Coupling of Fluid Dynamic and Structural Analysis of Rectangular Geometry Using ANSYS

A. A. Hassen

Nuclear fuel cycle Safety Department, Egyptian Nuclear and Radiological Regulatory Authority (ENRRA), Cairo, Egypt

Received: 4/9/2017 Accepted: 8/10/2017

ABSTRACT

Studying structural analysis of nuclear fuel plates in research reactors is very important in design and safe operation conditions. Finite element packages are recently used to solve multi-physics problems. In this paper, fluid dynamic analysis is carried out in a Fluid-Structure Interaction (FSI) problem. Pressure distribution is obtained around the plate. The analysis is carried out for different velocities. The resulted pressure is then used as an input in structural analysis study. The deformation of the plate for different inlet velocities is obtained and compared to the calculated analogous work as well as the experimental results.

Keywords: Plate Type Fuel / ANSYS / FSI / Structural Analysis

INTRODUCTION

Structural analysis studies of (Material Test Reactor) MTR plate type fuel are important and directly related to safety to avoid meltdown or structural damage of fuel. High coolant velocities are required to increase reactor power density which may lead to plastic deformation and structural failure of the plate (1).

Researchers used analytical and experimental techniques for assessing fuel plate deflection. Davis D. C. and Scarton H.A. (2) developed a methodology and criterion for predicting the plastic collapse of ETR fuel plates. The results were compared to the Miller critical velocity theory. The criterion resulted in the critical velocity depends on both the flow rate and the flow differential between adjacent channels.

Cekirge H. M. and Ural E investigated the response of parallel metal fuel elements to loads by the high coolant flow and computed the deflection analytically (3).

Pavon S. J. and Scarton H. A. (4) examined a single plate bounded by two channels and determined the static plate deflection as a function of the plate, channel and flow parameters. They used the classical thin plate equations with a nonlinear hydrodynamic loading function to express the external fluid forces on the plate surface. They concluded that it is necessary to distribute the flow as equally as possible in all channels to balance the viscous forces on the plate upper and lower surfaces. They found that the deflection results from linear plate theory are valid for small pressure and Reynolds number.

Kennedy J. C. et al (5) solved numerically the effect of coolant velocity on plate deflection. They coupled the CFD code, Star-CCM+, for solving fluid domain with ABAQUS code for solving structural domain.

Experimental work is performed at the University of Missouri on un-fueled flat plate to aid in the conversion to LEU (low enriched uranium) fuel in research reactors.
F. G. Curtis et al \(^{(6)}\) used COMSOL code to fully coupled multi-physics problem to solve the deflection of flat plate. They compared the results to the experimental results of University of Missouri.

Howard T.K. et al \(^{(7)}\) provides an approach to solving FSI problems using a 1-D, semi-analytical model. They compared the results to numerical and experimental work.

In this work, finite element code ANSYS is used to numerically solve multi-physics problem to study the effect of inlet coolant velocity on plate deflection in MTR reactors. The results are compared to ABAQUS code, COMSOL code, as well as MU experimental results.

## PLATE MODEL GEOMETRY

The fuel type model used in the University of Missouri experiments was a single flat plate made of aluminum with two thicknesses 0.8128 and 1.016 mm (32 and 40 mils) inside a 4.342 in wide Perspex channel.

The plate was 25.5 in. long and 4.392 in. wide. The dimensions selected for the plate are based on the dimensions of plate 23 from the new MURR (Missouri University Research Reactor), as this is the most extreme case in the design. MURR new assembly has curved fuel plates. In this paper, ANSYS model analyze a flat plate with the same dimensions determined previously to compare with other numerical and experimental data. The plate put in the coolant with channel gaps \((h_1\) and \(h_2\)) of 2.032 mm (80 mil) and 2.54 mm (100 mil) on either side as shown in Fig. (1).

