12th International Conference on CANDU Fuel
Holiday-Inn Waterfront Hotel
Kingston, Ontario, Canada, 2013 September 15-18

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

ABSTRACT-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 deformationwhen subjected to the thermal conditions in post-dryout heat transfer.

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 greatestinterest is pressure tube diametric creep. As the pressure tube ages with operational life, the diameter of the pressure tube begins to sag due to 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 valuewhich directly impacts where and when dryout will occur.

Post-dryout heat transfer can cause thermal gradients to develop across the sheath due to deteriorating heat transfer conditions. As a result, thermally induced phenomena such as element bowing can develop in these dryout locations[1]. 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[2]. 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 coupled thermal-structuralmodel is being developed to simulate the deformation behaviour of a section of a bundle at the onset of dryout.The purpose of this paper is to outline the characteristics of the 3D finite element analysis and describe how it will be used to explore the response of a fuel bundle to dryout conditions.

1. State of Art

The steady advancementof computer technology over the past decade has allowed for relatively fast multi-core systems to be purchased with large amounts of memory at reasonable costs. This increase in readily available computer power has aided the popularity of finite element modelling for research and virtual experimentation among many industries by allowing fully coupled multi-physics problems to be simulated.

In the nuclear industry, the ability to accurately model fuel phenomena in 3 dimensions would offer improvement to current computer codes which typically have physical and dimensional simplifications. Often, simulations are run using 1D or 2D models, and these limitations cause uncertainty in the precision of the fuel performance code. This leads to overly conservative operating limits to ensure adequate safety margins exist. In addition, validated 3D computer models can offer an alternative to costly experiments when studying a particular reactor scenario. The simplicity of changing multiple parameters in a computer model and rerunning a test requires considerably less resources, in terms of time and money, compared to the equivalent alterations that would be needed in a laboratory experiment. The difficulties in developing these finite element models arise when trying to accurately capture the conditionsand phenomena under study. Successful finite element modelling of nuclear fuel has been accomplished at both the academic and industry level. At the Royal Military College of Canada, the finite element program COMSOL has been used in predicting fuel performance such as temperatures, fission gas release, stress and strains, and sheath deformation[3][4]. At Atomic Energy of Canada Limited’s Chalk River Laboratory, the finite element program ANSYS was used to develop a bundle deformation model to benchmark against out-of-reactor tests to determine the feasibility of the finite element package[5]. These models include phenomena interacting on the scale of the pellet and sheath region for a single fuel rod and bundle respectively. Simulations of bundle deformation under post dryout conditions have not been widely published butthere is promising evidence that finite element modelling could be used for future fuel deformation studies. Even with these advancements in fuel modelling, including more of the geometry such as a full 37-element fuel bundle with the same amount of modelled physics is still limited by our current computing power. Therefore, in order to model larger fuel components, a balance is required between the physical size and the amount of physics in the model. Simulations should only include thespecific aspects that are required to accurately capture the results of interest, and should avoid being made withunnecessary complexities.

2. Safety Concerns with Post Dryout Heat Transfer and Potential Insights Offered through Modelling

Fuel sheath integrity is a very important parameter which characterises the integrity of the fuel bundle as a whole. The fuel sheath acts as a containment barrier and once it has been breached, radioactive and toxic fission products can migrate through the primary heat transport system. Under normal operating conditions, the fuel and sheath lie in contact with one another and heat ispredominantly removed from the fuel by conduction across the sheath.If dryout occurs on the sheath, the amount of heat that can be transferred from the fuel is reduced and the temperature of the fuel can rise. Since dryout can be localized, a non-uniform temperature gradient can develop which will cause the fuel to thermally expansion both radially and axially. This additional thermal strain is transferred to the sheath through direct contact, forcing it to balloon out and riskjeopardizing its integrity (see Figure 1). The preliminary model illustrated in Figure 1shows the initial gap between the fuel and sheath before entering reactor conditions. The bottom image shows the transition to gap closure as a result of fuel thermally expanding at a high linear power.Thispost dryout behaviouris one of the studies we wish to model in order to obtain the induced strain as well as the interface pressure between the fuel and sheath.We are hoping to capture how the different thermal conductivities between the fuel and sheath affect the rate of rise in fuel sheath temperature. The low thermal conductivity of the fuelmay limit the amount of thermal expansion that occurs immediately following dryout allowing some time to pass before large strains and pressures develop.

