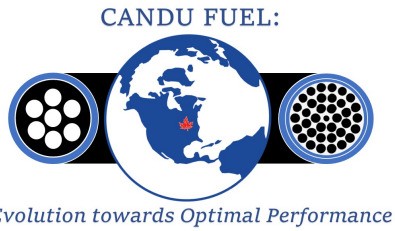
13th International Conference on CANDU Fuel
Holiday-Inn Waterfront Hotel
Kingston, Ontario, Canada, 2016 August 15-18
CW-126120-CONF-004 Rev 0 / UNRESTRICTED



Finite Element Simulations of a Fuel Element Simulator Sagging Experiment

A.F. Williams[1], S. Yatabe[1] and A. Belov[1]
[1]Canadian Nuclear Laboratories, Chalk River, ON, Canada
(Fuel & Fuel Channel Safety Branch, CNL, Chalk River, ON K0J 1P0,
613 584 3311 ext. 43362, tony.williams@cnl.ca)

ABSTRACT – In the IEC-I-1 test, an internally heated CANDU fuel element simulator was subjected to five temperature transients: three transients to a sheath peak temperature of 600°C, followed by two further transients to a sheath temperature of 800°C. A detailed 3D finite element model of this experiment was created using the ANSYS finite element software package. The model includes fuel pellets and sheath as separate components that interact via contact elements. The study included a detailed sensitivity study of a number of model parameters, including pellet-to-sheath friction, contact penalty function, rigid body damping, time step size and the creep rate ratio limit.

1. Introduction

The mechanical behaviour of a CANDU fuel pin is greatly influenced by the interaction of the fuel pellets and the sheath. If the fuel pellets are in tight contact with the fuel sheath then they substantially stiffen the fuel pin and resist lateral deflections. The interaction is further complicated by the behaviour of the sheath at the pellet-to-pellet interfaces. It is well known that circumferential stresses in the sheath are higher here due to pellet hourglassing [1], but also axial stresses are concentrated at the pellet-to-pellet interfaces. As the fuel pellets are heated and cooled they move in and out of contact with the sheath. The degree of stress concentration at the pellet-to-pellet interfaces is influenced by both the timing of when the pellet comes into contact with the sheath and the amount of friction between the pellet and the sheath. It should also be noted that the creep behaviour of the sheath material is a function of the local stress, and hence the deformation of the fuel pin is strongly influenced by the dynamics of the pellet-to-sheath interaction.

For a number of years, work has been conducted at the Chalk River laboratories to create a finite element based model of a fuel pin capable of determining fuel pin deformations and correctly accounting for the complex interaction of the pellets and sheath [2]. These computer modelling efforts have been conducted in conjunction with a series of fuel element simulator tests to explore the deformation behaviour of fuel pins under lateral loads and temperature gradients [3]. This paper describes the use of the finite element based model to simulate such an experiment.

It should be noted that the model described here cannot be considered as a full fuel performance model as it does not explicitly include several phenomena that influence fuel behaviour under irradiation conditions including: fission product production and associated fuel swelling, fuel densification, fuel pellet cracking, and irradiation induced creep of the sheathing. Many of these phenomena may be accounted for by using the model in conjunction

with more traditional fuel performance codes such as ELESTRES [5]. Work on developing and integrating model for such phenomena in to the finite element model is ongoing.

2. The Experiment

The IEC-I-1 experiment was a Fuel Element Simulator (FES) experiment designed to characterize fuel element sag at temperatures up to 800°C. The FES setup in the test chamber is shown in Figure 1.

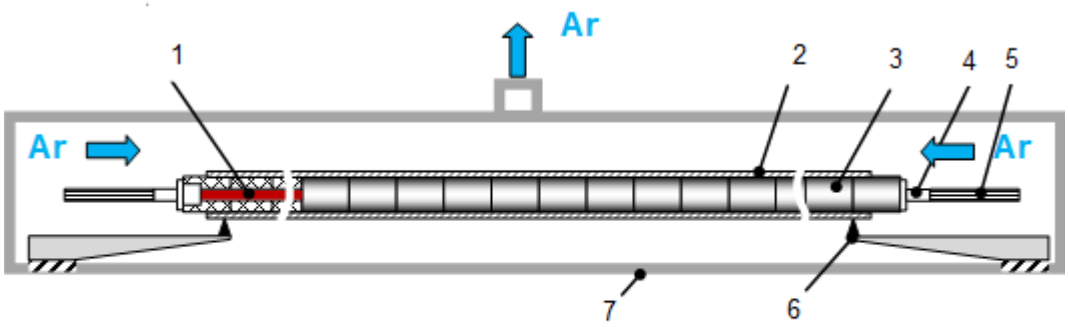


Figure 1. The Fuel Element Simulator

2.1 The fuel element simulator

The FES was constructed to represent a Pickering type fuel element with a Zircaloy-4 seamless sheath of 15.2 mm outer diameter with a wall thickness of 0.38 mm and a length of 490 mm. In the FES, the UO_2 fuel pellets were replaced with pellet simulators constructed from type AD-995 alumina ceramic supplied by the Coors Tek Company. The pellet simulators were of 14.32 mm diameter and 22.85 mm length with a 1.6 mm axial hole to accommodate the heating element. The heating element was tungsten – 25% rhenium alloy 1.5 mm in diameter. Each end of the heating wire was attached to a copper lead wire, which supplied the current from a manually controlled 10 kVA Polygon type step-down transformer. No CANLUB coating was used (as used in current CANDU reactors). No endcaps were used in the FES and the simulated pellets were free to expand axially.

