Development of a 3-D Finite Element Model to Examine Fuel Bundle Behaviour under Post-Dryout Heat Transfer Conditions

C. Krasnaj, P.K Chan and D. Wowk
Royal Military College of Canada, Kingston, Ontario, Canada
Cody.Krasnaj@rmc.ca

A Master's Level Submission

Summary

Elements in a CANDU fuel bundle can deform (i.e., deflect or bow) during their residence in the fuel channel by thermally-induced phenomena. Although small deflections are warranted, there is a risk that under post-dryout conditions, the induced thermal gradient may increase this effect and put the integrity of the fuel and fuel channel at risk. With reactor aging effects potentially affecting sub-channel thermalhydraulics, the chance of dryout occurring is increased. The purpose of this work is to develop a 3-D finite element model to examine element deformation when subjected to the thermal conditions in post-dryout heat transfer.

1. Introduction

With the age of many nuclear power plants either reaching or approaching the end of their design life, the effects of aging phenomena on both plant performance and safety analysis has become more of an interest to regulators and nuclear energy corporations. Although effects were considered individually in design and manufacturing, the rate and integrated impact on safety analysis is greater than expected.

One of the known ageing phenomena of greatest interest is pressure tube diametric creep. As the pressure tube ages with operational life, the diameter of the pressure tube begins to sage and creep. This has the effect of increasing flow by-pass over the fuel bundle resulting in less flow between the subchannels. Less flow in the subchannels will change the critical heat flux value which directly impacts where and when dryout will occur.

Post-dryout heat transfer can lead to thermally induced phenomena such as element bowing due to increased thermal gradients. These deflections can put the integrity of the fuel and fuel channel at risk. To prevent dryout, conservative parameters are used for the trip parameter acceptance criteria as stated in CNSC regulatory guide G-144. Limited in-reactor experiments have indicated that operational margins exist when compared to those stated in G-144. A 3-D model utilizing the finite-element method to examine fuel deformation under post-dryout heat transfer conditions could be used as a tool to help better understand the effects of thermally-induced phenomena. A fully coupled thermal-mechanical model is being developed to simulate the deformation behaviour of a section of a bundle at the onset of dryout.

2. Model Development

ANSYS is a commercially available finite element program used in a wide variety of engineering fields for its multiphysics simulation capability. Its ability to easily couple different physics packages together within complex systems makes it an ideal platform for this investigation. In addition, ANSYS can solve physical contact problems which is a necessity for this study as deflections of fuel elements are likely to come into contact with surrounding elements. The development of this model will take advantage of ANSYSs' structural and thermal analysis packages and Atomic Energy of Canada Limited's Industry Standard Toolset thermalhydraulic code, ASSERT-PV. Although all simulation configurations such as material properties, boundary conditions and applied loads will be run in ANSYS, information pertaining to post-dryout heat transfer conditions will be provided for input by ASSERT-PV. A more descriptive analysis of how each component will contribute is discussed below.

2.1 Physical Geometry

To simplify the fuel bundle model, the 30 individual UO_2 pellets in each element will be treated as one continuous UO_2 monolithic stack. Advantages are gained from reducing the computational complexity of the model and by avoiding the pellet-to-pellet interactions which could lead to contact convergence issues.

In a 37-element fuel bundle, there exist planes of symmetry that can be exploited to further simplify the model's computational requirements. Instead of modelling a full length fuel bundle, only a half-length bundle will be considered. Furthermore, the current intention is to expose only one element to dryout conditions. Considering the cyclic symmetry of the fuel elements arrangements, only 1/6th of the elements are necessary to model if the element experiencing dryout is chosen to be part of that cut section. This is illustrated in Figure 1.

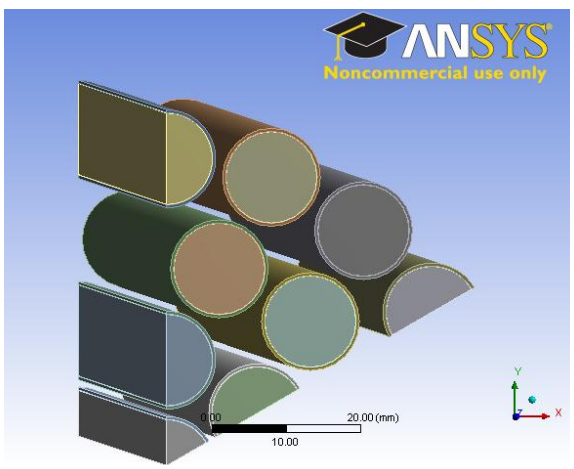

Figure 1 Cross section showing the fuel and sheath of 1/6th of a 37-element fuel bundle

Final pieces to be included within the geometry include the appendages such as bearing and spacer pads. These are necessary to model normal interaction between adjacent elements surrounding a subchannel.

2.2 ASSERT-PV

ASSERT-PV is a computer code that has been developed by Atomic Energy of Canada Limited to model thermalhydraulic behaviour in rod bundles by utilizing the concept of subchannel analysis applied to two-phase flow. It uses a 1D assumption for fluid flow as the velocity field is predominately in one direction which is along the fuel elements. Interaction between interconnecting subchannels can be considered depending on the choice of the mixing model chosen.

The ASSERT-PV code will provide heat transfer coefficients along axial zones for specified elements. Critical heat flux data can also be found and together this information will be used as inputs into the ANSYS thermal analysis package.