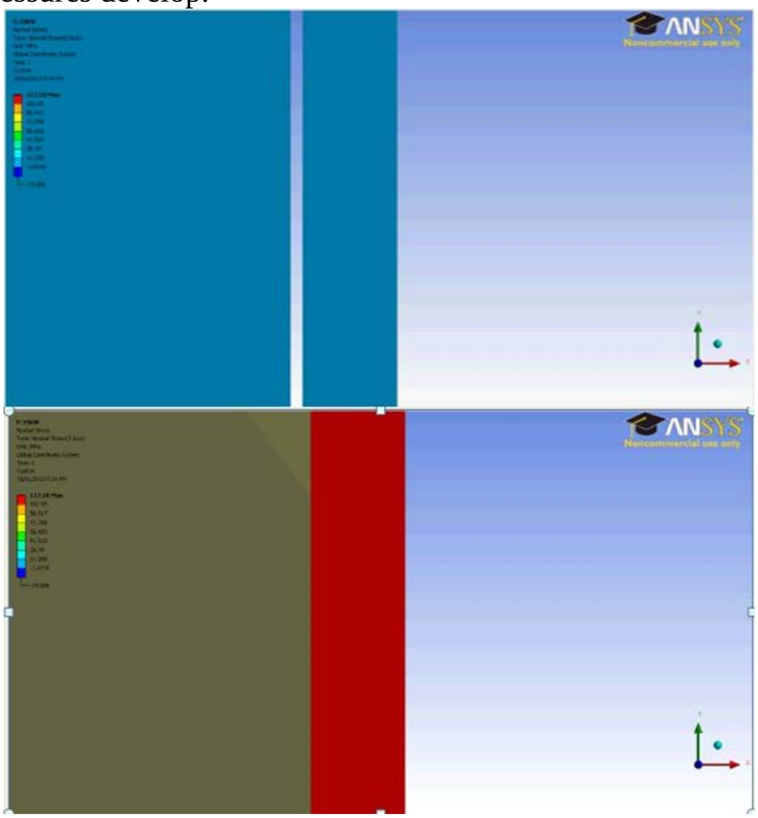

Figure 1: 2-D cross section of a fuel element. The top image shows the initial gap between the fuel and sheath before being subjected to reactor conditions. The bottom shows how at high temperature, the thermal expansion of the fuel forces the sheath outwards.

Furthermore, the differences among subchannel geometrywill impact the heat transfer efficiency of each element. The heat transfer coefficient for two-phase flow will vary drastically depending on the void fraction along the bundle. These differences can give rise to non-uniform axial temperature distributions. The difference in thermal expansion along the fuel and sheath can cause distortion of the bundle geometry from asymmetric expansion. The close proximity of adjacent fuel elements and their linked connections through the welded endplates allows for the deflection of one element to have an influence on the stress induced in another. If these small distortions become large enough, they can impose on the subchannel flows creating bottlenecks in the flow patterns. If the restriction of flow is large enough, subchannel thermalhydraulics can change and higher sheath temperatures can develop. If dryout were to occur on another element facing the same subchannel, an amplification of bowing can initiate a positive feedback effect.This could lead to coolant flow restriction and put the entire fuel channel integrity in jeopardy. Elements would continue to deflectuntil stopped by another element or the pressure tube. This amount of contact could very well cause fuel failure. With the finite element model being developed, we wish to investigate how an element on each bundle ring will deflect when subjected to a similar temperature gradient. A study will be performed to look at the relative deflections that are caused by non-uniform temperature distributions as a result of flux gradients, varying coolant conditions and high temperature creep. A study of this nature is important as it can be extended to determine how large of a temperature transient would be required to cause element-to-element contact away from spacer pads. By modeling these phenomena and their resultant effects, we hope to gain better insight into the range of conditions that ensure safe operation.