2.2 The test chamber and support

The FES was supported horizontally in a 75 mm inside diameter quartz flow tube on "knife edge" supports arranged 456 mm apart. The supports were constructed of quartz tubing to reduce the heat transfer through the supports and allowed for axial expansion and rotation of the FES. An inert atmosphere was maintained during the test by a steady flow of argon gas supplied into the confinement formed by the quartz flow tube, as shown in Figure 1.

2.3 Instrumentation

The transient bow of the FES was monitored continuously during the test by measuring element sheath lateral deflection in the mid-length radial plane with a non-contact Keyence laser displacement sensor (Model LK-G502). The laser beam was focused onto the bottom of the sheath through the quartz flow tube wall. The estimated measurement accuracy for the sensor alignment is 38 μm and the laser spot diameter is 1 mm.
Type K (Chromel – Alumel) spot welded thermocouples of 36 gauge rated for a 95°C to 1260°C temperature range measured the FES outer surface transient temperatures. A total of six thermocouples were installed at 20 mm, 1/4 - length, 1/2 - length, and 3/4 - length axial positions as measured from the element left end, as viewed in Figure 1.

2.4 Test procedure and conditions

The FES was preheated to a temperature of 300°C and then subjected to a temperature transient consisting of five temperature cycles applied in succession. The first three temperature cycles were conducted at a peak temperature of 600°C and the two following cycles at the peak temperature of 800°C, as shown in Figure 2. The electrical current through the heating element was adjusted manually to maintain near-constant heating rates and the target set-point temperatures over the high-temperature plateau periods.

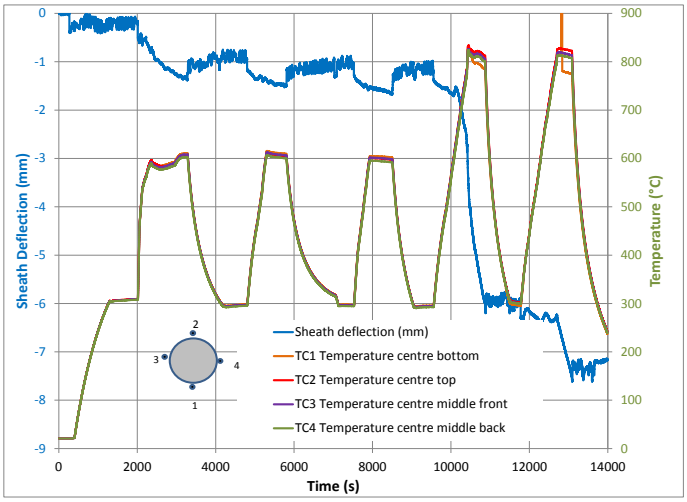


Figure 2. Experiment results showing measured sheath temperature and the deflection of the centre of the fuel pin

2.5 The test results

Figure 2 shows the measured downward deflection of the FES and the sheath temperature signals from the four thermocouples located at the FES midplane.

The total deflection from the three 600°C transients is relatively small at less than 2 mm, but at temperatures above 600°C the creep rate of the Zircaloy increases substantially, and by the end of the second 800°C transient the centre of the FES has sagged by over 7 mm. It is notable that the amount of sagging in the first 600°C transient is larger than in the subsequent second and third 600°C transients. It is also notable that the element shows some recovery during the cooldown period between these transients, and the accumulated sag at the end of these transients is less than might be expected due to this recovery. The first high temperature 800°C transient also shows a larger increase in sagging than the second 800°C transient. These features are important for comparison against the simulation results, as will become apparent later in this paper.

3. The finite element model

A three dimensional finite element model of the FES was constructed using the ANSYS finite element software package version 15.0 [4]. This model is similar to the fuel element model under development at the Chalk River site for a number of years [2]. An important aspect of these models is the complex interaction between the fuel pellets, or in this case the alumina ceramic pellet simulators, and the fuel sheathing. The ceramic pellets have a larger coefficient of thermal expansion than the Zircaloy sheath; as the pellets heat up they expand into hard contact with the sheath and in contact with each other, changing the overall stiffness of the assembly. Furthermore, the heat transfer between the pellets and the sheath is governed by the interface pressure between the components, implying that the thermal and mechanical behaviours are closely coupled. Any attempt to model the sagging behaviour of the FES must include this interaction and simulate both the thermal and mechanical behaviour simultaneously.

To reduce computational requirements, the model assumes two planes of symmetry: one extends along the axial midplane of the element (the $x = 0$ plane) and one cuts the element transversally at the midpoint (the $z = 0$ plane). These two planes of symmetry enable the fuel pin to be represented by a quarter-pin model.

3.1 The components and meshing

The components and meshing are shown in Figures 3 and 4. A major factor in the behaviour of a fuel pin is the interaction between the pellets and the sheath. For the IEC-I-1 experiment, these components are modelled as separate entities that interact via the contact modelling capabilities of the ANSYS code suite. The FES is modelled as a fuel pin with solid pellets, although the density of the pellets is adjusted to simulate the weight of the alumina pellet simulators and the hole for the heating wire. The heating effect of the wire is simulated by applying a time dependent heat generation rate to the centre elements of the pellets. The model includes endcaps (which are not present in the FES), but the model includes sufficient axial gap between the end of the pellet stack and the endcaps to ensure that the endcaps do not restrict the axial expansion of the pellet stack.

