13[th] International Conference on CANDU Fuel
Holiday-Inn Waterfront Hotel
Kingston, Ontario, Canada, 2016August 15-18

CANDU FUEL:
Evolution towards Optimal Performance

ANALYZING FUEL BUNDLE DEFORMATION

USING THE FINITE ELEMENT METHOD

R. Soni[1], P.K. Chan[1], A. Williams[2], and D. Wowk[3]
[1]Department of Chemistry and Chemical Engineering, Royal Military College of Canada, Kingston, Ontario, Canada
(Rabia.Soni@rmc.ca)
[2] Fuel and Fuel Channel Safety Branch, Canadian Nuclear Laboratories, Chalk River, Ontario, Canada
[3]Department of Mechanical Engineering, Royal Military College of Canada, Kingston, Ontario, Canada

ABSTRACT - Duringoff-normal or accident conditions, the temperature transient experienced by a fuel bundle may lead to deformation of the bundle via fuel element bowing between the spacer pads due to subchannel temperature gradients, or fuel element sagging under gravity. In these conditions, the deformation could impact the coolant flow distribution through the bundle, causing further degradation of the fuel cooling. The deformations of the fuel elements may also lead to fuel element contact with the pressure tube, leading to localised heating that may compromise the integrity of the pressure tube. The objective of this work is to create a three-dimensional finite element model to analyze bundle deformation behaviour under transient conditions. The preliminary results are presented herein.

1. Introduction

During off-normal or accident conditions, a fuel bundle may be subjected to thermal and mechanical loads when it is heated above its normal operating temperature. In these conditions, fuel elements may deform, either by bowing of the fuel elements due tochange in mechanical properties caused by thermal gradients or by sagging of the fuel elements under their self-weight[1]. The resulting deformation alters the coolant flow distribution through the bundle such that there is less flow between the sub-channels, and the coolantbypasses the bundle[2][3]. Due to reduced flow in the sub-channels, localized undercooling occurs. This exacerbates the cooling of the bundle and impedes heat transfer from the fuel elements to the coolant[4]. Hence, it is important to understand the conditions under which the fuel bundle will sufficiently deform such that the coolant flow distribution and the ability of the coolant to cool the fuel is adversely impacted.

A specific concern is the bowing deformation of a fuel element under dryout conditions. The thermal differentials across the fuel element may cause it to bow into contact with the pressure tube[1]. The thermal and mechanical loading on the pressure tube due to contact with the fuel element could compromise the integrity of both the fuel element and the pressure tube. The Canadian Nuclear Safety Commission (CNSC) has published a regulatory guide that details the trip parameters for the prevention of the onset of intermittent fuel sheath dryout[5]. However, there exists a potential to relax the trip criteria if it can be demonstrated that the fuel geometry and fuel sheath integrity are maintained underdryout or accident conditions.

The objective of this work is to develop a three-dimensional Finite Element Method model using ANSYS to predict the geometrical deformation of a CANDU fuel bundle in response to given thermal and mechanical loads that occur under conditions similar to those of dryout and a Large-Break Loss of Coolant Accident (LB-LOCA). A realized model would provide insight toevaluating the integrity of the fuel sheath geometryundertransient conditions and refining the safety analysis criteria for CANDU reactor licensing. The current regulations defined by the CNSC use the avoidance of fuel sheath dryout as a conservative surrogate criterion for fuel and pressure tube failure[5]. If it can be conclusively demonstrated that the fuel geometry and sheath integrity can be maintained under transient conditions, the trip parameters can be relaxed such that CANDUreactors can be operated at higher temperatures for a greater yield of power[6]. Increasing the amount of power that can be yielded safely would prove economically beneficial for nuclear power plants.

2. Model Development

Of the number of different commercial finite element analysis software platforms that exist, ANSYS was chosen to model fuel bundle deformation behaviour for its multiphysics simulation capabilities. ANSYS provides more robust contact algorithms that are of greater relevance to this work[7].For this project, contact modelling is a necessity as the deflected fuel elements will come into contact with surrounding elements via spacer pads, and with the pressure tube via bearing pads. The contact definition transfers the loading from the sheath onto the pressure tube and affects the behaviour of the fuel bundle. Furthermore, the software has the ability to couple multiple physical phenomena, such as solid mechanics, fluid dynamics, and heat transfer using the FEM to describe the mathematical models[8]. This multiphysics coupling ability is particularly advantageous as the results from the thermal analysis can be used as input for the structural analysis and the deformation resulting from the thermal load can be calculated.

