FUEL BUNDLE DEFORMATION MODEL

L.C. Walters[1] and A.F. Williams[2]

[1]Deformation Technology Branch
[2]Fuel and Fuel Channel Safety Branch
AECL
Chalk River Laboratories
Chalk River, Ontario
K0J 1J0

ABSTRACT

A finite-element model of a 43-element CANFLEX fuel bundle was developed using ANSYS, a commercial finite-element software package. The target application of this bundle deformation model is the prediction of the thermal and mechanical loads a fuel bundle could impart onto its pressure tube under CANDU® reactor accident transient conditions. The current version of the bundle model focuses on a small number of key components (fuel, sheath and end plates) and parameters (fuel thermal expansion, fuel/sheath interaction, and end-plate effects). This paper summarizes the current finite element fuel bundle deformation model, the testing done to date on individual components and the plans for future development.

INTRODUCTION

The objective of this paper is to present the initial development of a CANFLEX® 43-element fuel bundle deformation model using a commercial finite-element software package. The intended end product of this project will be an integrated bundle model, which will aid safety analysts in CANDU reactor licensing activity. The bundle model is currently in the early stages of development, therefore, the intention of the model described in this paper is not to be a comprehensive model, but a basic model upon which sensitivity studies can be conducted and into which phenomena may be added if deemed important in the future.

The CANFLEX fuel bundle contains 43 fuel pins (fuel elements), with 2 fuel pin sizes arranged in three rings of 21, 14 and 7 elements, respectively, around one central element [1]. Figure 1 shows a schematic of a CANFLEX 43-element fuel bundle. Thirty cylindrical UO_2 pellets are stacked end-to-end to form a fuel stack. A thin layer of graphite (CANLUB) coats the

---

CANDU® is a registered trademark of AECL.

CANFLEX® is a registered trademark of Atomic Energy of Canada Limited (AECL) and the Korean Atomic Energy Research Institute.

inside surface of the sheathing to reduce pellet/sheath interaction. The pellets are contained in a zirconium alloy (Zircaloy-4) sheathing closed at both ends by end caps. The end caps are resistance welded to the sheath extremities to (a) provide a seal for the contents of the element, (b) provide effective fuel stack termination for attachment to the end plate, and (c) provide the structural component for interfacing with the fuel handling system. Forty-three fuel pins are held together at both ends by Zircaloy-4 end plates to form a fuel bundle.

The desired separations at the transverse mid-plane of the bundle are maintained by spacers brazed to the fuel pins. The outer fuel pins have bearing pads of Zircaloy brazed on the outer-element sheaths at the middle and near the ends. The bearing pads support the bundle and protect the fuel sheaths from contact with the pressure tube under normal operating conditions.

Under accident transient conditions, the fuel pins (as part of a bundle) will be subjected to a set of thermal and mechanical loads. These loads can lead to fuel pin bowing and/or elongation with the possibility of the fuel pin contacting and transferring thermal and mechanical loads to the pressure tube. In order to predict the thermal and mechanical loads on the pressure tube under accident transient conditions a finite-element model of a CANFLEX fuel bundle has been developed using the ANSYS finite-element code [2]. The bundle model is intended to determine the overall geometrical deformation of the bundle and bowing of individual fuel pins in response to a given thermal load.

The framework of the bundle model initially concentrates on three basic components: the fuel, fuel sheath (including end caps), and end plate. The pressure tube was modeled as a non-deforming boundary. The scope of the current effort does not include modeling the effects of thermalhydraulics in the subchannels within the fuel bundle, radiation heat transfer between the fuel pins within the fuel bundle or the components beyond the pressure tube, including the pressure-tube/calandria-tube annulus, the calandria tube, garter springs, moderator, end fittings and tube sheet. Recognizing that there are many parameters that affect the behaviour of a bundle during a transient, the current version of the bundle model includes a small number of key parameters (for example, fuel and sheath temperatures, fuel thermal expansion, fuel/sheath interaction, and end-plate effects). The temperature distribution within the fuel and sheath is provided as input and the integrated mechanical response of the system is calculated based on the thermal and mechanical properties of the individual components.

THE MODELING TOOL

The ANSYS [2] computer program is a multi-purpose finite-element program which may be used for performing several classes of engineering analyses. The analysis capabilities of ANSYS include the ability to conduct static and dynamic structural analyses, steady-state and transient heat transfer problems, mode-frequency and buckling eigenvalue problems, and various types of field and coupled-field applications. The program contains many special features which allow nonlinearities or secondary effects to be included in the solution, such as plasticity, large strain, hyperelasticity, creep, swelling, large deflections, contact, stress stiffening, temperature dependency, material anisotropy, and radiation heat transfer.

