Simulation and Analysis of the Delivery Machine in a Sagged Nuclear Fuel Channel

Tong Zhou

Atomic Energy of Canada Limited (AECL)

Abstract

In CANDU® nuclear reactors, the pressure tube (PT) of a fuel channel will progressively sag, elongate and expand in diameter due to creep and growth of the tube wall. The AECL designed Advanced Delivery Machine (ADM), which is used to deliver fuel channel inspection tools and consists of three straight ram tubes (RTs), must be qualified for use at the end-of-life fuel channel configuration. During inspections, significant contacts occur between the straight RTs and the sagged PT. In this paper, a finite-element model is developed using ANSYS to simulate the inspection processes and the contact situations. The results show reasonable RT-PT behaviours and interactions, as well as acceptable stresses on both the PT and the RTs. Comparisons between analytical results and experimental data also validate the ANSYS model.

Introduction

As part of the AECL fuel channel inspection system, the ADM is used to deliver fuel channel inspection tools. The ADM system uses a three-stage telescoping ram to move the inspection tools through the length of a PT. Over its operating life, the PT will progressively sag, elongate and expand in diameter due to creep and growth of the tube wall. The original ADM was designed for PTs early in their operating life. However, inspections are required to be performed on PTs at all stages of their operating lifetime. The current ADM design has not been qualified for use at or near an end-of-life channel configuration.

Due to the sag of PTs, both the PT and RTs may be subjected to significant bending loads at the ADM fully extended position. In order to qualify our ADM design, it is essential to demonstrate analytically and experimentally that during PT inspections, no damage would occur to either the RTs or the PTs. Therefore, stress analyses and tests on a full-scale highly sagged fuel channel are required to confirm RT-PT integrity, and also to confirm acceptable ADM performance during the RT extension and retraction processes.

This paper summarizes the first part of the ADM qualification work, i.e., the development of a computer model of the ADM RT assembly and a sagged PT, and the analysis of their contacts, stresses and deflections. This analysis also provides a baseline for the second part of the qualification work (i.e., the actual tests), and provides recommendations for the set-up of test rig (e.g., optimal measurement locations).

FEA Model Development

The FEA model is developed with ANSYS 5.5.3 Multiphysics, and it consists of two parts: a sagged PT and the ADM RT assembly. Contact pairs are used at the interfaces between these two parts. The following sub-sections provide detailed descriptions of the model development.

ANSYS Model of a Sagged PT

Based on design specifications, a PT has a nominal diameter of 108.23 mm with an average wall thickness of 4.45 mm and a total length of 6452 mm. The Young's modulus (E) and Poisson's ratio (ν) used for the PT (i.e. Zr-2.5Nb) are 94.46 GPa and 0.39, respectively [1]. The maximum sag reaches 82.70 mm at the middle of the PT. The sagged profile corresponds to the fuel channel at its end-of-life configuration, that is after 210k effective full-power hours of operation, see Figure 1.

3-D linear shell elements (i.e., SHELL63, 4-node elastic shell elements) are used for PT meshing. Mapped meshing is achieved by using a linear quadrilateral element shape.

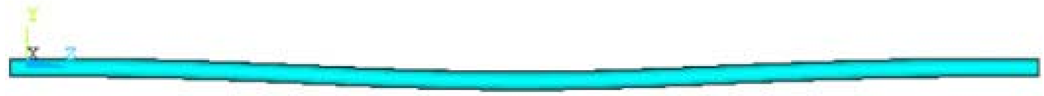

Figure 1 - Sagged PT Model

The boundary conditions (B.C.s) applied to the PT define one end as being clamped, and the other end as being pinned with free movement along the axial direction. The PT's mass is not included in the modeling, because the deformation caused by gravity has been integrated into its sagged profile.

