Nuclear Engineering and Design 333 (2018) 76–86

Contents lists available at ScienceDirect

Nuclear Engineering and Design

journal homepage: www.elsevier.com/locate/nucengdes

Numerical methodology for fluid-structure interaction analysis of nuclear fuel plates under axial flow conditions

Javier González Mantecón*, Miguel Mattar Neto

Instituto de Pesquisas Energéticas e Nucleares, IPEN-CNEN/SP, Av. Prof. Lineu Prestes, 2242 – Cidade Universitária, CEP 05508-000 São Paulo, SP, Brazil

ARTICLE INFO

Keywords:
Plate-type fuel element
Fluid-structure interaction
Critical velocity
Research reactor
Hydroelastic instability

ABSTRACT

Shell-type fuel elements are widely used in nuclear research reactors. The nuclear fuel is contained in parallel shells, flat or curved, that are separated by narrow channels through which the fluid flows to remove the heat generated by fission reactions. A major problem of this fuel assembly design is the hydraulic instability of the shells caused by the high flow velocities. The objective of the study presented here is the development of a fluid-structure interaction methodology to investigate numerically the onset of hydroelastic instability of flat-shell-type fuel elements, also known as plate-type fuel assemblies, under axial flow conditions. The system analyzed consists of two nuclear fuel plates bounded by three-equal coolant channels. It is developed using the commercial codes ANSYS CFX for modeling the fluid flow and ANSYS Mechanical to model the plates. The fluid-structure interaction methodology predicts a behavior consistent with other theoretical and experimental works. Particularly, the maximum deflection of the plates is detected at the leading edge and it is a linear function of the square of the fluid velocity up to the Miller's theoretical value. For velocities above this value, a nonlinear relationship is observed. This relationship indicates that structural changes are taking place in the plates. Furthermore, for fluid velocities greater than the Miller's velocity, an extra deflection peak is observed near the trailing edge of the plates. Thus, structural alterations also happen along the length of the flat-shells.

1. Introduction

Shell-type fuel elements are widely used in nuclear research reactors. The nuclear fuel is contained in parallel shells, flat or curved, that are separated by narrow channels through which the fluid flows to remove the heat generated by fission reactions. These shells have to be thin enough to prevent excessive internal temperature generation and yet strong enough to maintain stable configuration. A major problem of this fuel assembly design is the hydraulic instability of the shells caused by the high fluid velocities.

One of the first published reports regarding flow induced deflection in a shell-type fuel element was written by Stromquist and Sisman (1948). The study was performed in a mockup and it was found that when the coolant velocity was raised to sufficiently high values, fuel shells deformed plastically. In 1958, the same problem was detected by Ronald Doan during the fuel element design, development and construction for incorporation into the Engineering Test Reactor (ETR). Doan qualitatively discussed a critical flow field related to the onset of plastic deformation of the fuel shells (Doan, 1958). Later, Daniel Miller used the Doan's hypothesis and established a method to determine the maximum velocity that a series of shells, flat or curved, with fixed sides can sustain before collapse (Miller, 1958). This velocity is also known as "Miller's critical velocity". His analysis gives a basic idea of how the shells within a fuel assembly tend to deflect. Adjacent shells move in opposite directions at high flow rates, causing alternate closing and opening of the coolant channel. The main limitation of the Miller's method is that the 3D effects of the system are neglected. However, up to the present, it is applied during the design of new fuel elements.

Flat-shell-type fuel element, also known as plate-type fuel element, has been studied more than any other geometric shape due to its vulnerability to failure and its relatively weak structural capabilities with respect to other geometric forms (Marcum, 2014). The major part of the studies have been conducted by using analytical and experimental methods (Alvim de Castro, 2017; Cekirge and Ural, 1978; Davis and Scarton, 1985; Groninger and Kane, 1963; Guo and Païdoussis, 2000; Howard et al., 2015; Jensen and Marcum, 2014; Johansson, 1959; Kim and Davis, 1995; Li et al., 2012; Liu et al., 2011; Pavone and Scarton, 1983; Scavuzzo, 1965; Smissaert, 1969, 1968; Smith, 1968; Swinson et al., 1995; Swinson and Yahr, 1990; Weaver and Unny, 1970).

In the last years, the Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) techniques have demonstrated its feasibility and potential to solve many industrial issues. However, few

* Corresponding author.
E-mail address: javier.mantecon@ipen.br (J. González Mantecón).

https://doi.org/10.1016/j.nucengdes.2018.04.009
Received 6 November 2017; Received in revised form 9 February 2018; Accepted 9 April 2018