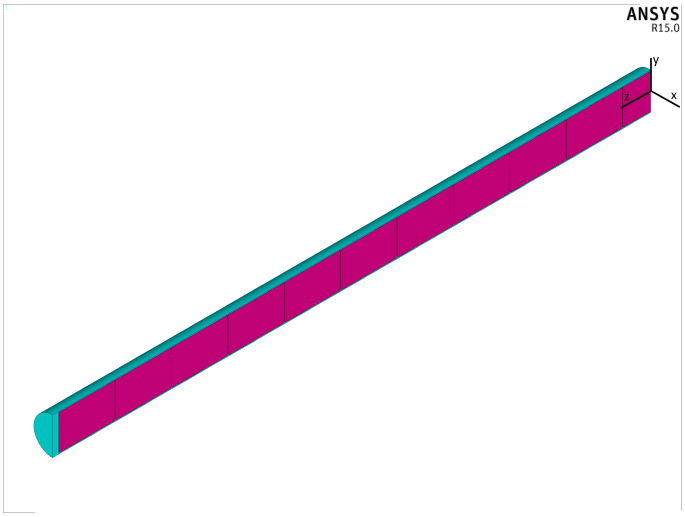


Figure 3. Full meshed fuel pin showing two planes of symmetry

The 3D solid components are meshed with hexahedral 20-node finite elements of ANSYS type SOLID226, which allow for both mechanical and thermal degrees of freedom and the simultaneously coupled mechanical and thermal solutions. Note that these simulated pellets have a chamfer, but do not have a dished end common to many fuel pellet designs. A feature that can easily be added to the model.

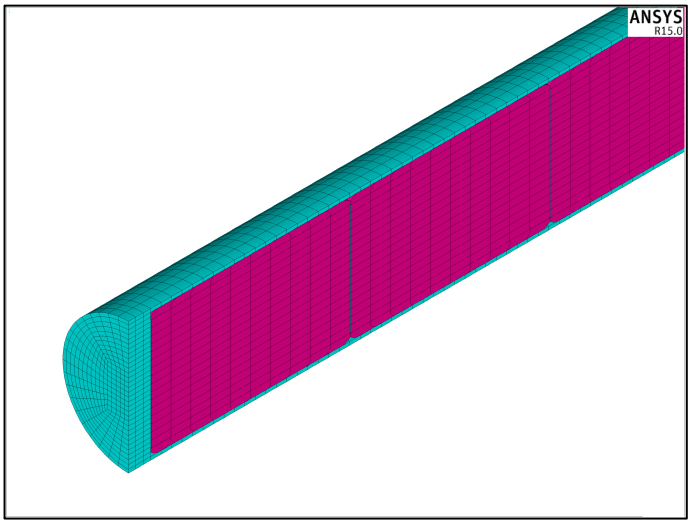


Figure 4. Close-up detail of fuel pin and mesh

Note that the modelled fuel sheath is one element thick. This would generally be considered inadequate to simulate bending moments through the wall thickness if 8-node linear elements were to be used. The use of 20-node elements overcomes this restriction, as tested during model development.

3.2 Material properties

The material properties of the fuel pellets are chosen to match those of the alumina insulators used in the FES and are obtained from the Coors Tek data sheets. They include temperature dependent specific heat capacity, thermal conductivity, Young's modulus and coefficient of thermal expansion. The density at the reference temperature of 300°C was adjusted to ensure that the weight of each pellet correctly represented the weight of the pellet simulators with the central hole for the heating wire.

The material properties of the Zircaloy-4 fuel sheathing were chosen to be consistent with the models used in the CANDU Industry Standard Toolset (IST) fuel codes ELESTRES [5] and ELOCA [6]. These models include temperature dependent specific heat capacity, thermal conductivity, Young's modulus and coefficient of thermal expansion. An important aspect of simulating the IEC-I-1 test is the creep behaviour of the fuel sheathing, and the model used is a simplification of the microstructural creep model developed for the ELOCA code [6]. The full microstructural model of Sills and Holt [7] includes deformation by a number of mechanisms including diffusional creep, dislocation creep and athermal glide, and tracks the microstructural evolution of the sheath including the swelling due to the α to β phase change. This model was implemented as a subroutine in the ELOCA code as the NIRVANA model [8].

While it would possible to couple the NIRVANA model to the ANSYS fuel model, this would substantially increase the complexity of the model and increase the expected simulation times. Instead it was noted that for the combination of stress and temperature in the sheath the mechanism of diffusional creep is dominant, and the other creep mechanisms may be ignored. The diffusional creep rate is given by:

$$\varepsilon = F\left(\frac{\sigma_a}{d}\right)^m e^{\frac{-Q}{T}} \qquad (1)$$

where:

ε = strain rate (s^{-1})
F = creep rate coefficient = $6.34\times10^6/G^2$ (G is the bulk modulus; see below)
σ_a = applied stress (MPa)
d = grain size (μm) (in the current model d = 3μm for the α-phase and 100μm for the β-phase)
m = exponent = 2.0 for the α-phase and 1.9 for the β-phase
Q = activation energy / the gas constant R (K^{-1})
T = temperature (K)
G = the bulk modulus (MPa) given by the empirical expression used in ELOCA [6]
= 1000 (36.3-0.0223(T-273))

This expression is of the Norton form and can be directly implemented in the ANSYS model.

3.3 Contact modelling

The modelling of contact between the separate components of this model is one of the most important and challenging aspects of the model, and simulating the correct sagging behaviour requires that the model correctly determine how the pellets and the sheath interact both mechanically and thermally. It is also through this interaction that the thermal and mechanical behaviours are coupled.