The modelincludes three major components: the fuel bundle with appendages, the end-plates, and the pressure tube. Given that the geometry of the bundle is extremely detailed, the pressure tube will be modelled as a fixed boundary condition. The application of this boundary indicates that the pressure tube will not undergo deformation, allowing the focus of the model to be placed upon the interaction between fuel elements and between fuel elements and the pressure tube in response to the thermal load. Although there are many factors that affect the deformation behaviour of a bundle under transient conditions, the model will include selected factors, such as thermal expansion of the sheath, theinteraction between adjacent fuel elements, structural effects of the end-plates, and interaction between the outer fuel elements and pressure tube. The scope of the model will not include the temperature distribution of the fuel pellets, radiative heat transfer, and interaction between the fuel pellets and the fuel sheath [9].

2.1 Physical Geometry

The objective of this work is to model the deformation behaviour of the entire CANDU bundle, which comprises of 37 fuel elements, two end plates, and the encasing pressure tube. Brazed appendages, such as spacer pads and bearing pads, were included because it is expected that the fuel elements apply mechanical loads to each other directly through contact between spacer pads and indirectly through weld contact at the endplates due to bowing [1]. Given that the deformation results from the out-of-reactor experiment against which the model is intended to be validated are asymmetrical, planes of symmetry will not be used in this model.

As there are a number of physical phenomena that need to be modelled, it is necessary to simplify the geometry of the model before the asymmetric temperature profile and structural phenomena are added as loading conditions for the model. Simplifying the geometry serves a two-fold advantage as it reduces computational time and enables debugging at a smaller scale as problems can be resolved locally rather than globally for multiple fuel elements. For the initial models, the geometry was simplified to one fuel element with brazed appendages. The geometry for the initial model is illustrated in Figure 1.

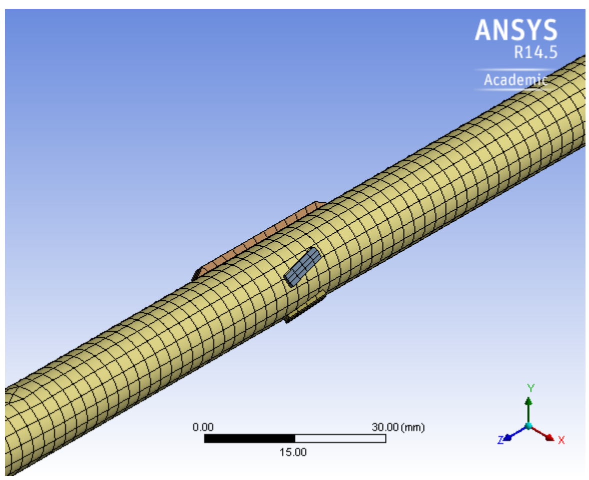

Figure 1: Fuel Sheath with Brazed Appendages.

The subsequent models will include the end-plates welded onto the sheath and the pressure tube to ensure that the model is appropriately constrained and does not undergo rigid body motion. Moreover, adding the endplates and pressure tube would result in a more realistic model because it removes the displacement constraints added to the models to achieve convergence.

2.2 Material Properties

To determine the integrity of the fuel sheath accurately, predicting its mechanical response is the crucial first step. The stresses and strains induced by the bundle due to thermal loading are determined by a number of properties of the fuel sheath material, Zircaloy-4. Fuel sheath material properties include the elastic modulus, thermal conductivity, plasticity curves, creep rate and thermal strain of the material at different temperatures[10]. These properties need to be entered manually in ANSYS Engineering Data to define the behaviour of Zircaloy-4 under the thermal loading conditions specified in the out-of-reactor experiment. As fuel pellets will not be modelled, the material properties and physical phenomena associated with the fuel pellets are excluded.