ANSYS Model of the ADM RT Assembly

The ADM system consists of three nested RTs, namely RT1, RT2 and RT3, respectively, from inner to outer. The RTs are straight and relatively flexible comparing with the PT, so that during an inspection, they can be bent to fit the sagged PT profile. By reviewing the design drawings and the operating processes, it can seen that Ram Tube 3 (RT3) will not contact the PT, even when the ADM is at its fully extended position. RT3 only provides a clamping B.C. for RT2. Meanwhile, during ADM extension, RT1 and RT2 are not supported by any other means except RT3 and contacts with the PT. Therefore, only RT1 and RT2 are included in the analysis, and are modelled as a cantilever beam. RT1 has a nominal diameter of 44.45 mm with 6.35 mm wall thickness and 3508 mm total length, while RT2 has a nominal diameter of 69.60 mm with 6.58 mm wall thickness and 3665 mm total length. The material properties are chosen as E = 196.5 GPa, and ν = 0.29 [2].

In addition to RT1 and RT2, there are several other components in the ADM RT assembly: the inspection head, the locking tool that provides resistance to allow the three RTs to lock, and the RT2 retainer that keeps RT1 in RT2. The inspection head has two universal joints that allow some compliance, so that it does not produce any bending stress on either the PT or the RTs. Therefore, the inspection head is eliminated from this analysis. Due to the fact that the nominal diameter is used in modelling the PT, the outer diameters (ODs) of both the locking tool and the RT2 retainer are adjusted by the wall thickness of the PT to simulate the actual contacts. Thus the locking tool has an effective outer diameter of 106.05 mm with 393.70 mm length, and the RT2 retainer has an effective outer diameter of 98.43 mm with 83.82 mm length. It is also assumed that these components are much more rigid than the RTs and the PT. Therefore, an arbitrarily large Young's modulus is selected for both of them. The solid model of the entire ADM RT assembly is shown in Figure 2.

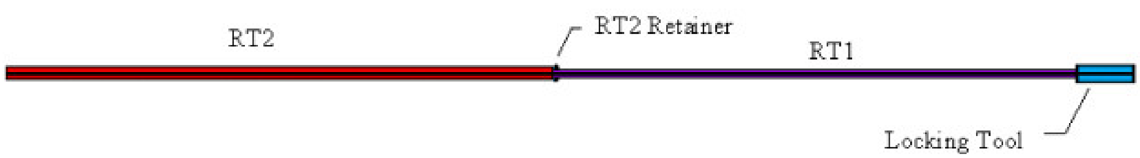

Figure 2 - ADM RT Assembly Model

3-D linear shell elements (i.e., SHELL63, 4-node elastic shell elements) are used for meshing the ADM RT assembly. Mapped meshing is achieved by using linear quadrilateral element shape. The meshed RT2 retainer assembly is shown in Figure 3. The locking tool has a diametrally floatable distance with respect to RT1. This floatable distance reduces the severity of contacts between the locking tool and the PT. It is modeled by two gap elements (i.e., COMBIN40, 2-node 1-D combination elements). The gap's initial width is set to the maximum floatable distance. When its width reduces to zero, contact occurs and RT1 moves together with the locking tool.

The B.C.s applied to the RT assembly are as follows: the locking tool end is free and the other end of RT2 is clamped, which form a cantilever beam. During simulation, the clamped end is forced to move axially into the sagged PT until the inspection head passes the whole length of the PT. The masses of the RT assembly are not included in the modeling due to the lack of related information.

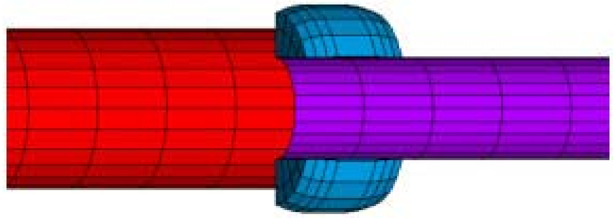

Figure 3 - Meshed RT2 Retainer with Sections of RT1 and RT2 Solid Contacts

During ADM extension/retraction, solid contacts occur at various locations between the ADM RT assembly and the PT. To capture all possible contacts, eight pairs of contact surfaces are defined in the ANSYS model:

1. locking tool upper half vs. PT upper half
2. locking tool lower half vs. PT lower half
3. RT1 upper half vs. PT upper half
4. RT1 lower half vs. PT lower half
5. RT2 retainer lower half vs. liner tube lower half
6. RT2 retainer lower half vs. PT lower half
7. RT2 retainer upper half vs. PT upper half
8. RT2 lower half vs. PT lower half

For contact pair 5, the liner tube lies in the fuel channel end fitting and provides support to the RT2 retainer when RT2 is inserted into the end fitting. Since the liner tube is not within the scope of this analysis, a small rigid surface is created to simulate its bottom part and to provide a restraint on the RT2 retainer.