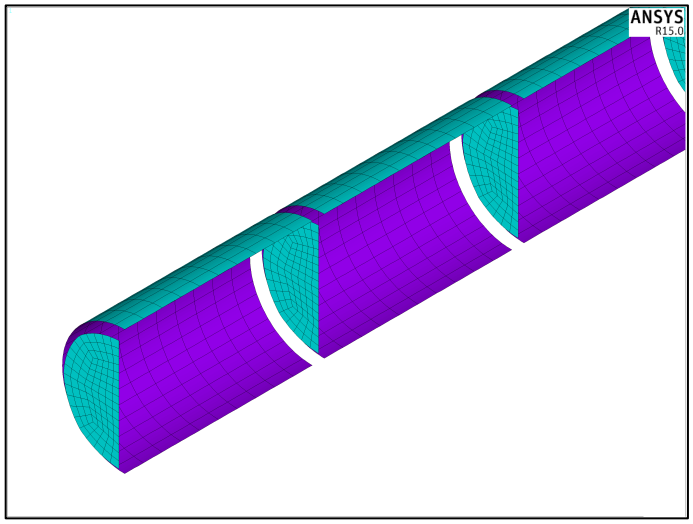


Figure 5. Contact elements

Contact modelling in ANSYS is implemented via the use of contact elements, and in this model the fuel pellets (or alumina pellet simulators) interact with each other and the sheath via contact and target elements of ANSYS type CONTA174 and TARGE170. These two dimensional elements are mapped onto the outer surfaces of the pellets and the inside surface of the sheath, and are shown in Figure 5. These elements include both mechanical and thermal degrees of freedom and determine both the mechanical forces between the surfaces and the heat transfer between the surfaces.

The mechanical contact is determined using an augmented Lagrangian method, which requires assignment of a penalty value to the intersection of overlapping contacts and targets. In ANSYS this penalty value may be specified as a factor, which is then used to determine the local penalty value from the local stiffness of the contacting surfaces. Large penalty factors result in accurate solutions with low interpenetration of the surfaces, but often result in difficulty obtaining a converged solution. Conversely, low penalty factors converge quickly but may lead to inaccurate solutions due to interpenetration of the surfaces. Experience has shown that penalty factors between 0.01 and 1 work best for these fuel models, and specifically for this model the pellet-to-sheath contacts have a penalty factor of 0.1 and the pellet-to-pellet contacts have a penalty factor of 0.01. The ANSYS contact algorithm also allows for the specification of a penetration tolerance to force the continued iteration of the solution until the penetration is less than the specified tolerance. For the fuel models of the type described here, the penetration tolerance is usually chosen to be of the order of the surface roughness of the materials, and for this model was chosen to be 1 μm.

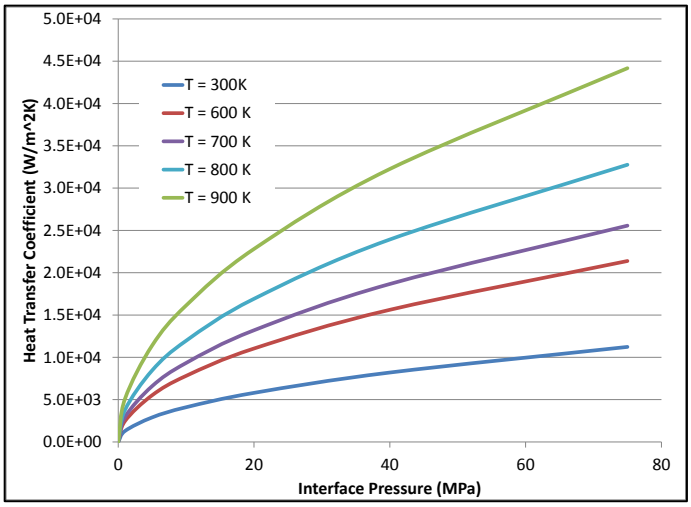


Figure 6. Pellet-to-sheath heat transfer as a function of interface pressure and temperature

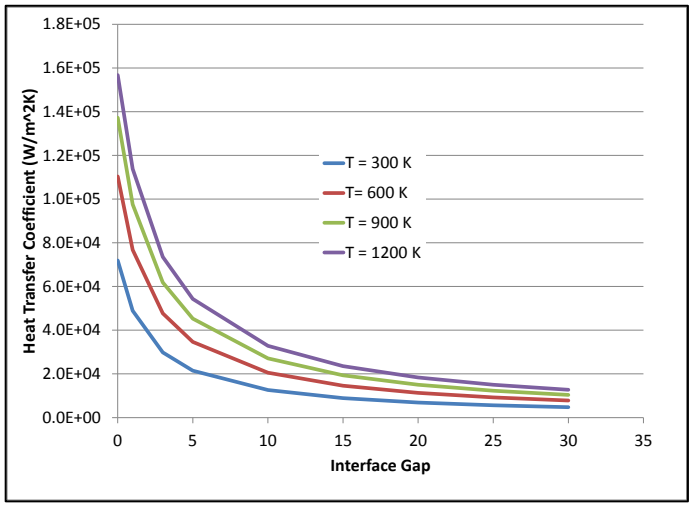


Figure 7. Pellet-to sheath heat transfer as a function of gap size

Heat transfer between the fuel pellets and the sheath is a function of the temperature of the contacting surfaces and either the interface pressure between the surfaces or the size of the gap between them. For this model, a simplified form of the Ross and Stoute [9] model is used. Figure 6 shows how the heat transfer coefficient changes as a function of temperature and interface pressure and Figure 7 shows how the coefficient of heat transfer varies as a function of temperature and gap size.

For the purposes of this model, it is assumed that the pellet-to-pellet heat transfer coefficient has the same functional form as the pellet-to-sheath heat transfer coefficient. While this

assumption may not be entirely accurate, the axial temperature gradients in this situation are small and any resulting error is expected to be minimal.

3.4 Constraints, initial conditions and boundary conditions

The two planes of symmetry of the model are implemented by imposing the constraint that the displacement in the x direction is zero for all nodes on the $x = 0$ plane, and displacement in the z direction is zero for all nodes on the $z = 0$ plane (Figure 3). The fuel pin supports are simulated by constraining to zero the y displacement of the node at the bottom of the fuel sheath closest to the support location (i.e., $z = 228$ mm) location.