2.3 Thermal Analysis

Modelling the deformation of a fuel bundle necessitates the input of an accurate temperature profile along the length of the bundle. The non-uniform thermal distribution was imported from an out-of-reactor experiment conducted by the Canadian Nuclear Laboratories (CNL) [11]. In addition to the temperature profile, fuel sheath material properties, such as thermal conductivity and coefficient of thermal expansion,need to be added to the model.

The asymmetric temperature distribution to be modelled in this work was obtained from an out-of-reactor bundle heat-up experiment. The experiment consisted of heating an empty instrumented 37-element fuel bundle in a transparent quartz tube, which was contained within a heated chamber with quartz windows, allowing for optical view along the sides of the bundle. The aim of the bundle tests was to evaluate the deformation of the outer fuel elements once the sheath temperature had reached at least 800 ºC for 100 s [11]. Equipped with thermocouples, thermal imaging data was acquired throughout the duration of the experiment and the maximum deformation for the bundle at the highest temperature was recorded. The temperature distribution across the length of the bundle that correlated with sheath temperatures of at least 800 ºC was selected as the input variable for modelling the bundle deformation as it eliminates the need to model physical phenomena, such as convection by directly providing the thermal loading necessary for the structural analysis. Additionally, it is beneficial to use the temperature profile from the experiment as the deformation results from the model can be validated against the results obtained from the out-of-reactor experiment.

The thermal imaging acquisition software employed by CNL recorded real-time temperature and deformation data of the fuel bundle at certain locations [11]. Data from the sheath thermocouples, as well as values from the surrounding gas thermocouples interspersed through the bundle, were used to create the location and time dependent temperature boundary conditions required by the ANSYS model.

2.4 Structural Analysis

Using the results from the thermal analysis, a structural analysis was conducted to compute the deformation resulting from the thermal load. This structural analysis provides the mechanical response of the fuel bundle, which will be used to ascertain the structural integrity of the bundle by assessing the overall deformation and the stresses induced within the bundle. The total strain in the fuel elements is the sum of strain caused by elastic behaviour, plastic behaviour, thermal expansion and creep of the Zircaloy-4fuel sheath material.[10]. As the material properties are temperature dependent, the impact on bundle deformation under post-dryout conditions is substantial [10].

In addition to accounting for complex material properties, it is crucial to apply the correct loading and boundary conditions to simulate the experimental conditions, and to ensure rigid body motion does not occur. Rigid body motion describes the idealization wherein the deformation between two given nodes of the body remains the same, regardless of the external loading conditions applied, as the model is under-constrained[9]. For the single fuel element models, the deflection of the fuel element is constrained in all directions at the upstream end, and in the X and Y directions at the downstream end. Specifying these boundary conditions ensure that the fuel element is free to expand axially due to the thermal load applied and does not undergo rigid body motion. For the contact model, the elements are constrained through the contact of the bearing pads with the pressure tube. These contacts assume that the bearing pads remain in contact with the pressure tube, but allow for frictional sliding between the contacting surfaces.

Contact is expected to occur between the spacer pads of the adjacent fuel elements, and between bearing pads of the outer fuel elements and the pressure tube. ‘No separation’ contact mechanism was used to specify the contact between the bearing pads and the pressure tube. This contact definition meant that the bearing pads of the two bottom outer elements and the pressure tube remain in contact throughout all loading steps. The contact between the spacer pads of the adjacent fuel elements was defined to be frictional to imitate real life conditions as the spacer pads of adjacent fuel elements can come into and out of contact with the application of the thermal load.

The pressure tube that holds the bundle will be fixed in space. This ensures that rigid body motion of fuel elements does not occur because of the contact definition between the bearing pads of two outer elements and the pressure tube. For the preliminary models, the fuel elements were assumed to be in a vacuum and only a uniform thermal load was applied; however, mechanical loads due to the effects of gravity and creep will be added to the model at a later stage of development to simulatethe deformation behaviour of the modelmore accurately.