3. Modelling Tool Selection

ANSYS is a commercially available finite element program used in a wide variety of engineering fields for its multiphysics simulation capability.It has the ability to easily couple different physics packages together within complex geometries and systems. The primary focus of the current work requires the ability to model thermally driven phenomena which are strongly coupled to the mechanical behaviour.Both the thermal and structural ANSYS analysis packages are widely acknowledged for the accuracy of their results and thus make it an ideal platform for this investigation[6]. Furthermore, ANSYS has strong contact modelling capabilities. This is a necessity for this study as contact will be present between the pellet and sheath as well as deflected fuel elements that may come into contact with surrounding elements or the pressure tube. The modelled contact will provide the load path from the pellet to the sheath to the pressure tube and as a result will influence the entire bundle behaviour.Thus, ANSYS provides the necessary tools for this work and past success modelling fuel deformation fulfils the projects requirements for an appropriate solver.

4. Model Outline

Dryout spots along the fuel sheath are a very complex phenomenonthat depends significantly on fuel channel thermalhydraulic conditions upstream of the dryout location rather than on the local conditions at the dryout location. When modelling heat transfer between a solid and a fluid, there are generally two approaches used to model the convective cooling. The first approach is to use heat transfer coefficients to describe the thermal dissipation along the boundary and the second is to extend the model to describe the flow and heat transfer in the surrounding fluid. To avoid the complexity of modelling two-phase fluid flow, the first approach will be used where the heat transfer coefficients for the relevant thermalhydraulic conditions will be supplied externally by the Industry Standard Toolset (IST) code ASSERT-PV. ASSERT will be run as an external code to generate input parameters for the ANSYS model.

Within ANSYS, both the thermal and structural analysis packages willbe coupled together. Each analysis will be responsible for including the appropriate boundary conditions and physics being studied. A flow chart of the development structure is given in Figure 2.

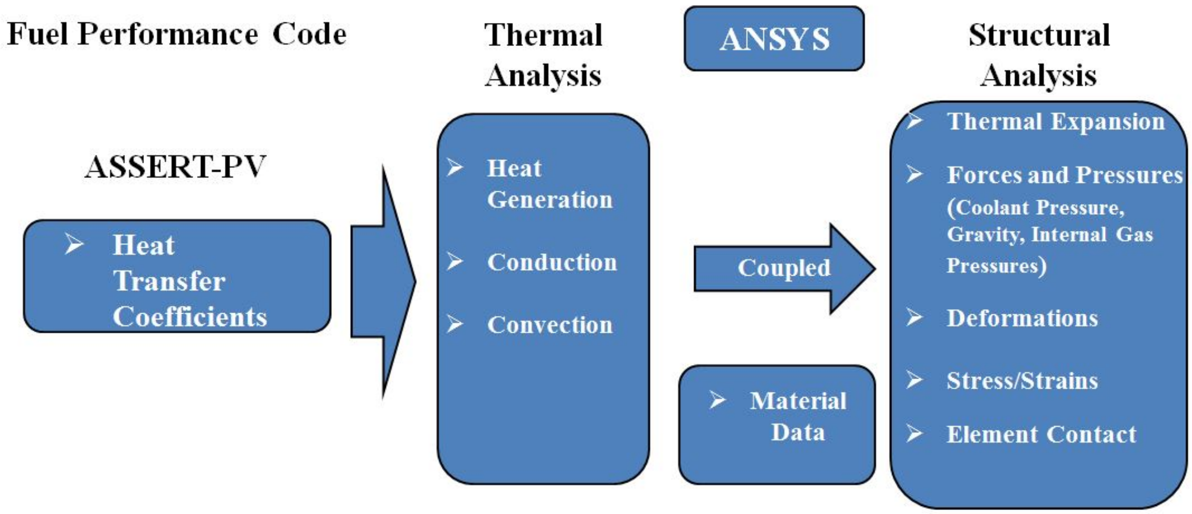

Figure 2: Flow chart for the model implementation

A more descriptive discussion on how each will be implemented in given below.