The initial temperature of the entire model, and the reference temperature for thermal expansion, are assumed to be 300 K.

The two remaining time dependent boundary conditions are the measured sheath temperature and the heating rate applied via the heating wire. The model is capable of accepting spatially viable boundary conditions for the outer sheath temperature, but as shown in Figure 2, the temperature gradients at the sheath midplane are small (as determined by the difference between separate thermocouple measurements). A single spatially uniform time dependent value for the sheath temperature was applied to the model, and the applied value is shown in Figure 8.

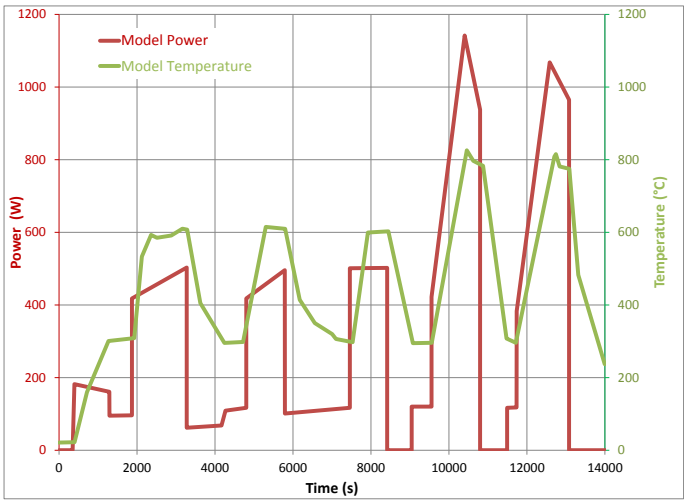


Figure 8. Sheath temperature and equivalent power applied to the model

The heating wire was not explicitly simulated in this version of the model, but its effect was simulated by applying an equivalent volumetric heat generation rate to the elements surrounding the axial centre of the fuel pellets.

3.5 Solution method

As implemented, the model requires approximately 5 GB of computer memory using the ANSYS direct matrix solver, assuming a symmetric matrix. This allows the simulation to be conducted on a desktop workstation of modest power. The simulations presented here require approximately 24 – 48 hours computation time for 6 CPU cores. A small study was conducted to confirm that the assumption of matrix symmetry was valid for these simulations. The simulations were conducted using the ANSYS automatic time step control with a minimum allowed time step of 10 s and a maximum time step of 200 s. Note that for part of the sensitivity study, smaller minimum time step sizes were used.

4 The simulation results

Figure 9 compares the results of the simulation with the measured deflection at the centre of the fuel pin. The results have some notable features. Over the first 600°C transient, the simulation results closely match the measured deflection up to the point of cooldown at approximately 3300 s. At this point, the measured deflection shows a "recovery" in the sag of the fuel pin that is not reflected in the simulation. The next two 600°C transients show very similar behaviour. Apart from the lack of "recovery" in the simulation, the sagging rates of the fuel pin during the transients match the measured values well. Possible reasons for the "recovery" observed in the experiment are discussed below.

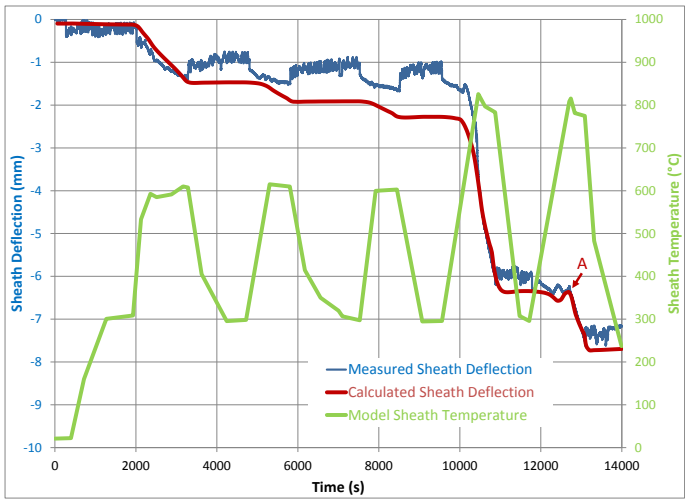


Figure 9. Results for the base case simulation

As expected, above 650°C the sagging rate increases dramatically, reflecting the increase in creep rate of the Zircaloy sheath, and this behaviour is also reflected in the model. Given the high creep rate and large deflection (>6 mm) seen in the first 800°C, the match between the simulation and measured deflection is very good, indicating that both the creep behaviour of the Zircaloy sheath and the pellet-sheath interaction have been accurately captured in the model.

The additional deflection during the second 800°C transient is less than in the first, as might be expected, as the sagging during the first transient has forced the top edges of the pellets into contact, effectively stiffening the assembly. The simulation shows some additional features due to the interaction of the pellets and the sheath, including a notable increase in sagging rate during the initial phase of cooldown. Detailed examination of the simulation results shows that this is due to the pellets contracting out of contact with the fuel sheath as they cool, reducing the stiffness of the assembly before the sheath has sufficiently cooled to reduce the creep rate. The simulation also shows a "hump" or recovery (feature A in Figure 9) during the second 800°C transient. Again this is due to the interaction of the pellets and fuel sheath. At the start of the second 800°C transient, the pellets are not in hard contact with the sheath due to the radial creep of the sheath during the first 800°C transient. As the temperature rises the pin begins to sag while the pellets are loose within the sheath. As the temperature of the pellets rises they finally come into hard contact with the sheath and begin to stiffen the assembly and pellet-to-pellet axial contact causes a small recovery in sag. As the temperature of the sheath continues to rise, the sheath softens and the sagging rate of the assembly again increases.