The contact pairs are meshed with TARGE170 and CONTA173 surface elements. With the exception of contact pair 5, which is a rigid-surface-to-flexible-surface contact, the rest contact pairs are flexible-surface-to-flexible-surface contacts. For the contacts between the PT and the RTs, the wall thicknesses are taken into consideration by shifting the contact surfaces from the mid-plane to the top (or bottom) surfaces of the shell elements. For the contacts between the locking tool and the PT, or between the RT2 retainer and the PT, the wall thicknesses have been included in the effective outer diameters, as given in the above sub-section. A typical coefficient of friction of 0.3 is used to account for the friction stress.

The initial position of the ADM RT assembly in the PT is established in such a way that the locking tool just touches the upper part of the PT, as shown in Figure 4. A small initial penetration exists between the contact surfaces.

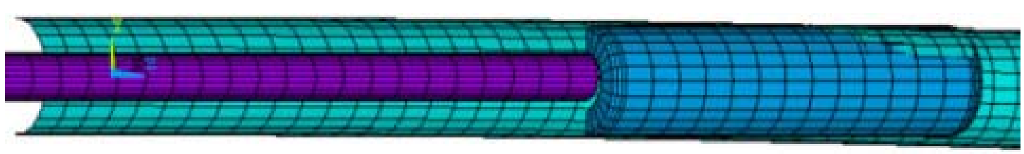

Figure 4 - Initial Contact between Locking Tool and PT

Summary of FEA Modeling

The models created in the above sub-sections are symmetrical about the PT sagging plane. To save computer resources, only half of the geometry is considered and a symmetrical B.C. is applied at the PT sagging plane.

The final ANSYS model consists of a total of 6144 shell elements, 12805 contact elements and 2 gap elements, which results in a grand total of 18951 elements and 6699 nodes. The duration of a typical run is about 12 to 14 hours on an SGI64 dual-processor computer with UNIX operating system, and each run takes about 1 GB of hard disk space.

Analysis Results and Discussions

The validation of FEA modeling can be achieved in two ways: a) using engineering judgement to see if the RT/PT interactions and behaviours are reasonable, and b) using test results for comparison to see if the analytical results match the experimental data. In this section, both validation methods are provided to show that the ANSYS model is acceptable.

The simulations start at the "just-touching" position as shown in Figure 4. This initial axial position of the RT assembly is set to be zero in the following results. The RT assembly is then moved axially 5334 mm into the sagged PT from the initial position. The FEA is carried out at a series of positional snapshots of the RT assembly as it moves in the PT. Forty-two sets of FEA results are recorded, at roughly one data set per 127 mm of RT movement. Some nonlinear solution options are used, such as "SOLCONTROL, ON", "PRED,ON,,ON", "NLGEOM,ON", and "LNSRCH,ON", etc. The iteration convergence values are set to 9.0 N for forces and 34 N-mm for moments (i.e., CNVTOL, …). The simulations show reasonable behaviours and interactions of the RT-PT system. Detailed results are given in the following sub-sections.

Inspection Stages and Solid Contacts

Based on the simulation results, the entire insertion of the RT assembly into the PT can be categorized by five distinctive stages, according to contact situations and variations in stresses:

Stage 1: Locking tool is twisted down, as shown in Figure 5. At this stage, the front end of the locking tool is bent down by its two contacts with the PT. As the arm of moment is only the length of the locking tool, stresses are quickly built up on both the PT and RT1. This stage represents the axial movement of the RT assembly from its initial contact position to about 1.2 m forward, see Figures 10 and 11. Note that the locations where the maximum stresses are obtained are not always at the same spots on the PT or the RTs.