The ANSYS/Mechanical Version 6.1 is the tool selected to be the "framework environment" for the bundle model. ANSYS has many built-in modeling capabilities, which means that most of the phenomena required for the bundle deformation model can be implemented without significant model development effort. ANSYS includes a full complement of nonlinear and linear elements, material laws and a comprehensive set of solvers. It can handle complex assemblies, including those involving nonlinear contact and is suitable for determining stresses, temperatures, displacements and contact pressure distributions of the type expected in a CANDU fuel bundle.

THE BUNDLE MODEL

The three-dimensional solid model of the fuel bundle was assembled from the usual ANSYS components: keypoints, lines, areas and volumes. Keypoints were placed at locations defining the boundaries of the entities in the model. Lines were created between the keypoints to define the boundaries of the areas and volumes. Volumes were used to generate the finite-element mesh for the model. The bundle model was meshed using the ANSYS SOLID5 finite element. This eight-noded hexagonal finite element has three dimensional thermal and structural field capabilities with coupling between the fields. The nodalization of this mesh is sufficiently detailed to allow for radial, azimuthal and axial temperature distributions to be applied within each fuel pin and throughout the bundle. Figure 2 shows the finite-element mesh of the fuel-bundle model. A total of 150,000 finite elements were used in this model.

Thermal expansion, radial and axial heat transfer and mechanical response in the fuel stack and sheath are modeled by defining the respective material properties for UO_2 and Zircaloy. The material properties are determined by the analyst. In the model presented here, these properties were chosen to be consistent with the existing fuel analysis codes ELESTRES and ELOCA. To account for the latent heat during a phase change, the enthalpy of the sheath was defined as a function of temperature.

Surface-to-surface contact problems are modeled using the ANSYS general contact elements. Friction, specific to each contact pair as determined by the analyst, was modeled between the fuel and the sheath and between the spacers of neighbouring fuel pins. If the friction between the surfaces is sufficiently large, the two materials will behave much like a layered composite material in which strain values in one component are matched in the other. On the other hand, if the interfacial friction is low, the two materials will not behave as a composite, but rather as two separate objects in simple contact. In the models presented here, the coefficient of friction between the fuel and the sheath was selected to be 0.2. Between the spacers of neighbouring fuel pins the coefficient of friction was estimated (no reported data available) to be 0.1.

At this stage in the development of the bundle model, the modeling of the fuel was greatly simplified to be one continuous, solid cylinder instead of thirty individual pellets stacked end-to-end inside the sheath. A detailed model of the fuel is not included, not because it is inherently difficult to model, but rather to reduce the complexity of the model. This assumption that the fuel pellets act as a monolithic fuel stack neglects the thermal resistance at the interface between the pellets and the friction/slippage between the pellets in cases of buckling and bowing. The fuel was modeled in contact with the bottom inside surface of the sheath. A gap between the top of the fuel and the sheath of 0.07 mm was assumed present. The diametral clearance between the CANLUB-coated sheath and fuel was between 0.04 mm and 0.11 mm throughout the fuel pin length (tolerance on fabrication dimensions). The axial clearance between the end of the fuel and the end caps was between 1.0 mm and 3.0 mm.

The presence of CANLUB coating on the inside surface of the sheath is expected to alter the characteristics of the fuel stack / sheath friction interaction by lowering the interfacial friction. In the presence of significant sagging, bending or bowing of the fuel pin, the mechanical interaction between the fuel stack and the sheath becomes more important. The two dissimilar materials will deform with different characteristics, leading to slippage between the fuel and sheath. The friction between these two surfaces will influence the degree of slipping, or lack thereof. The friction between the fuel stack and sheath is a function of a number of parameters, including fuel surface roughness, sheath inside surface roughness, sheath phase (alpha, beta), presence of surface oxidation, interfacial pressure, local material temperatures, differential slippage velocity and local presence or absence of CANLUB coating.

The end caps are modeled assuming perfect contact with the sheath. The junction between the end cap and the end plate is a continuous metal interface due to the end cap/end plate weld. Heat will flow through this as it would through normal Zircaloy, and loads will be transmitted as well. Details such as chamfers and rounded edges on the end plates have not been modeled. If calculations with this version of the bundle model indicate that high stresses are present in the end plate, then fillets will be added to the model.

To model the pressure tube as a non-deforming boundary, constraints were placed on all the nodes of the bearing pads so that no deformation in the radial direction could occur.

Boundary conditions were used to account for subchannel thermalhydraulic effects and the heat generated in the sheath as the result of radiation heat transfer and zirconium-steam reaction at high temperatures. No radiative heat-transfer elements were used to impose radiative boundary conditions. Boundary conditions were also used to account for fission-gas pressure applied on the inside surface of the sheath and the coolant pressure applied on the outside surface of the sheath. The temperature and pressure boundary conditions were applied to the finite-element nodes via input tables. These tables (in ASCII format) can be produced by external codes. This approach limits the bundle model as it is dependent on external codes and their range of applicability.