The simulation illustrates the complex behaviour of the pellets and sheath and the observed behaviours appear to be physically correct. Unfortunately, the resolution of the measured

results is not sufficient to confirm (or refute) that these behaviours occurred during the experiment.

4.1 Simulation including the heating wire

The attachment of the heating wire to the electrodes at the ends of the assembly was specifically designed to accommodate axial thermal expansion and contraction of the wire and to minimise its impact on the FES behaviour. However, it was speculated that the presence of the wire may be responsible for the recovery of the FES observed during the cooldown periods following the 600°C transients. The friction between the wire and the pellets and the other attachments may have been sufficient to constrain the wire as it cooled and contracted, causing it to "pull the fuel pin straight". To test this theory, a new version of the model was created in which the wire was simulated as a separate component. For expediency, the wire was modelled with a square cross section corresponding to the centre elements of the fuel pellets that were removed. Contact elements were created between the wire and pellets. The model is shown in Figure 10.

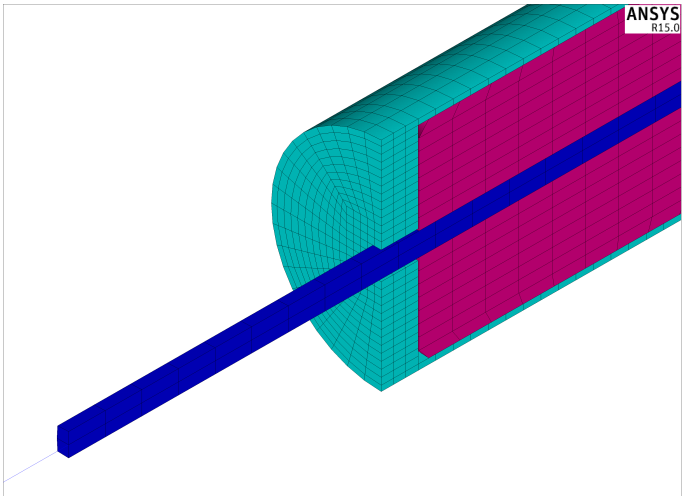


Figure 10. Model with the inclusion of the heating wire as a separate component

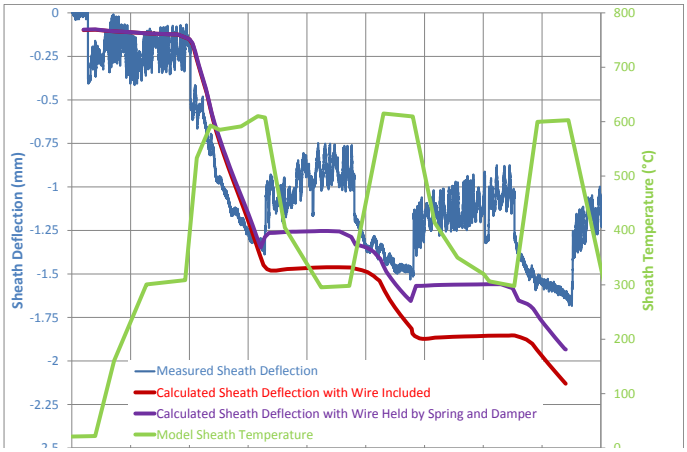


Figure 11. Results of simulation including heating wire

Figure 11 shows the results of the simulation, assuming a coefficient of friction of 1. This simulation does not display the recovery in sag observed during the experiment. In this simulation, the ends of the wire remain unconstrained. Further simulations were conducted using various constraints on the end of the wire including springs and additional frictional constraints. Some combinations did show recovery in sag during the cooldown periods of the simulation, but they also affected the overall sagging rate and none of the simulations matched the measured sag behaviour well. One such simulation is also shown in Figure 11. Further simulations using various combinations of springs and friction at the ends of the wire would likely yield a good match to the experiment, but as there is no experimentally measured way of determining this constraint, the attempts to "calibrate" the model to the experiment would be of limited value.

5. The sensitivity study

As with any modelling exercise it is important to understand how the model results respond to changes in both input and internal parameters, in order to understand both the magnitude and nature of the uncertainty in the model results. Without understanding the response of the model to these uncertainties, no confidence can be placed in the model results. In this study we looked at how the simulation results change in response to a number of parameters, of which a selected sample of the most important is presented here.

5.1 Pellet-to-sheath friction

Perhaps the most obvious unknown parameter in this model is the coefficient of friction between the pellets and the sheath, as it is expected to have a significant impact on the overall stiffness of the fuel pin assembly. Pellet-to-pellet friction also plays a role but its influence is much less than that of the pellet-to-sheath friction. Simulations were conducted varying the coefficient of friction from 1 to an almost frictionless 0.001. The results are presented in Figure 12. As can be seen, the impact on the fuel simulator deformation is relatively small over the three first 600°C transients and there is no obvious systematic variation in sagging as a function of friction. The scatter in the results may be entirely due to numerical noise. In contrast, the results show significant deviations over the large deformations of the two 800°C transients, with the higher coefficient of friction producing larger deformations. This is likely due to the friction between the pellet and the sheath causing a higher stress concentration at the pellet-to-pellet interfaces, and hence enhancing the creep rate. Best matches between the simulation and experiment are obtained for values of 0.5 or less for the coefficient of friction, but the uncertainty in the simulation results due to uncertainty in the other model parameters mean that it is not possible to make definitive conclusions.