4.1 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. The pellet and sheath will be considered separate parts and a contact mechanism will be established between them. Characteristic dimensional values for the fuel bundle and its components will be used.

Taking advantage of symmetry and reducing the model geometry can help reduce the overall size and complexity of the model, but can only be applied if the geometry, loading and response are all symmetrical. One has to be cautious when simplifying geometries as removing or adding critical features may have an impact on the progression of the phenomena under study.In a 37-element fuel bundle, there exists a plane of symmetry halfway along the length of the fuel bundle. This can be exploited to reduce computational requirements by only modelling a half-length bundle. Although thermalhydraulic conditions are not symmetric about the centre mid-plane, the proper boundary conditions will be used such that the simplification is justified. The thermal analysis will have a specified heat flux boundary condition along the plane of symmetry to mimic the temperature gradient along a full fuel element.The structural analysis will have a roller boundary condition such that only axial deflections are constrained. The symmetrical placement of spacer pads along the mid-plane of fuel elements as well as the axial support offered by the welded endplates allow for these assumptions to be used. Furthermore, the intention is to expose only one element at a time to dryout conditions. If the element experiencing dryout is chosen to be part of the third or fourth element ring, only a subsection of the bundle needs to be modelled as surrounding elements are the ones of most interest(see Figure 3).The final pieces to be included with the geometry will include the inner bearing pads and outside spacer pads. These are necessary to model normal interactions between adjacent elements in a subchannel.

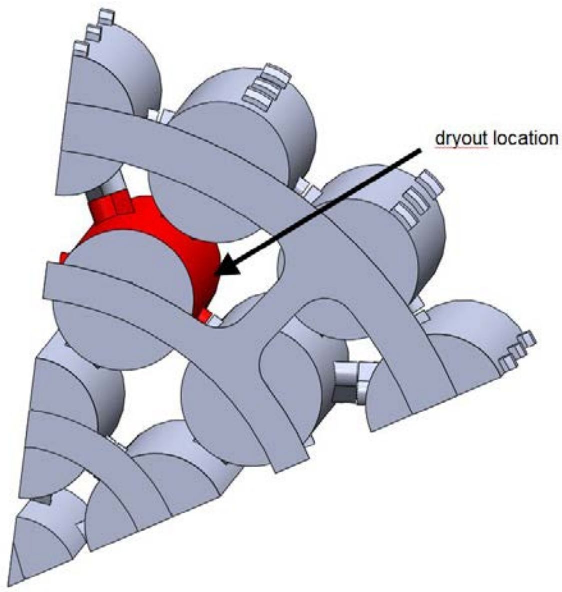

Figure 3: Cross section showing the fuel and sheath of 1/6th of a 37-element fuel bundle

4.2 Material Properties

For each type of analysis in ANSYS, there are basic material properties that must be included. The thermal analysis will require the thermal conductivity, heat capacity and density whereas the structural analysis will require Young's Modulus, Poisson's ratio and the coefficient of thermal expansion for each material in the model. In addition, ANSYS is capable of handling many special features allowing nonlinearities and other secondary effects to be included in the solution, such as plasticity, creep, and material anisotropy. These additional features require nonlinear material properties to be specified such as the yielding criteria and hardening scheme for plasticity as an example. Other features like creep have predefined general form expressions with material coefficients which can be specified by the user to match a specific creep expression.

Temperature dependent thermal, elastic and plastic material properties for UO_2 and Zircaloy-4 will be retrieved from MATPRO and manually entered into the material database of ANSYS[8]. Both of these materials are exotic from the default material database in ANSYS and therefore must be defined explicitly.