\[ \begin{align*}
  a & = 1.016 \text{ mm (40 mil)} \\
  w & = 110.29 \text{ mm} \\
  h_1 & = 2.032 \text{ mm (80 mil)} \\
  h_2 & = 2.54 \text{ mm (100 mil)} \\
  L_{IN} & = 640.7 \text{ mm (25.5 in)} \\
  L_p & = 190.5 \text{ mm (7.5 in)} \\
  L_{OUT} & = 76.2 \text{ mm (3 in)} \\
  \dot{m} & = \text{ variable}
\end{align*} \]

Fig. (1): Simulated geometry

Where \(a\) and \(w\) are plate thickness and width respectively, and \(\dot{m}\) is inlet coolant flow rate.
FINITE ELEMENT SIMULATION

Multi-physics package ANSYS is used to solve the present case of FSI. The CFD analysis is conducted first, and then the calculated pressures at the fluid-structure interface (boundary of interaction where the results of one analysis are passed to the other analysis as loads) are applied as loads in the subsequent mechanical analysis (8).

The fluid flow problem is defined by the laws of conservation of mass, momentum, and energy. These laws are expressed in terms of partial differential equations (Navier – Stockes) which are discretized with a finite element based technique.

Assumptions about the fluid and the analysis model are as follows:
- There is only one phase of incompressible liquid water.
- Turbulent model (k-ԑ) is used.

The boundary conditions are:
- The side wetted walls of the water are considered as wall boundaries.
- The plate is considered fixed.
- Velocity inlet boundary condition is used.
- Atmospheric outlet pressure is used.

RESULTS AND DISCUSSION

The velocity increased from 3 m/sec to 8 m/sec in six cases modeled by ANSYS to solve the multi-physics problem. All the cases are performed using three dimension models for 40 mil (1.016 mm) thick aluminum plate with free leading edge. Sweep method used for element mesh. Leading edge pressure fields for the 7 m/sec case are shown in Fig. (2).

![Pressure fields for the 7 m/sec case](image)

Fig. (2): Pressure fields for the 7 m/sec case

The deflection of 7 m/sec case is shown in Fig. (3). As shown in the figure, the maximum deflection occurs at the center with 0.49 mm. The maximum deflection for each case is extracted and the results are plotted in Fig. (4) to show the effect of velocity increase on the leading edge deflection. For the purpose of code comparison the results are compared with Curtis F. G. et al (6) who used COMSOL code to fully couple multi-physics problem to solve the deflection of flat plate. The results
are also compared with Kennedy J. C. results\(^{(5)}\) who coupled the CFD code, Star-CCM+, for solving fluid domain with ABAQUS code for solving structural domain. Solving the multi-physics problem by ANSYS resulted in better results than ABAQUS results.

ANSYS results are slightly smaller than experimental results and COMSOL results\(^{(6)}\) in low velocities (3-5 m/sec) and the difference is enlarged for higher velocities. The large difference is due to low number of finite elements (29,365 elements in ANSYS model compared to 494,496 elements in COMSOL model). This number depends on the simulated model and the computer used. Core I7, 8 GB ram computer is used to simulate the model. More accurate results could be obtained using workstation or computer with faster processor.

### CONCLUSION

- Using multi-physics Finite Element Method (FEM) can produce more accurate results than decoupling physics. ANSYS get results better than ABAQUS results by solving multi-physics problem. The results are compared well against experimental results and COMSOL results at low and moderate velocity (3-5 m/sec) and the difference is enlarged with increasing the velocity due to low number of elements simulating the case study (29,365 elements).
- More accurate results could be obtained using higher number of elements although it needs a much faster computer processor.
- ANSYS gives under estimated results regarding to the experimental results.

![Deflection (in meters) for the 7 m/sec case](image)

**Fig. (3):** Deflection (in meters) for the 7 m/sec case
Fig. (4): Effect of velocity increase on the leading edge deflection.

**ACKNOWLEDGEMENT**

The author gratefully acknowledges the help and contributions made by Prof. Dr. K. Ahmed (Nuclear and Radiological Regulatory Authority, Egypt). Thanks are also given to Prof. Dr. M. Nasr for his important discussions.

**REFERENCES**

8. ANSYS workbench user manual, version 11, Providence RI, USA; (2007).