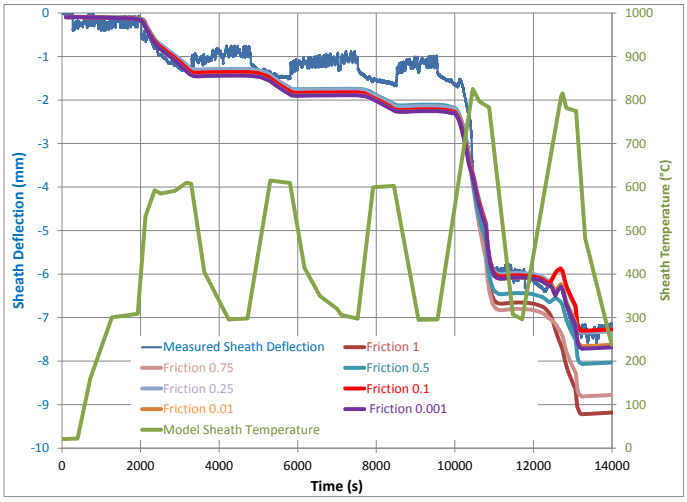


Figure 12. Results of simulation testing sensitivity to the coefficient of friction

5.2 Contact penalty factor

The contact penalty factor is an important internal parameter used in the modelling of contact between components, as described in section 3.3. If the penalty factor is too high then the model will have problems attaining convergence, and if the penalty factor is too low then the surfaces will interpenetrate and the results will be inaccurate. For a limited range of values for the penalty factor, the results should be insensitive to the value chosen. Figure 13 shows the results using a range of penalty factors for simulations with a coefficient of friction of 1. As the sheath material is substantially softer than the pellet the best results were obtained by using a larger penalty factor for the pellet-to-sheath contact than for the pellet-to-pellet interfaces. Using a single penalty factor of 1 for all surfaces the model failed to reach convergence for the high temperature transients when the pellets come into axial contact with each other. Using penalty factors of 0.1 (between pellets and sheath) and 1 (between pellets) appears to produce an accurate result, whereas values of 0.01 and 0.1 produce comparable results except that these lower values mimic the behaviour of a simulation with a lower coefficient of friction. This suggests that the penalty factor simulation has underestimated the contact pressure between the pellets and the sheath. Based on these results the optimum values of the penalty factors are 0.1 for contact between the fuel and the sheath, and 1 for contact between the pellets.

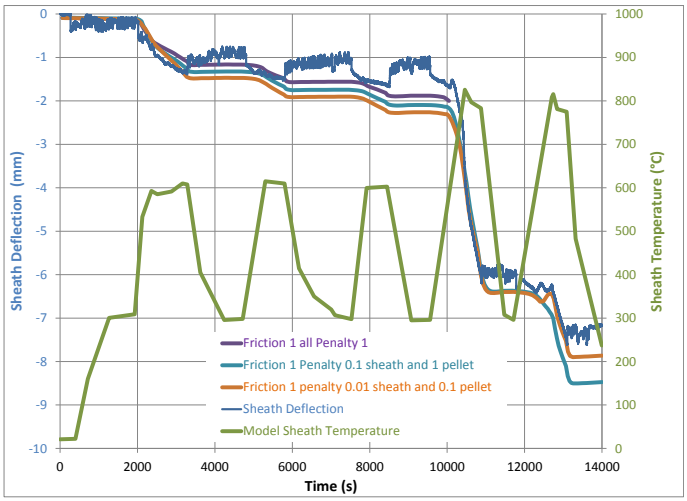


Figure 13. Results of simulation testing sensitivity to contact penalty factor

5.3 Rigid body damping

A major problem encountered in contact modelling is that of the rigid body motion of objects that become unconstrained once contact is broken. This was a major hurdle to be overcome in the development of this model. Upon cooling of the fuel the pellets contract, lose contact with the fuel sheath and become unconstrained, making convergence of the solution difficult. ANSYS includes a damping factor, termed the "opening stiffness" to damp out the motion of bodies on the opening of contact surfaces. Again the correct choice of the damping factor is vital to establish a converged solution

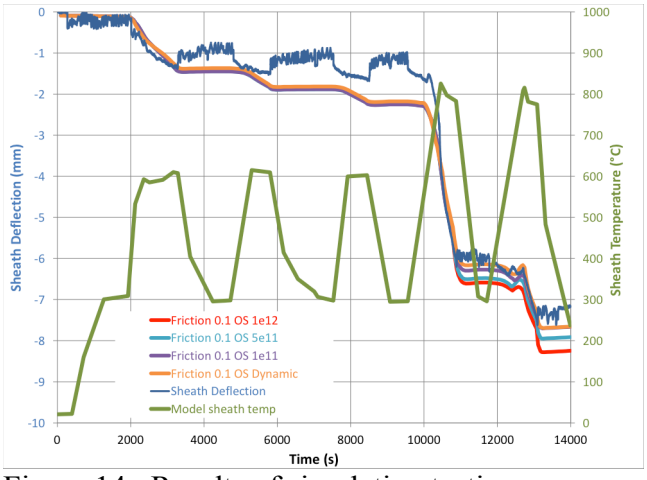


Figure 14. Results of simulation testing sensitivity to rigid body damping (opening stiffness)

without affecting the accuracy of the results. Figure 14 shows the results of the simulation using a range of damping factors between 10^{11} and 10^{12}. Use of smaller damping factors resulted in convergence difficulties. The influence of damping over the low temperature transients is minimal. Over the high temperature transients larger damping factors resulted in larger amounts of sagging, although the overall effect is relatively small, given the large amounts of deformation observed. In addition to using a constant damping factor it was also possible to make the damping factor a function of the gap between the contact surfaces. The simulation results in Figure 14 marked as "OS dynamic" used a damping factor that changed linearly from zero to 10^{12} as the gap between the surfaces changed from zero to 10 µm. The use of this variable damping factor was found to produce the best compromise that attained a stable converged solution, without unduly influencing the simulation results.