4.3 Post-dryout Heat Transfer Conditions

ASSERT-PV is an IST 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[7]. It uses a 1D assumption for fluid flow as the velocity field of the coolant is predominately in one direction parallel to the fuel elements. The choice of the mixing model will determine whether interaction between interconnecting subchannels will be considered.

The ASSERT-PV code can be used to draw results from a thermalhydraulic analysis for input as boundary conditions into the ANSYS model. Although the model itself will not contain thermalhydraulic modelling, information pertaining to post-dryout heat transfer will be incorporated from ASSERT. This is accomplished in the form of heat transfer coefficients and heat flux data along axial zones for specified elements. Also, critical heat flux values can be found and together this information will be used as inputs into the ANSYS thermal analysis package.

4.4 Thermal Analysis

Modelling the thermal response of a fuel element requires accurately solving for the temperature profile across the entire body. This is an essential step as the prime loading on the fuel bundle is thermal loading. In analysing post-dryout conditions, the reduced heat transfer coefficient will increase local temperatures over the dryout region. These non-uniform temperature variations over a fuel element will influence how thermal phenomena like bundle deformation develop.
A steady-state thermal analysis will be used to solve for the fuel element temperature profile, which will represent the response at one specific point in time. To satisfy a steady-state energy balance, the heat supplied must be equal to the heat removed. In this model, the heat supplied is given by the internal heat generation within the UO_2 fuel pellets. The heat outlet accounts for the heat removed by the coolant during convective heat transfer on the outer surface of the sheath. The appropriate heat transfer coefficients supplied by ASSERT-PV will be applied in axial segments along the sheath surface. Temperature distribution profiles will be saved and used as inputs into the mechanical model.

4.4.1 Heat Conduction

The distribution of heat after it has been generated by fission is governed by the heat conduction equation. Solving the heat conduction equation throughout the element will provide the temperature distribution. The equation is given by:

$$\rho C_p \frac{\partial T}{\partial t} = \nabla \cdot (k \nabla T) + Q \tag{1}$$

where ρ is the fuel density (kg m^{-2}), k is the thermal conductivity (W m^{-1} K^{-1}), T is the temperature (K), and Q is the volumetric heat generation rate (W m^{-3}). The parameter C_p is the heat capacity of the fuel (J mol^{-1} K^{-1}). The source term, Q, accounts for the heat generated by fission within the pellet. The amount of fission within the pellet depends on the neutron flux across the body. The circular geometry of the fuel pellet creates a radial flux depression which must be taken into account. The probability of how far a neutron will travel before it is captured is contained within the neutron diffusion length. The volumetric heat generation rate (Q) as a function of distance from the fuel centreline, r (m) is given by:

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

where P_{lin} is a given linear power rating (W m^{-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}) [9].

4.5 Structural Analysis

Modelling the mechanical response of the fuel elements can give an indication of the structural integrity based on the induced stress, strain and overall deflection. There are a number of physical processes that can cause strain and each contribution can be added individually to give the cumulative total. In this model, elastic,plastic and thermal strains will be investigated.

A steady state structural analysis will be run using the temperature profile supplied by the thermal analysis. Induced stresses and strains from thermal expansion will be computed. Mechanical loading caused by external coolant pressures on the sheath, internal gas pressures from fission gas release and gravity will be added. In addition, advanced features like a thermal creep model and plasticity for the Zircaloy sheath can be included. Regions where contact is expected will have the appropriate contact mechanisms in place. For example, this includes areas like between the fuel and sheath, between adjacent bearing pads and between fuel elements. Since only a half-length bundle is being modelled, appropriate roller constraints will be used along that plane of symmetry. The pressure tube will be assumed as a rigid body such that the bearing pad facestouching the pressure tube, under these loading conditions, can be constrained as part of the boundary conditions. Places along the sheath away from the bearing pads will be free to move and contact any surrounding elements or tube. The 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 in a cumulative manner. Displacement of finite element nodes will be used to determine the deformation profile.