3. Preliminary Progress Results

In the first phase of this work, three preliminary models were created to test the various parameters that will be added to the final model. In all three cases, the geometry was simplified to reduce computation time and troubleshoot the software at a local level rather than globally for multiple fuel elements. All models used a Dirichlet boundary condition of 850 ºC and accounted for basal plane symmetry in evaluating thermal strain of the empty fuel elements [10]. A more physically meaningful representation of the thermal loads in the future

The first preliminary model compared the effects of using solid and shell elements to model fuel elements on the deformation profile of the fuel bundle. The hollow Zircaloy-4 fuel sheath can be modelled as either a solid or a shell. A one-way coupled thermal and structural analysis was conducted to compare the effects solid and shell elements had on the deformation results of a fuel sheath without bearing and spacer pads added. As shown in Figure 2 below, the results indicated that the sheaths of the fuel rods heated to 850 °C deformed to the same extent in the positive and negative X-direction for both solid and shell elements.

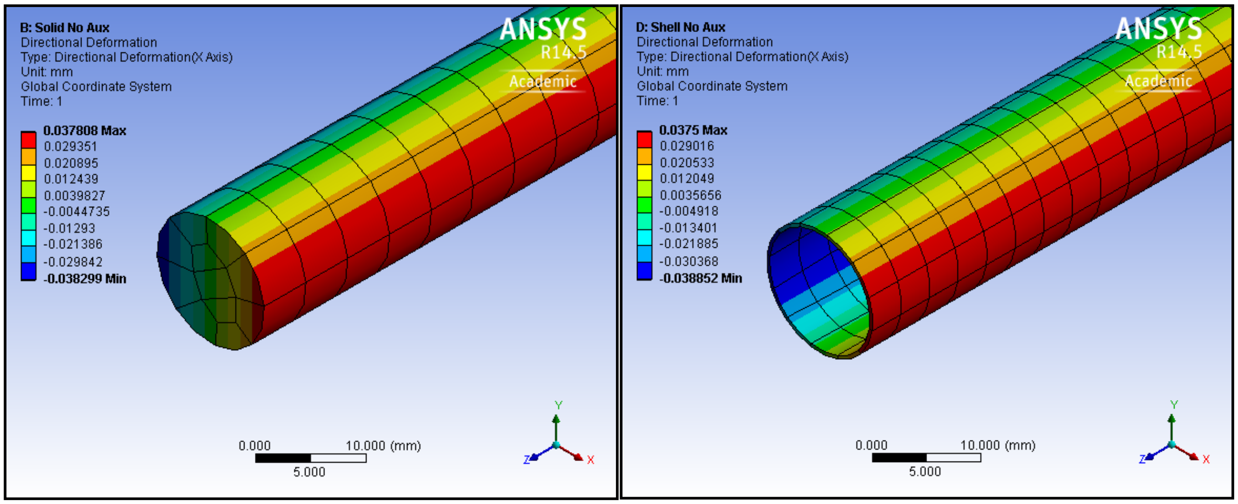

Figure 2: Comparison of the Effects of Solid Elements (left) and Shell Elements (right) on Deformation of a Fuel Sheath without Appendages

Since the deformation profiles between the two models were similar, the comparison study demonstrated that three-dimensional shell elements could be used instead of solid elements to model the fuel sheath. This preliminary result sets the stage for the final model as shell elements are less computationally intensive and less prone to negative Jacobian errors than solid elements for thin-walled structures [12]. Reduced computation time would be valuable in this study due to the sheer volume of fuel elements that need to be modelled in future work.

The second model analyzed the impact of the appendages on the deformation profile of the fuel bundle. It is critical for the final geometry to include bearing and spacer pads to apply the appropriate boundary conditions because it is expected that the fuel elements will apply mechanical loads to one another via contact at spacer pads in future modelling work [13].For comparison purposes, solid and shell elements were used to model the sheath while the appendages were represented by solid elements. The resulting deformation profiles are illustrated in Figure 3.