Once the first development stage of the bundle model was completed, i.e., all the required components and associated phenomena were modeled, a simple test case was performed to demonstrate the qualitative behaviour of the model. In this test, the fuel bundle was constrained in the vertical direction only at the bearing pads to simulate a bundle resting on a non-deforming pressure tube. A constant temperature of 300°C was applied to the entire bundle (reference temperature of 25°C). Figures 3 and 4 show the deformation of the fuel bundle due to thermal expansion. In Figure 3 the dashed lines show the undeformed boundaries of the model and the solid colours represent the deformation of the model. The fuel bundle uniformly expands by a fraction of a millimetre. This simple test case demonstrates the feasibility of the ANSYS finite-element package and the modeling techniques towards further development of the fuel bundle deformation model. Sensitivity and verification studies using the full model or separate components can now be used to identify which phenomena and feedback mechanisms will need to be further developed.

TESTING OF THE FUEL PIN MODEL

Suitable data for verifying thermally induced bow, caused by a circumferential temperature distribution around a fuel pin, was found in [3]. Bow is the magnitude of deflection measured normal to the restraint plane. A fuel element simulator with an off-set tungsten heater was developed and used to study the influence of major parameters affecting bow up to a maximum sheath temperature of 600°C. The bottom sheath temperature was held constant during each test at ~300°C to represent the reactor coolant conditions. The distance between the points of restraint was held constant at 0.25 m. The results show that circumferential temperature distribution, pellet-to-sheath mechanical interaction and creep are the major factors affecting bow.

In the three sets of experiments performed (grouped together based on geometry, see Table 1), it was found that transient bow increases with top-to-bottom sheath temperature difference, the bottom sheath temperature and mechanical interaction between the pellet and the sheath. Within each set of tests, the transient bow in the fuel pin was measured using various top-to-bottom sheath temperature differences and different bottom sheath temperatures (ranging from 264°C to 312°C throughout the three sets of tests). Transient bow for a top-to-bottom temperature difference of 300°C, was about 0.6 mm for a large diametral gap of 0.82 mm, corresponding to low mechanical interaction. For the same conditions the bow was about 1.5 mm for a smaller gap of 0.004 mm, corresponding to strong mechanical interaction.

A finite-element model of a single fuel pin was developed using geometrical dimensions for the fuel and the sheath consistent with those used in the fuel element bow tests, summarized in Table 1. In the experiments, eighteen fuel pellets were placed end to end inside the sheath. However, in the single fuel pin model, the eighteen fuel pellets were modeled as a solid monolithic cylinder 0.27 m long. A large part of the motivation for comparison against the experimental data was to determine the limits of the solid fuel model and determine if it is necessary to include separate pellets into the model. The sheath, modeled slightly longer than the fuel 0.29 m long, was constrained in the vertical direction at two points near the ends of the sheath, 0.25 m apart. Temperature of the fuel and the sheath was input from experimental data.

Results from Tests 1 and 2 compare well with the experimentally measured transient bow as shown in Figures 5 and 6, respectively. In these experiments the eighteen fuel pellets thermally expanded across the small fuel/sheath gap (0.02 mm in Test 1 and 0.004 mm in Test 2) into contact with the sheath, interacted with the sheath, and remained in axial contact with each other. The fuel/sheath then acts as a layered composite material, in that strain values in the fuel are matched in the sheath. Therefore, modeling the eighteen fuel pellets as a monolithic stack adequately approximated the behaviour of the fuel.

The simulations of Test 3, where the fuel was once again modeled as a monolithic fuel stack, do not compare well with experimental results, as shown in Figure 7. The simulations over-predict the transient bow. In this case, the fuel/sheath gap is larger (0.082 mm) and, therefore, the mechanical interaction between the fuel and sheath was likely low. Subsequently, the pellets did not come into tight contact with the sheath or remain in axial contact with each other. Instead of driving the deformation in the fuel pin, the fuel pellets acted as individual weights reducing the overall transient bow. Therefore, the monolithic fuel stack was not a good approximation of the fuel and it was necessary to develop a detailed fuel pellet model.

The modeling of fuel pin was refined to include all eighteen separate fuel pellets. The dimensions of the modeled sheath and gap remained consistent with the geometry used in the experiment. The pellets were modeled without dishes at the ends. Contact elements were modeled between the pellets and the sheath but not between each individual pellet and its neighbouring pellets. Effects such as "pellet hourglassing" caused by the thermal expansion of the fuel pellet have not been modeled at this stage. Results from simulations with the fuel pellets produced results consistent with experimental data from Test 3, as shown in Figure 7.

The results from the single fuel pin verification study demonstrate that when mechanical interaction between the fuel and the sheath is high, the fuel and sheath act as a composite layered material capable of producing a large thermally induced bow. A solid, monolithic fuel stack successfully emulates the response of the fuel under this condition. However, when the diametral gap is similar to that of the CANFLEX bundle and the mechanical interaction between the fuel and the sheath is low, the two materials do not act as a layered composite material and the thermally induced bow is smaller. Consequently, individual fuel pellets must be modeled to capture the response of a fuel pin under accident transient conditions.