5. Model Development

An outline of the different phases to be completed is given in Figure 4.

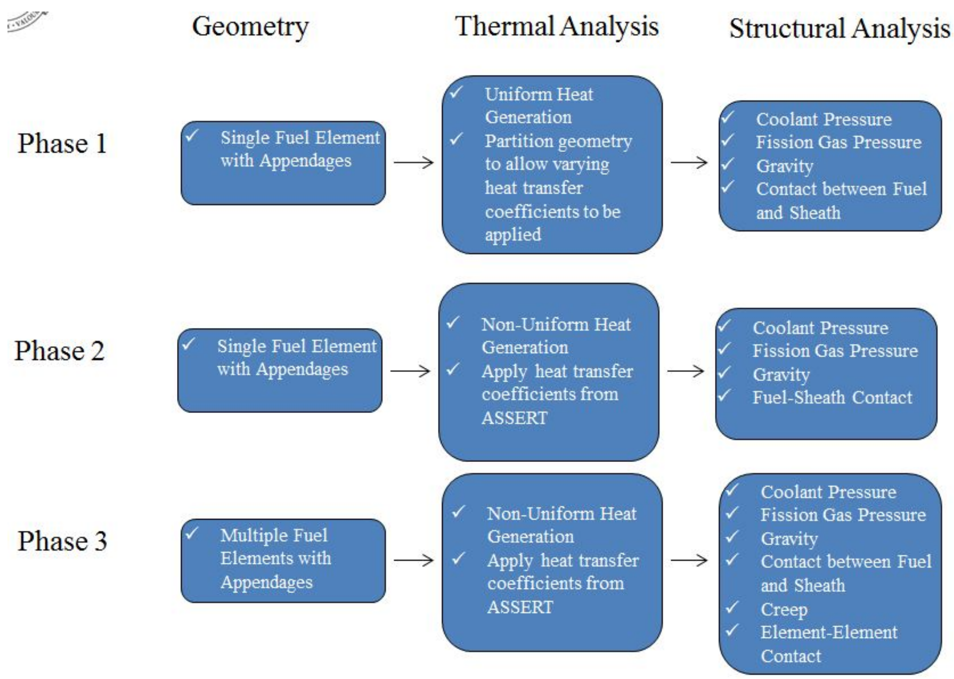

Figure 4: Phase development of project

The first phase of the project is the development of a fuel element model with basic thermal and mechanical loads. The geometry of the fuel element will include spacer and bearing pads and the fuel stack sitting in the sheath. This amount of detail is sufficient to allow the appropriate loads and boundary conditions to be set at the correct locations along the element. The steady state solution of Phase 1 will be compared with the output file from the normal operating condition IST code, ELESTRES[10], to ensure the model behaves as expected. Once a single fuel element model is running for normal operating conditions, the second phase will incorporate the beginning of the post-dryout information supplied by ASSERT-PV and the addition of the non-uniform heat generation. The heat generation will likely have to be applied manually over the fuel as the Bessel functions cannot be directly insert into ANSYS. The heat transfer coefficients from ASSERT will contain the appropriate thermalhydraulic conditions during the onset of dryout. After the single element has successfully been setup with dryout conditions, the transition to Phase 3 involves the incorporation of more fuel elements to form a subchannel. The completed geometry will have multiple surrounding elements. Advanced features such as creep and element-to-element contact will be added in this stage.

Limited experimental data exists for benchmarking post-dryout deformation and contact behaviour of a fuel bundle in a pressure tube. As part of the model validation process, sensitivity studies will be performed to ensure confidence in the model behaviour along with estimating element deflections expected from induced temperature gradients.

6. Preliminary Progress Results

The work completed to date has focused on Phase 1 from Figure 4 in which a single fuel element with the pellets modelled as a monolithic stack with an outer sheath has been created. Uniform internal heat generation with pellet-sheath contact modelling 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 5.

