOL has demonstrated its potential to solve transient fully-coupled multiphysics problems. Future work, including the solution to deflections of involute plates, is expected to produce similar results as those presented in this paper.

References

1. Stromquist, W. K. and Sisman, O., "High Flux Reactor Fuel Assemblies Vibration and Water Flow," Tech. Rep. ORNL-50, ORNL, 1948.
2. Miller, D. R., "Critical Flow Velocities for Collapse of Reactor Parallel-Plate Fuel Assemblies," Tech. Rep. KAPL-1954, Knolls Atomic Power Lab, 1960.
3. Groninger, R. D. and Kane, J. J., "Flow Induced Deflections of Parallel Flat Plates," *Nuclear Science and Engineering*, Vol. 16, 1963, pp. 218–226.
4. Smissaert, G. S., "Static and Dynamic Hydroelastic Instabilities in MTR-type Fuel Elements Part 1. Introduction and Experimental Investigation," *Nuclear Engineering and Design*, Vol. 7, 1968, pp. 535–546.
5. Kennedy, J. C., Jesse, C. J., Slater, R. W., and Solbrekken, G. L., "Fluid-Structure Interaction Modeling and Experimental Benchmarking," Tech. Rep. TDR-MU-CALC-201210-005A, University of Missouri, 2012.
6. Johansson, E. B., "Hydraulic Instability of Reactor Parallel-Plate Fuel Assemblies," Tech. rep., General Electric Company, 1959.
7. Kane, J. J., "The Effect of Inlet Spacing Deviations on the Flow-Induced Deflections of Flat Plates," *Nuclear Science and Engineering*, Vol. 15, 1962, pp. 305–308.
8. Rosenberg, G. and Youngdahl, C., "A Simplified Dynamic Model for the Vibration Frequencies and Critical Coolant Flow Velocities for Reactor Parallel Plate Fuel Assemblies," *Nuclear Sci. and Eng.*, Vol. 13, 1962.
9. Schlösser, J., "On the stability of flat plates under the influence of coolant flow," *Journal of Nuclear Energy. Parts A/B. Reactor Science and Technology*, Vol. 16, No. 7, 1962, pp. 351 – 354.
10. Smissaert, G. E., "Static and Dynamic Hydroelastic Instabilities in MTR-type Fuel Elements Part 2. Theoretical Investigation and Discussion," *Nuclear Engineering and Design*, Vol. 9, 1969, pp. 105–122.
11. Doan, R. L., "The Engineering Test Reactor - A Status Report," *Nucleonics*, Vol. 16, 1958, pp. 102–105.
12. Ho, M., Hong, G., and Mack, A., "Experimental Investigation of Flow-induced Vibration in a Parallel Plate Reactor Fuel aAssembly," *15th Australasian Fluid Mechanics Conference*, 2004, pp. 13–17.
13. Li, Y., Lu, D., Zhang, P., and Liu, L., "Experimental Investigation on Fluid–structure Interaction Phenomenon Caused by the Flow Through Double-plate

Excerpt from the Proceedings of the 2013 COMSOL Conference in Boston

Structure in a Narrow Channel," *Nuclear Engineering and Design*, Vol. 248, 2012, pp. 66–71.

14. Liu, L., Lu, D., Li, Y., Zhang, P., and Niu, F., "Large-amplitude and narrow-band vibration phenomenon of a foursquare fix-supported flexible plate in a rigid narrow channel," *Nuclear Engineering and Design*, Vol. 241, No. 8, 2011, pp. 2874–2880.
15. Ha, T. and Garland, W. J., "Hydraulic Study of Turbulent Flow in MTR-type Nuclear Fuel Assembly," *Nuclear Engineering and Design*, Vol. 236, 2006, pp. 975–984.
16. Swinson, W., Battiste, R., Luttrell, L., and Yahr, G., "An Experimental Investigation of the Structural Response of Reactor Fuel Plates," *Experimental mechanics*, Vol. 35, No. 3, 1995, pp. 212–215.
17. Roth, G. D., *CFD Analysis of Pressure Differentials in a Plate-Type Fuel Assembly*, Master's thesis, Oregon State University, 2012.
18. Marcum, W., "Computational Fluid Dynamics Study of Flow and Pressure Differentials Surrounding the Generic Test Plate Assembly: Ideal Boundary Conditions," Tech. Rep. OSU-HMFTF-991000-CALC-001, Oregon State University, 2011.
19. Wilcox, D. C., *Turbulence Modeling for CFD*, Vol. 3, DCW Industries, 2006.

Excerpt from the Proceedings of the 2013 COMSOL Conference in Boston