CONCLUSIONS

A finite-element model of a 43-element CANFLEX fuel bundle was generated using ANSYS, a commercial finite-element software package. The target application of the bundle deformation model is the prediction of thermal and mechanical loads a fuel bundle could impart onto its pressure tube under CANDU reactor accident transient conditions. Test cases where a thermal load was applied to the model have shown that the full fuel bundle model, in this initial stage of development, demonstrates the correct qualitative response. Simulations of a single fuel pin under thermal loads compared well with experimental data. These verification tests have shown that under transient accident conditions, a small diametral gap between the fuel and the sheath leads to high mechanical interaction where the sheath and the fuel come into tight contact and the two materials deform as a composite layered material. This scenario has been successfully modeled using a solid, monolithic fuel stack. However, when the diametral gap is large and the mechanical interaction between the fuel and sheath is low, the fuel pellets act as individual bodies and must be modeled as such. A model using individual pellets produced results that were in very good agreement with experimental data, even though details such as pellet dishes and pellet hourglassing were not modeled.

Future development of the bundle model will depend on feedback from potential users of the model and the availability of experimental and operational data for further testing of components and phenomena. Sensitivity studies will also be performed to identify which phenomena and feedback mechanisms will need to be developed further.

REFERENCES

1. W.R. Inch and P. Alavi, "CANFLEX MK-IV Qualification Program and Readiness for Implementation," Seventh International Conference on CANDU Fuel, 2001 September 23-27, Kingston, Ontario.

2. ANSYS Inc., "ANSYS Theory Reference 6.1", Canonsburg, PA, 2002 April.

3. P.M. Mathew, S. Baset, J. Veeder and E. Kohn, "Quantification of Factors Affecting Thermally-Induced Bow in CANDU Fuel Element Simulator", 5th International CNS CANDU Fuel Conference, 1997, p.342.

Table 1. Dimensions used in the ANSYS Simulations of the Fuel Element Bow Tests [3]

Test	Pellet OD	Sheath ID	Sheath OD	Pellet/Sheath Gap
1	14.392 mm	14.412 mm	15.233 mm	0.02 mm
2	14.392 mm	14.399 mm	15.210 mm	0.004 mm
3	14.330 mm	14.412 mm	15.233 mm	0.082 mm