2.3 Thermal Analysis

Temperature dependent material properties for UO_2 and Zircaloy will be manually entered into the material database of ANSYS. ANSYS is capable of handling many special features which allow nonlinearities and other secondary effects to be included in the solution, such as plasticity, large strain, creep, contact, stress stiffening and material anisotropy. Internal heat generation within the fuel pellets will be applied and the appropriate heat transfer coefficients supplied by ASSERT will be applied for convective heat transfer on the sheath surface. The appropriate contact mechanisms between the fuel stack and sheath will be established as well as proper symmetry conditions. Temperature distribution profiles will be saved and used as inputs into the mechanical model.

2.4 Mechanical Analysis

Within the ANSY mechanical package, further boundary conditions like external coolant pressures on the sheath as well as internal gas pressures from fission gas release can be applied. As well, a creep model for the Zircaloy sheath and gravity can be added here. The temperature distributions will be loaded into the mechanical framework and the appropriate integrated mechanical response based on the thermal and mechanical properties of the individual components will be found. Stress and strain values in the pellet stack and sheath due to the pressures and thermal expansions will be computed.

3. Model Progress

Preliminary progress has been made thus far. A single fuel element with the pellets modelled as a monolithic stack with an outer sheath has been made. Uniform internal heat generation with pellet-sheath interaction due to thermal expansion has been established. For preliminary testing purposes, normal operating condition heat transfer coefficients have been applied for convectional heat transfer on the sheath. An illustration of the temperature distribution found assuming a 30kW/m power density is shown in Figure 2.

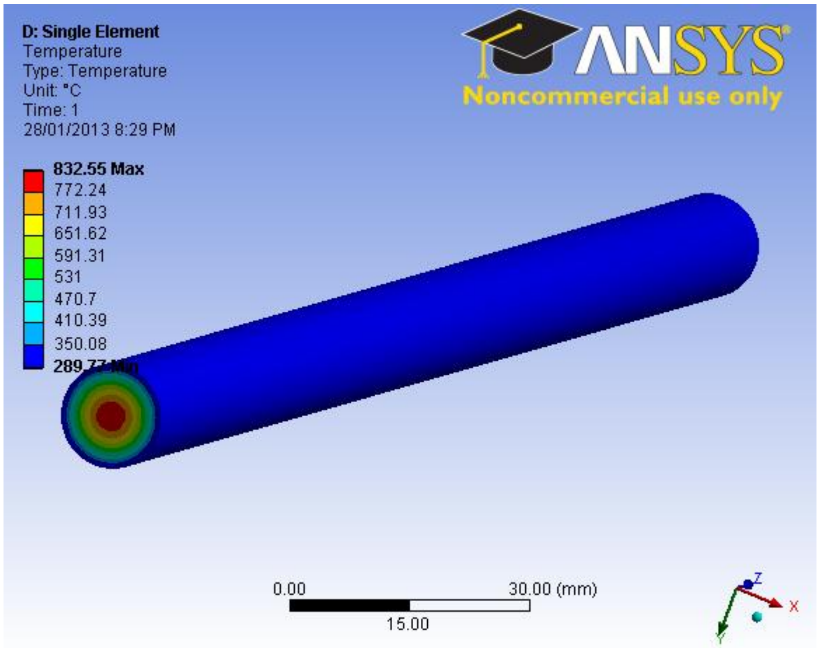

Figure 2 Temperature distribution profile for normal operating conditions operating at 30kW/m.

Both internal and external forces have been applied to represent internal gas pressure and coolant pressure respectively. Gravity has also been included although the preliminary model does not included bearing pads which prevent accurate boundary conditions from being enforced at this stage.

4. Future Work

There is still much work that must be done and altered before a working model exists. Currently, the internal volumetric heat generation is not a function of burnup or radial position. Since this model is investigating thermal phenomena it is important that the proper heat generation is modelled given by

$$Q = \frac{P_{lin}}{\pi a_{pel}^2}\left(\frac{\kappa a_{pel}}{2 I_1(\kappa a_{pel})}\right) I_0(\kappa r) \quad (1)$$

where P_{lin} is a given linear power rating (Wm^{-1}), I_1 and I_0 are the first and zeroth order modified Bessel functions of the first kind, respectively, a_{pel} is the initial radius of the pellet (m) and κ is the inverse neutron diffusion length (m^{-1}). [1]

In addition, the bearing and spacer pads must be added to the physical geometry so that appropriate boundary conditions and loads can be applied at the correct locations of the element.

Once a single fuel element model is running for normal operating conditions, applying the heat transfer coefficients from ASSERT to account for multiphase flow during the onset of dryout must be changed. After the single element has successfully been setup with dryout conditions, the incorporation of more elements to form a subchannel will follow. The completed geometry will have the same number of elements shown in Figure 1.

5. Conclusion

With reactor aging effects increasing the risk of dryout, more insight into determining the fuel bundle behavior post-dryout could prove to be a useful tool for safety analysis. Work has begun on the development of a 3D finite element model to examine fuel bundle deformation under post-dryout conditions. Preliminary results from the work mentioned in Section 3 will be presented along with further progress.

6. References

[1] D.R. Olander, “Fundamental aspects of nuclear reactor fuel elements”, *Division of Reactor Development and Demonstration*, 1976, pp. 131.