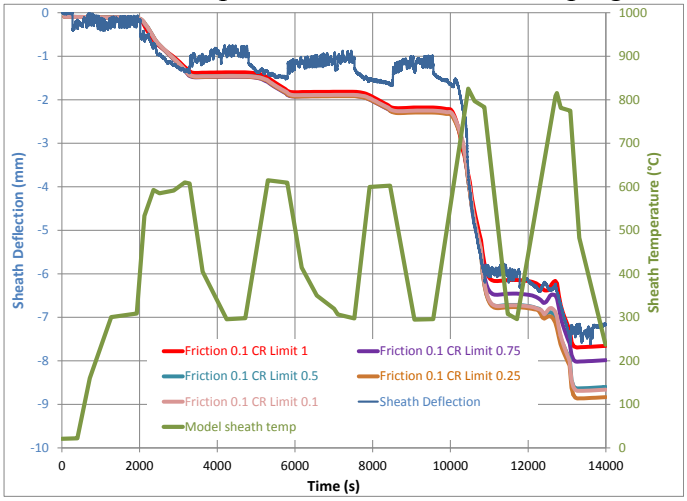


Figure 15. Results of simulation testing sensitivity to creep ratio limit

5.4 Creep ratio-limit and time step size

When simulating the behaviour of materials that undergo creep deformation, ANSYS uses the concept of the creep ratio-limit to control the time step size. The creep ratio is defined as: the increment in the creep strain over a time step divided by the current elastic strain present in the material. If the ratio exceeds a preset limit then the next time step is reduced in size. The default creep ratio limit is set at one, but for situations in which the rate of creep is high then a lower limit is required for accuracy. Figure 15 shows the sensitivity of the IEC-I-1 simulation to the creep ratio limit. Because the creep rate becomes very high during the high temperature transients, using the default creep rate limit of 1 results in an underestimate of the amount of deformation. Figure 15 shows that a creep ratio limit of 0.5 and below is required to ensure accuracy for the IEC-I-1 experiment at sheath temperatures above 700°C.

6. Conclusions

A variation of the ANSYS-based model created for CANDU fuel [2] has been successfully used to simulate the sagging under gravity of a fuel element simulator [3]. The calculated sagging rates matched sagging rates observed in the experiment, indicating that the Zircaloy creep correlation used in the ANSYS model is adequate for a gravitationally loaded fuel pin at temperatures up to 800°C. The simulations did not fully replicate the recovery from sagging observed between temperature cycles in the experiment. It is suspected that this recovery is due to the presence of the heating element used in the fuel element simulator, but it was not possible to accurately replicate this behaviour by including the heating element wire in the model. Further study on the potential source of this discrepancy is under investigation. An extensive sensitivity study was conducted to

determine the response of the simulations to a number of model parameters: friction between the components, contact penalty factor, rigid body damping (opening stiffness), and creep ratio limit. The impact of all of these parameters is effectively insignificant at sheath temperatures below 600°C, but at the higher creep rates that occur between 600 and 800°C, these parameters may have a significant impact on the simulation results. Based on the comparison of the simulation against the IEC-I-1 experiment, a contact penalty factor of 0.1 (for contact between the fuel and the sheath) and 1.0 (for contact between pellets), a rigid body damping factor between 10^{11} and 10^{12} and a creep ratio limit of 0.5 or less should be used. The sensitivity to friction between the pellets and sheath was investigated and a friction coefficient of less than 0.5 was found to produce a close match to the experimental results. A direct measurement of this friction coefficient would be desirable to confirm this result.

7. References

[1] J.M. Blair and J.I. Veeder, "The Elastic Deformation of a Circular Rod of Finite Length for an Axially Symmetric End Face Loading", Journal of Applied Mechanics, 1969 June.

[2] A.F. Williams and S. Yatabe, "An ANSYS Based 3-Dimensional Thermomechanical Model of a CANDU Fuel Pin", CW-110740-CONF-001, Submitted to AECL Nuclear Review, 2015 June 29 (for publication in 2016 June issue).

[3] A.I. Belov, R.W.L. Fong, B.W. Leitch, T. Nitheanandan and A.F. Williams, "Characterizing High-Temperature Deformation of Internally Heated Nuclear Fuel Element Simulators", CNL Nuclear Review, Vol. 5, Number 1, June 2016

[4] ANSYS Inc., 2013, "ANSYS Mechanical Application User's Guide", Release 15.0, distributed as online help with ANSYS Release 15.

[5] M. Tayal, 1987, "Modelling CANDU Fuel under Normal Operating Conditions: ELESTRES Code Description", AECL Report AECL-9331; CANDEV-86-00110, Atomic Energy of Canada Limited, Chalk River, ON, Canada, 25 p.

[6] H.E. Sills, 1979, "ELOCA: Fuel Element Behaviour During High-Temperature Transients", AECL Report AECL-6357, Atomic Energy of Canada Limited, Chalk River, ON, Canada, 88 p.

[7] H.E. Sills and R.A. Holt, 1978, "Predicting High-Temperature Transient Deformation from Microstructural Models", Proceedings of the Fourth International Conference of Zirconium in the Nuclear Industry, 1978 June 26-29, Stratford-upon-Avon, England, pp. 325-341.

[8] H.E. Sills and R.A. Holt, "NIRVANA, a high-temperature creep model for Zircaloy fuel sheathing", AECL Report AECL-6412, Atomic Energy of Canada Limited, Chalk River, ON, Canada, 1979 May.

[9] A.M. Ross and R.L. Stoute, 1962, "Heat Transfer Coefficient Between UO2 and Zircaloy-2", AECL Report AECL-1552; AECL Report CRFD-01075, Atomic Energy of Canada Limited, Chalk River, ON, Canada, 1962 June.