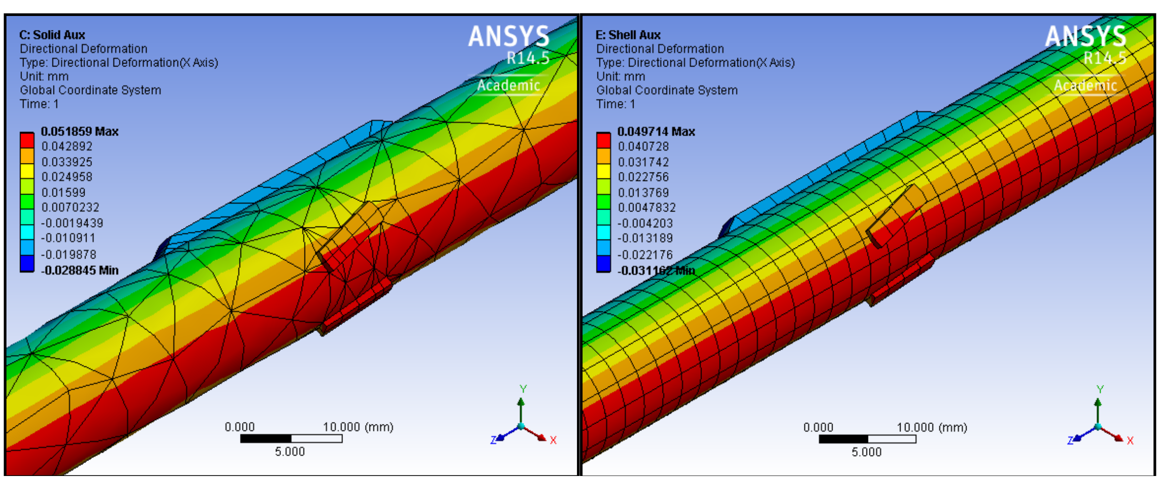

Figure 3: Effects of Solid Brazed Appendages on Solid Element Sheath (left) and on Shell Element Sheath (right) on Deformation.

By visual comparison, it is evident that the deformation profile of the solid model is less smooth than the deformation profile of the shell model. However, the deformation values of the two models are sufficiently close to each other to conclude that the shell model with solid appendages bonded to its surface provides an accurate deformation profile of a fuel element and can be used in the next phase of fuel bundle deformation models pertaining to this study.

The third simplified model was created to simulate the contact between fuel elements at the spacer pads and between the bearing pads and pressure tube as a result of uniform thermal expansion. The assembly under consideration in this analysis was a three-dimensional solid element model thatincluded three fuel rods, having anouter diameter of 13.5 mm and an inner diameter of 13.1 mm respectively, with appropriately positioned spacer pads and bearing pads. The simplified assembly is presented in Figure 4.

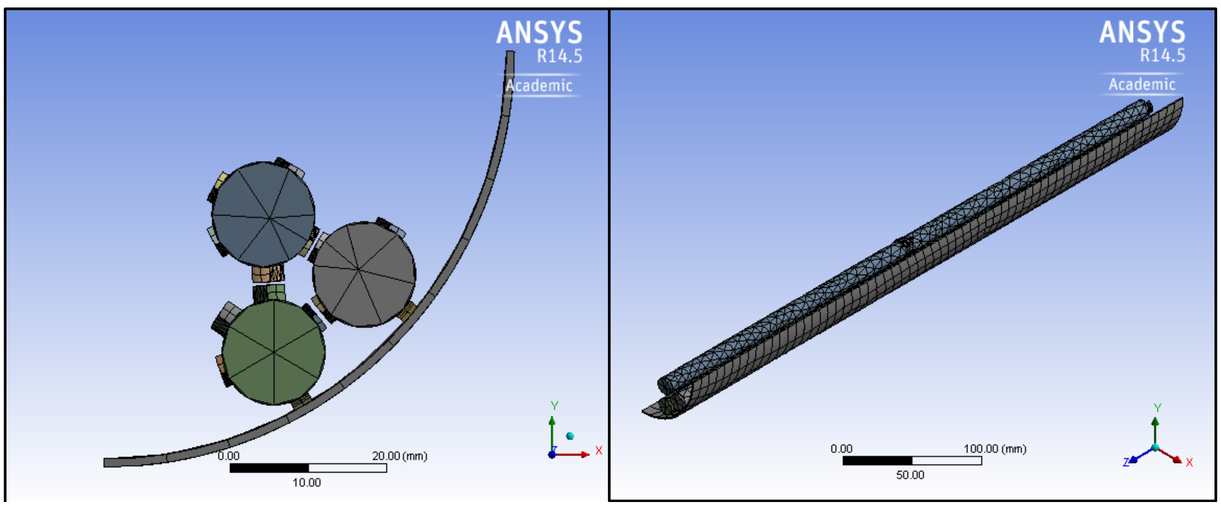

Figure 4: Geometry of the Assembly used for Contact Modelling Analysis

A three-dimensional surface body in the shape of a quarter arc was added to simulate the pressure tube. The addition of a section of the pressure tube eliminated rigid body motion as the bearing pads of the outer fuel elementsare constrained by contact with the fixed pressure tube arc. As per design, the initial gap between the fuel elements is kept open to allow the expansion of the sheaths, and spacer pads to initiate the contact with spacer pads of the adjacent elements. Defining the alignment of the fuel elementsin this manner serves to bring the elementssufficiently close to one another such that initial contact may be detected as thermal expansion occurs.