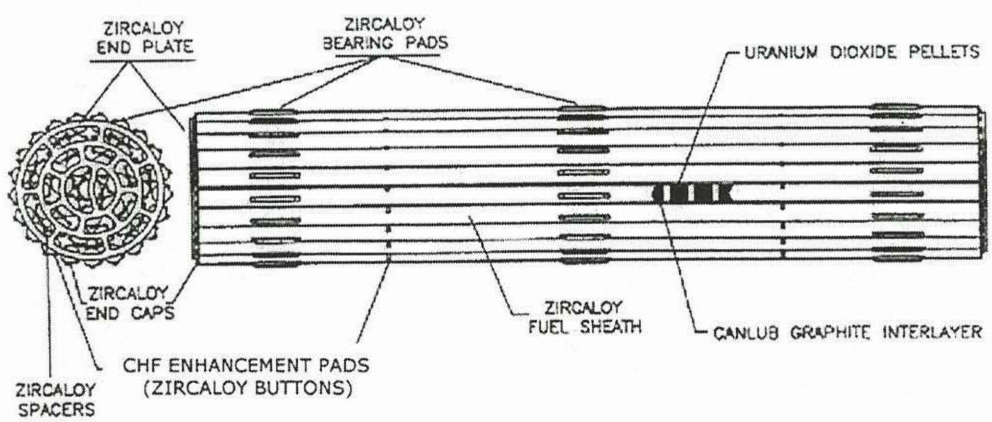

Figure 1. Schematic of a CANFLEX 43-Element Fuel Bundle

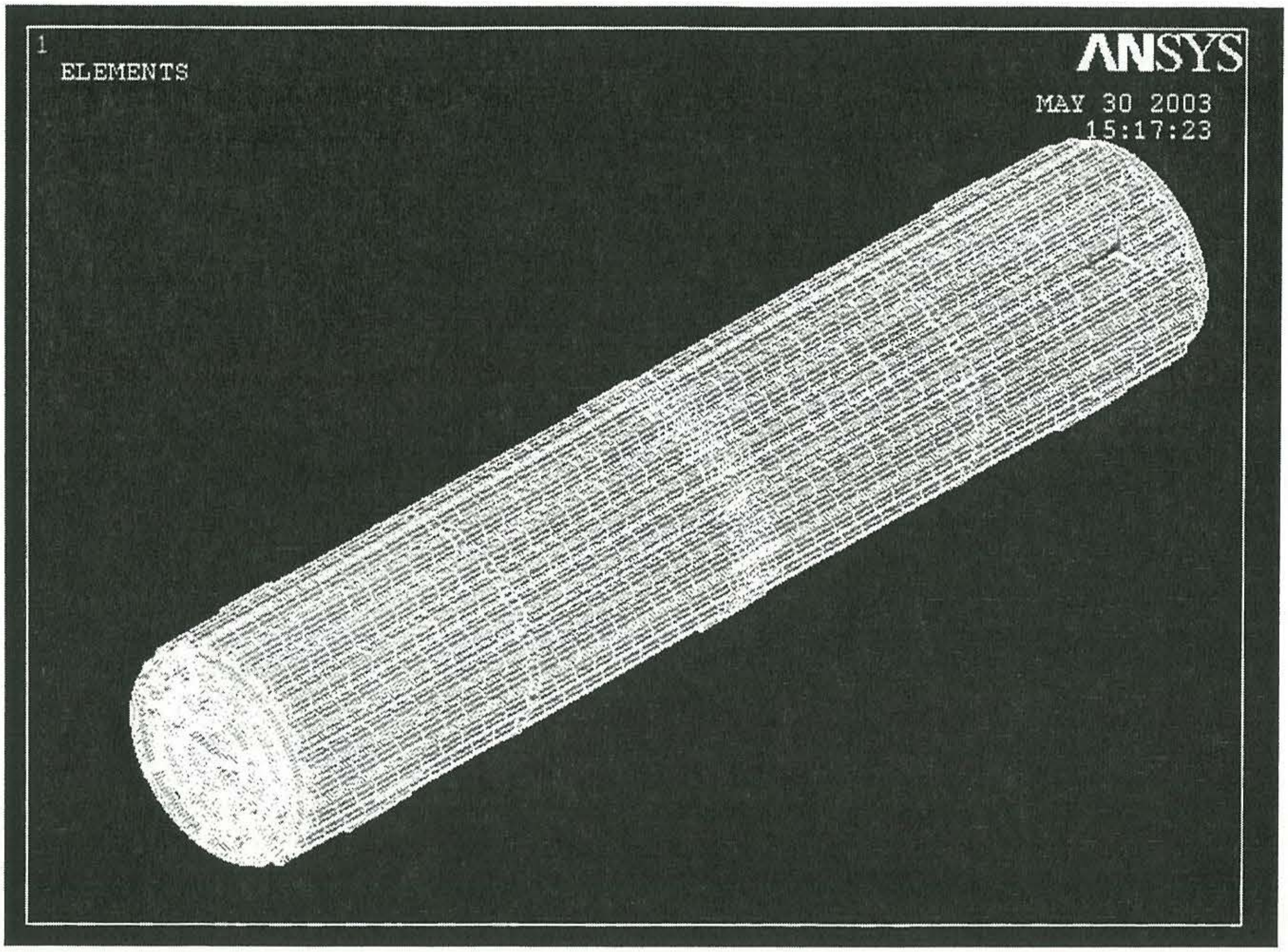

Figure 2. Finite Element Mesh of the 43-Element Fuel Bundle Model