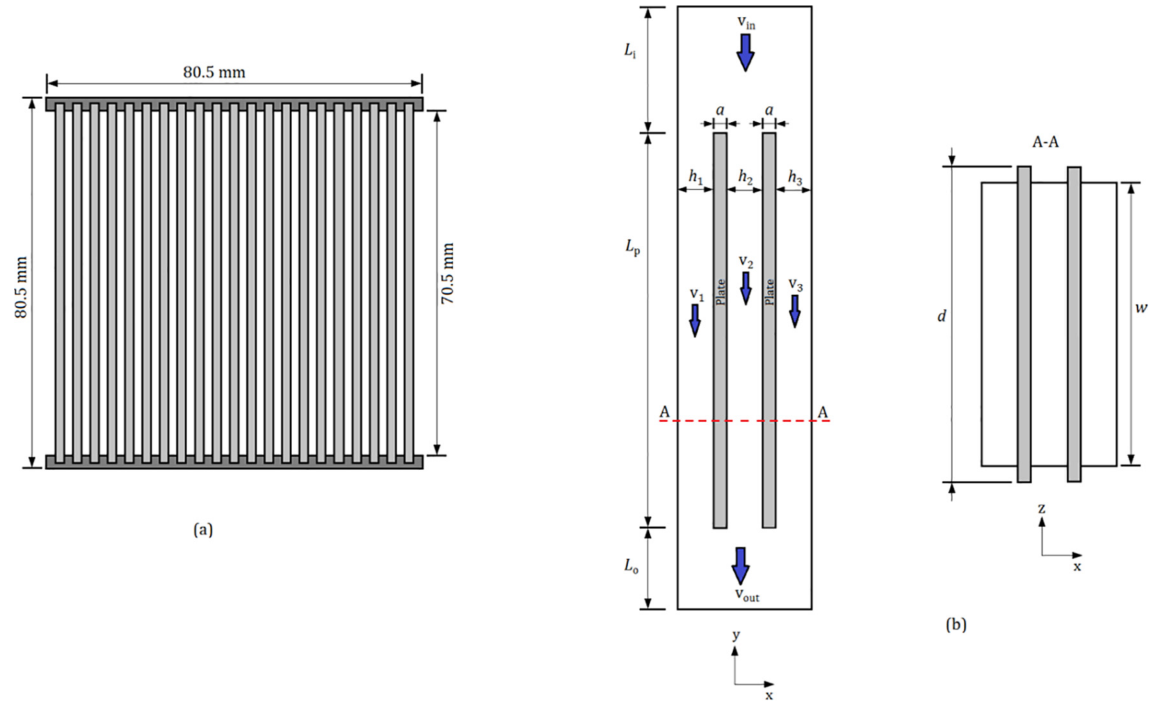

Fig. 1. Top view of the standard fuel element (a) and schematic diagram of the domain under consideration (b).

Table 1
Geometric specifications and material properties.

Parameter	Value
Coolant channel thickness, h [mm]	2.45
Plate thickness, a [mm]	1.35
Coolant channel width (wetted plate width), w [mm]	70.5
Plate width, d [mm]	75
Inlet length, L_i [mm]	190
Coolant channel (plate) length, L_p [mm]	655
Outlet length, L_o [mm]	70.0
Fluid properties (Water)	
Density, ρ [kg/m^3]	997.561
Dynamic viscosity, μ [Pa·s]	8.887e-4
Plate properties (Aluminum Alloy 6061-T6)	
Density, ρ_p [kg/m^3]	2700
Young's modulus, E [GPa]	68.9
Poisson's ratio, υ	0.33
Tensile yield strength, σ_y [MPa]	276

investigations concern fluid-structure interaction (FSI) problems in plate-type fuel assemblies by applying these techniques. After a deep search, the authors can list a reduced number of references with direct relevance to the subject discussed (Curtis et al., 2013; Jesse, 2015; Kennedy and Solbrekken, 2011; Kennedy, 2015). These studies were restricted to model a single plate with flow channels on either side. According to Smissaert, the presence of neighboring plates affects the lift force acting on the inlet edge (Smissaert, 1969). Therefore, in this work, we present a numerical analysis using more than one plate, which allows us to consider that effect.

The main objective of the study presented here is the development of a fluid-structure interaction methodology to investigate numerically the onset of hydroelastic instability of plate-type fuel elements under axial flow conditions. An additional purpose is to further the understanding of the behavior of the plates in fluid media and assess how its mechanical stability is related to the hydraulic loads. The FSI model consists of two fuel plates bounded by three coolant channels. It is developed using the commercial CFD code ANSYS CFX for modeling the fluid flow and the FEA code ANSYS Mechanical to model the plates. It is important to highlight that this methodology assists greatly in gaining insights of fluid-structure interaction problem in fuel elements of research reactors, allowing the analysis of complicated geometries under different conditions that would be costly to investigate experimentally. This methodology is important for updating the current analysis techniques applied to research reactor cores design.

2. Multi-field simulations

Fluid-structure interaction is a two field problem: one fluid flow and one structural field. On the one hand, the fluid field results in the load on the solid structure and is thereby a driving force. On the other hand, the structure reacts by means of stresses and deformations to the fluid field. The resulting deformation of the solid field has an impact on the fluid field that has to adapt to the modified boundary (Rao, 2003).

Solution strategies for FSI simulations are mainly divided into monolithic and partitioned methods; this work focuses only on partitioned methods. In the partitioned strategy, the fluid and structure subproblems are solved iteratively. This requires two codes: one for solving the fluid equations and the other for the structural part. This technique is further divided into two categories: one-way and two-way coupling (ANSYS Inc., 2017a).

The method used here is the two-way coupling between ANSYS CFX and ANSYS Mechanical (Transient Structural module). Fluid and solid domain/physical models are created in the ANSYS CFX-Pre and ANSYS Mechanical user interfaces, respectively. Coupling data transfers and controls are specified in the ANSYS CFX-Pre. Both have unlike meshing requirements and different meshes can be generated for the fluid field and the solid field. The meshes must not be equal at the interface but must consist of the identical geometric surface. During two-way coupling calculations, both solvers execute the simulation throughout a sequence of multi-field time steps, each of which consists of one or more coupling iterations. The FEA solver acts as a coupling master process to which the CFD solver connects. Once the connection is established, the solvers advance through a sequence of pre-defined synchronization points (SPs). At each of these SPs, information is exchanged between the solvers (ANSYS Inc., 2017a). This process of data sharing between structure and fluid field continues till the required conditions are satisfied.