The results of this analysis indicate that the fuel elements heated to 850 °C deformed downwards by about 1.5056 mm and deformed laterally by 1.0987 mm in total. These values are lower than the values obtained from the out-of-reactor experiment, where some parts of fuel bundle deformed downwards by 2.0 ± 0.2 mm and increased the lateral diameter by 4.2 ± 0.2 mm [11]. The lower magnitude of vertical and lateral deformation observed in the model is partially attributable to the applied boundary conditions, which may be over-constraining the current model. A graphical representation of the vertical and lateral deformation as the pins begin to come into contact is shown in Figure 5.

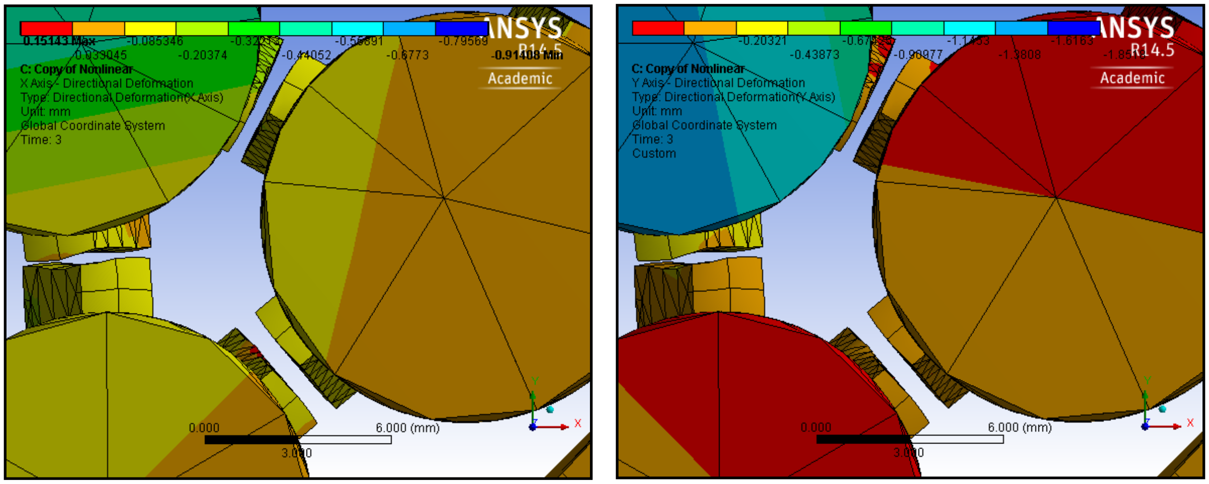

Figure 5: Maximum Lateral (left) and Vertical Displacement (right) Between Three Fuel Elements as a Result of the Thermal Load Applied

The deformation profile does not seem to be as smooth as intuitively expected. The rough deformation contours are attributable to the application of boundary conditions at the contact regions, andthe bonded contact algorithm used. Using a uniform thermal profile should have resulted in a more uniform distribution than the one seen above; however, there is a clear non-uniformity in the deformation results, with the contact between the inner fuel element and the outer element on the left being more pronounced than the top right contact between the inner and outer elements. Since the contact algorithm used in this model proved to underestimate the extent of deformation, the contact between the fuel elements will be defined using the frictional contact definition in future models.

4. Future Direction

While the preliminary results provide a comprehensive foundation for the development of the model, there are a number of physical phenomena that need to be added tothe thermo-mechanical analysis before a fully coupled fuel bundle model is created. Additional physical phenomena, such as creepand gravity, need to be added to the model to replicate the circumstances of the in-laboratory experimentaccurately.