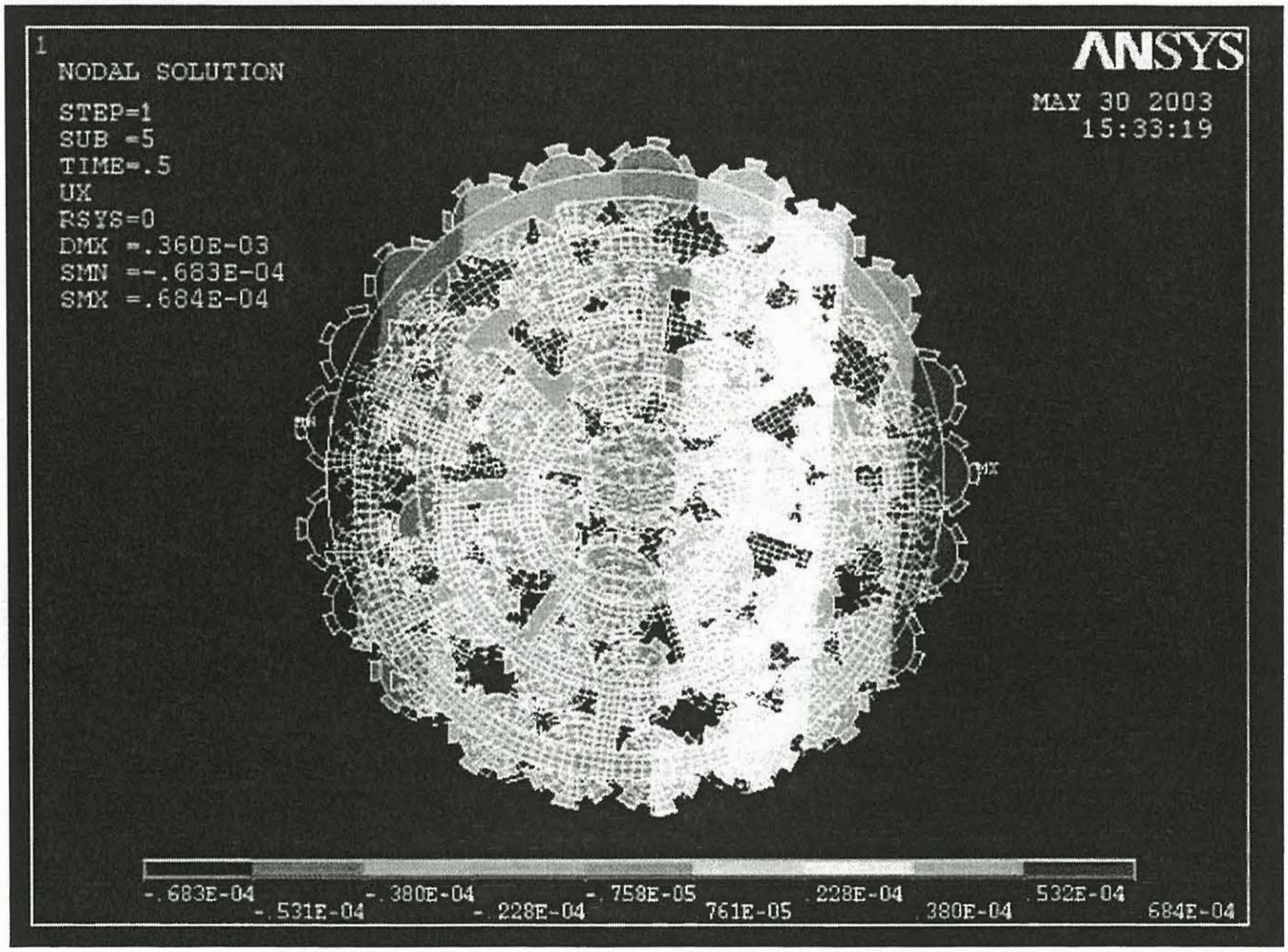

Figure 3. Horizontal Component of Displacement From the Full Bundle Test Case
(Front View, Bundle Temperature 300°C, Constraint on the Bottom Bearing Pad)