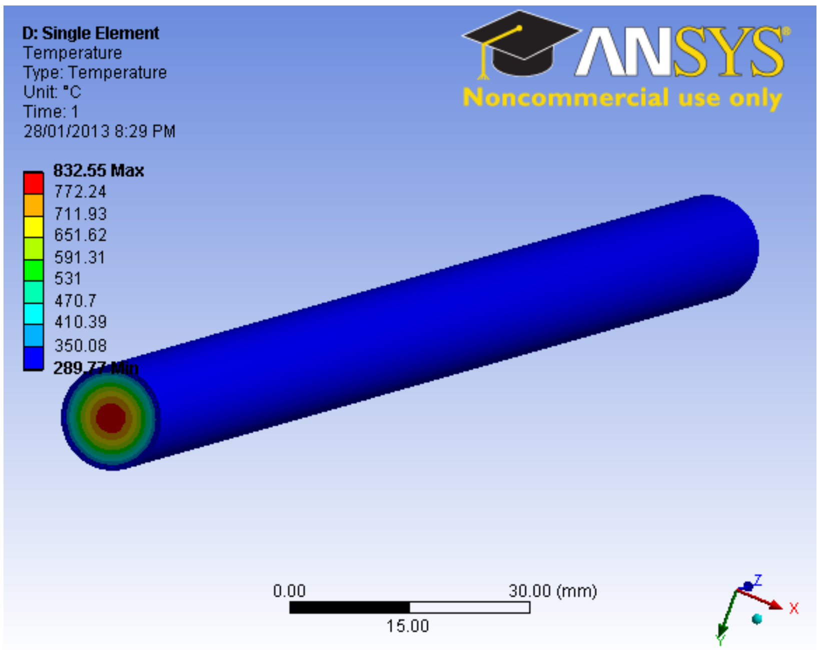

Figure 5: Temperature distribution profile for normal operating conditions at a linear power of 30kW/m.

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.

7. Conclusion

The risk of fuel sheath dryout at high power densities is a known concern and regulatory limits are in place to prevent accidents from arising due to fuel sheath dryout. 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. This work proposes to utilize finite element analysis to model bundle deformation in three dimensions. The fuel performance code, ASSERT-PV, is capable of performing a thermalhydraulic analysis in post-dryout conditions. Results from ASSERT will be used to supplement the coupled thermal-structural analysis in order to incorporate the post-dryout heat transfer conditions. Preliminary work has begun on modeling a single fuel element subjected to post-dryout heat transfer and thermally induced contact. ANSYS' ability to perform contact analysis will be useful in developing a full deformation model once more elements are added. Final stages of the work and additional temperature transient studies will offer insight into the physical effects resulting from thermally induced post-dryout deformation.

8. 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] "Trip Parameter Acceptance Criteria for the Safety Analysis of CANDU Nuclear Power Plants," May 2006.

[3] D. Morgan, "A thermomechanical model of CANDU fuel," Royal Military College of Canada, Kingston, Ontario, 2007.

[4] K. Shaheen, J.S Bell, A. Prudil, and B.J Lewis, "Modelling CANDU Fuel Element and Bundle Behaviour for Performance of Intact and Defective Fuel," in *International Conference on Future of Heavy Water Reactors*, Ottawa, Ontario, 2011, p. Paper 011.

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

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

[7] M. B. Carver, C. J. Kiteley, R. Q.-N. Zhou, and S. V. Junop, "VALIDATION OF THE ASSERT SUBCHANNEL CODE - PREDICTION OF CRITICAL HEAT-FLUX IN STANDARD AND NONSTANDARD CANDU BUNDLE GEOMETRIES," *Nuclear Technology*, vol. 112, no. 3, pp. 299-314, 1995.

[8] 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.

[9] D. R. Olander, *Fundamental Aspects of Nuclear Reactor Fuel Elements*.: U.S Department of Energy, 1976.

[10] M. Tayal, "Modeling CANDU Fuel under Normal Operating Conditions, ELESTRES Code Description," February 1987.