Theelement-to-element contact definition needs to be tested to ensure that the fuel elements are free to come into and out of contact with each other, which is the expected behaviour of the fuel elements given the observations from the experiment. The endplates also need to be included to remove artificial boundary constraints added to achieve convergence. Furthermore, modelling the entire bundle will impose additional deformation due to the presence of temperature gradients across the bundle, and due to the contact loads imposed by the other fuel elements in the bundle.

Once the contact model for the three fuel elements is running for asymmetric thermal distribution, the remaining fuel elements willbe incorporated to form the entire bundle. The results from this final bundle deformation model will be validated against the deformation results obtained from the out-of-reactor experiment to corroborate that the model simulates the behavior of a fuel bundle in transient conditions. Furthermore, a wide range of convergence studies will be conducted to evaluate the sensitivity of the model. This model validation process will illustrate confidence in the model behaviour and ensure that the model can be used in future studies to predict the extent of deformation of a fuel bundle correctly.

5. Conclusion

A preliminary finite-element model of an empty 37-element CANDU fuel bundle has been developed. The objective of this initial model is to predict the geometrical deformation of the fuel sheath as part of the bundleand to model the contact between fuel elements correctly before adding the complication of fuel pellet-to-fuel sheath interaction. In the lead up to thedevelopment of a more sophisticated bundle deformation model, the general behaviour of a single fuel element and a group of three adjacent fuel elements have been correctly modelled.For off-normal and accident conditions, the model willnow be extended to a full 37-element bundle, without fuel pellets, and compared toin-laboratory experiment.The modelling approachused andthe progress results will be presented.

6. References

[1] J. Veeder and M. H. Schankula, "Bowing of Pelletized Fuel Elements: Theory and In-Reactor Experiments," *Nuclear Engineering and Design*, vol. 29, pp. 167-179, 1974.

[2] W. Hartmann and J. Y. Jung, "Safety Analysis Methodology for Aged CANDU-6 Nuclear Reactors," *Nuclear Engineering and Technology*, vol. 45, no. 5, pp. 581-588, 2013.

[3] E. Kohn, R. Hu, G. I. Hadaller and R. A. Fortman, "Understanding CANDU Fuel Bowing in Dryout: An Industry Approach," 1997. [Online]. Available: http://www.iaea.org/inis/collection/NCLCollectionStore/_Public/31/006/31006128.pdf. [Accessed 9 March 2016].

[4] D. R. Olander, "Fundamental aspects of nuclear reactor fuel elements," U.S Department of Energy, 1976, p. 131.

[5] "Trip Parameter Acceptance Criteria for the Safety Analysis of CANDU Nuclear Power Plants," Canadian Nuclear Safety Commission, May 2006.

[6] J. Pachner, P. Kumar and J. C. Almagro, "Assessment and Management of Ageing of Major Nuclear Power Plant ComponentsImportant to Safety: CANDU Pressure Tubes," International Atomic Energy Agency, Vienna, 1998.

[7] J. S. Bell and B. J. Lewis, "CANDU fuel bundle deformation modelling with COMSOL," *Journal of Nuclear Engineering and Design,* vol. 250, pp. 134-141, 2012.

[8] ANSYS Inc., ANSYS Mechanical APDL Verification Manual, Canonsburg, PA: Release 14.0, November 2011.

[9] M. Tayal, "Modeling CANDU Fuel under Normal Operating Conditions, ELESTRES Code

Description," AECL Report 9331, February 1987.

[10] SCDAP/RELAP5 Development Team, Volume IV: MATPRO--A Library of Material Properties for Light-Water-Reactor Accident Analysis, Idaho Falls, ID: Idaho National Engineering and Environmental Laboratory, Oct 1997.

[11] C. Thiriet, R. Dickson and K. Colins, "Fuel Bundle Behaviour Test in the Fuel Bundle Behaviour Apparatus," Canadian Nuclear Labs, Chalk River, 2015.

[12] S. Moaveni, Finite Element Analysis, 2003: Pearson Education, New Jersey.

[13] L. C. Walters and A. F. Williams, "Fuel Bundle Deformation Model," in *8th International Conference on CANDU Fuel*, Honey Harbour, ON, 2003.