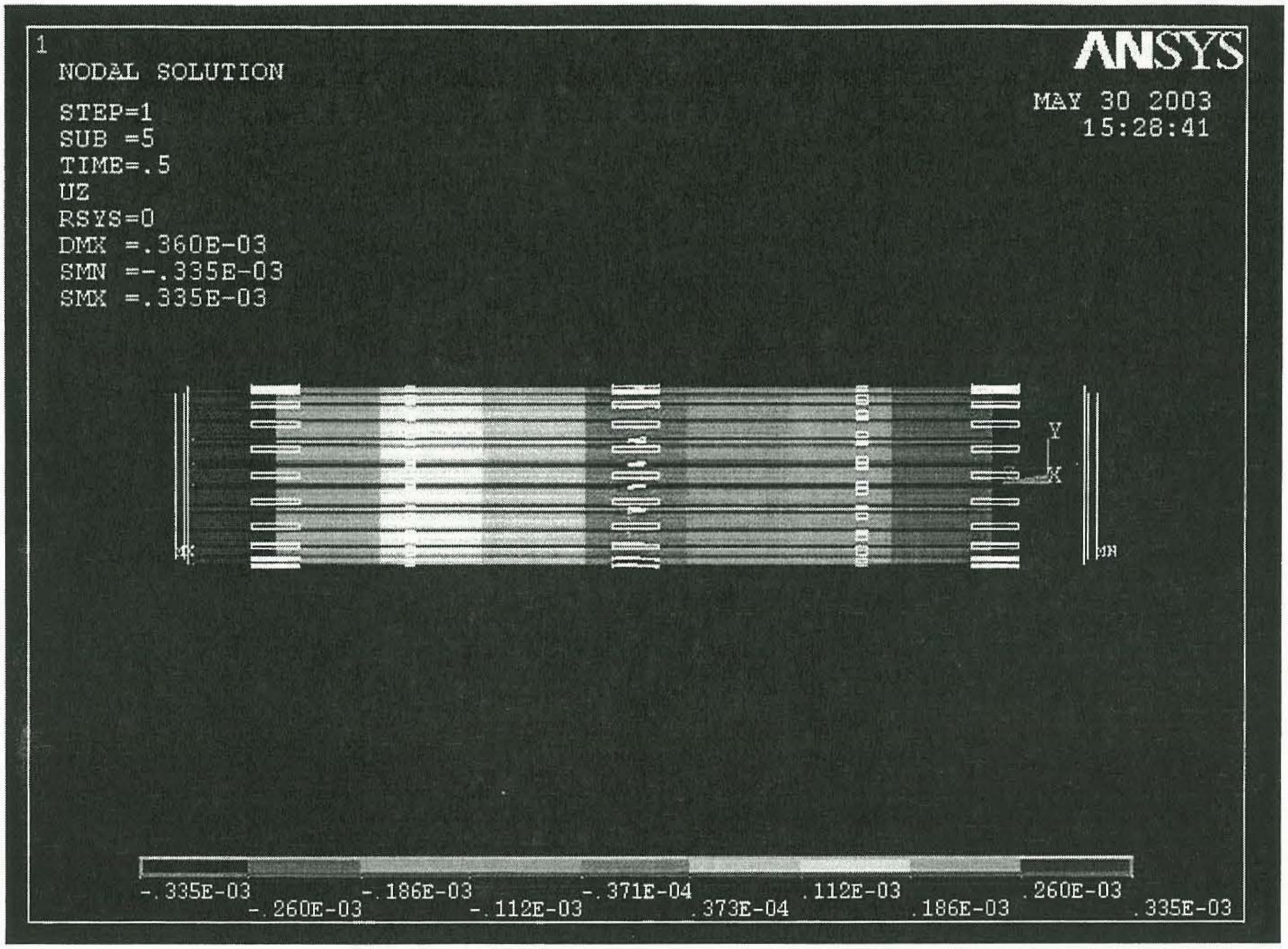

Figure 4. Displacement in the Axial Direction From the Full Bundle Test Case (Side View, Bundle Temperature 300°C, Constraint on the Bottom Bearing Pad)

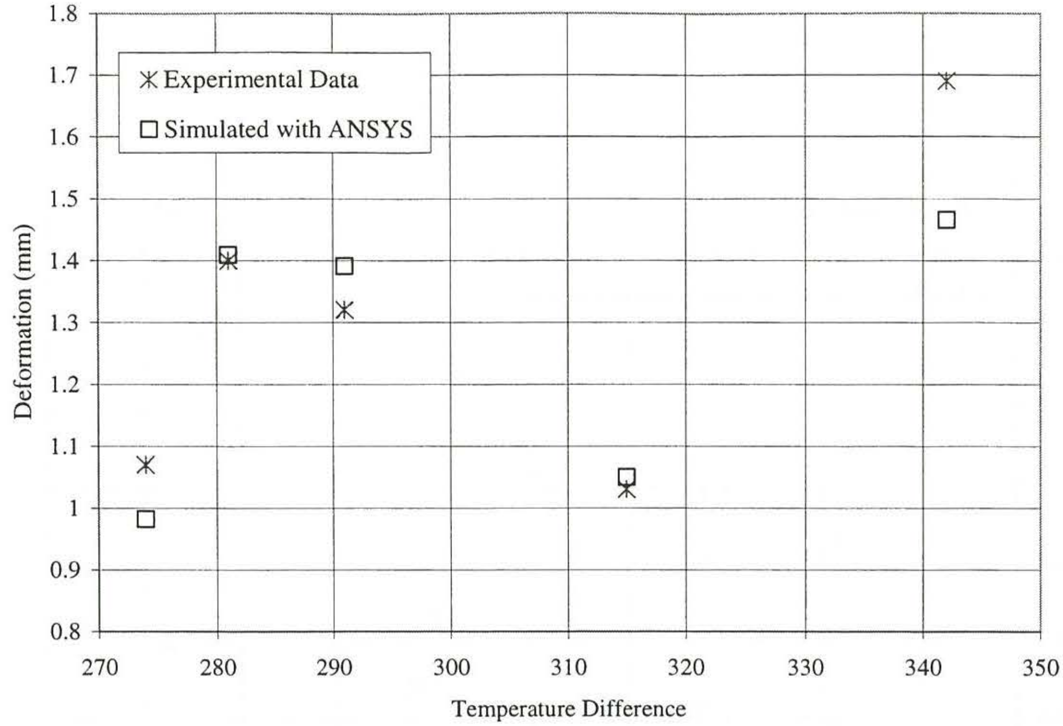

Figure 5. Comparison of Experimental Data from the Fuel Element Bow Test #1 (Fuel/Sheath Gap = 0.02 mm) [3] and the Finite Element Simulation

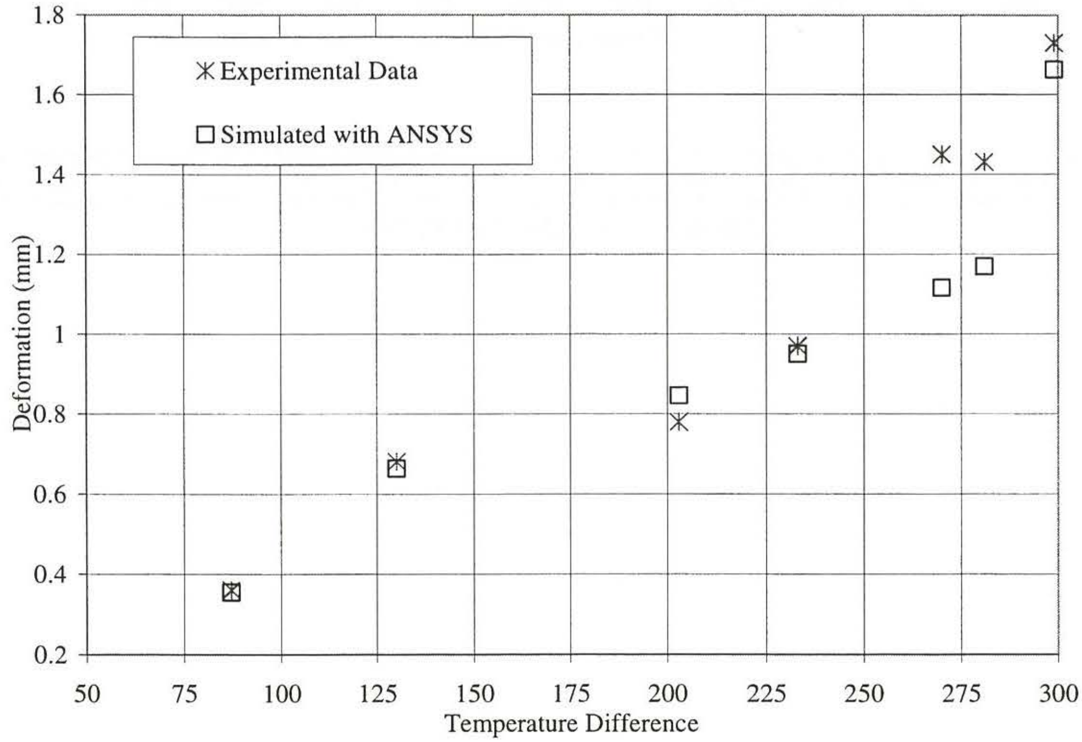

Figure 6. Comparison of Experimental Data from the Fuel Element Bow Test #2 (Fuel/Sheath Gap = 0.004 mm) [3] and the Finite Element Simulation

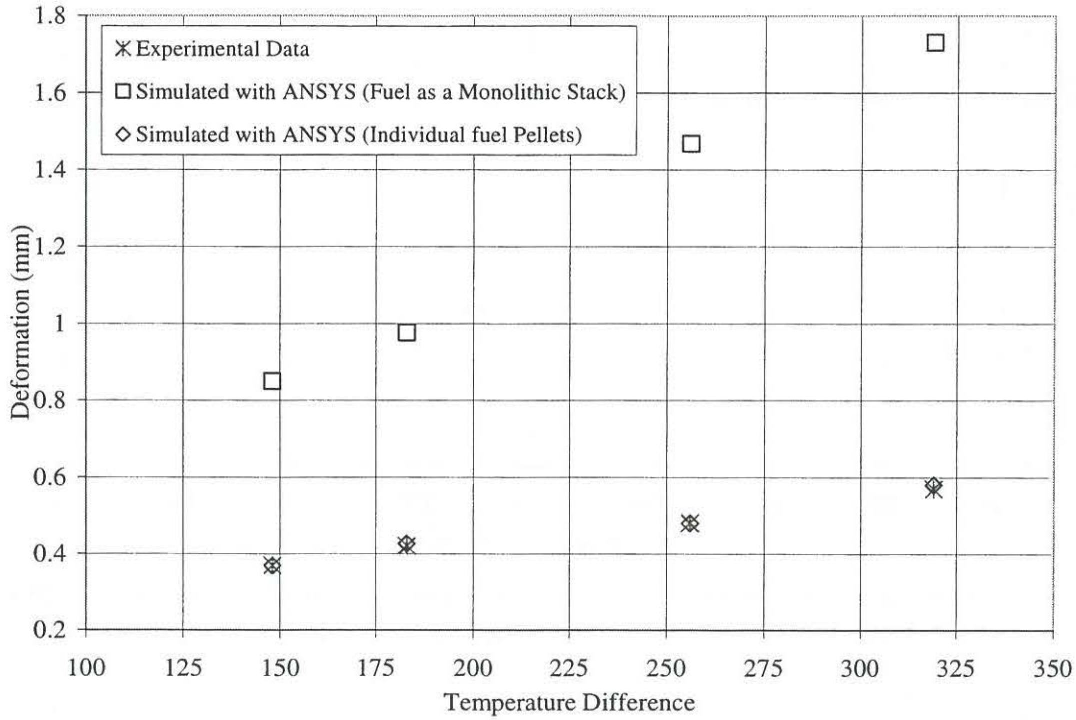

Figure 7. Comparison of Experimental Data from the Fuel Element Bow Test #3 (Fuel/Sheath Gap = 0.082 mm) [3] and the Finite Element Simulation with the Fuel Modeled as a Monolithic Stack and as Individual Pellets