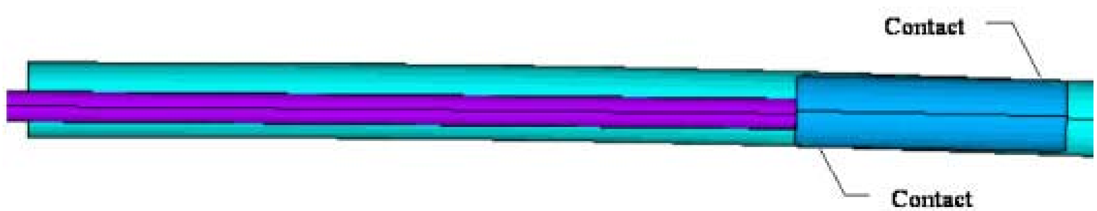

Figure 5 - Inspection First Stage: Locking Tool Twisted Down

Stage 2: The RT2 retainer starts to move into the end-fitting liner tube and finally reaches the PT, Figure 6. At this stage, there are two major contact points: one at the top of the locking tool with the PT, and the other at the bottom of the RT2 retainer with the liner tube. There may be a third contact between the RT1 bottom and the PT bottom, which does not contribute much stress to either the PT or the RTs. The arm of moment is now the total length of RT1, so that the stress on the PT is quite benign as shown in Figure 11. This stage represents the axial movement of the RT assembly from about 1.2 m to 2.3 m.

Figure 6 - Inspection Second Stage: RT2 Retainer Moving in

Stage 3: Locking tool is twisted up, Figure 7. At this time, similar to Stage 1, the front end of the locking tool is bent up by its two contacts with the PT. The contacts on the locking tool have a very short arm of moment, so that they generate very high localized stresses on both the PT and the RTs. This stage represents the axial movement of the RT assembly from about 2.3 m to 3.6 m.

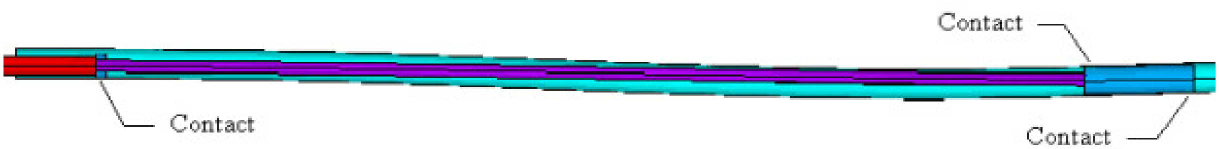

Figure 7 - Inspection Third Stage: Locking Tool Twisted Up

Stage 4: Simply supported RTs, Figure 8. At this stage, the RT assembly appears as a simply supported beam whose one end sits on the locking tool and the other sits on either the RT2 retainer or RT2 bottom surface where the contact occurs. The beam is subjected to a relatively benign load from the contact with the maximum PT sag section. This stage represents the axial movement of the RT assembly from about 3.6 m to 4.8 m.

Figure 8 - Inspection Fourth Stage: Simply Supported RTs

Stage 5: Contact at the top of the RT2 retainer, Figure 9. At this stage, the RT2 retainer contacts the PT, while RT1 loses its top contact with the PT. Since the retainer's OD is much larger than that of RT1, the RT assembly has to be bent much more than in its previous stages. Thus, the stress increases sharply on the RTs. This stage is the worst situation during the entire inspection, and it represents the axial movement of the RT assembly from about 4.8 m to the end of simulations, i.e. 5.3 m.

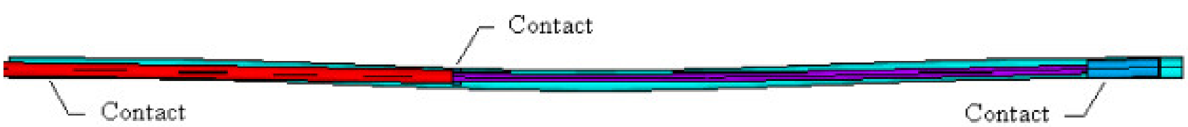

Figure 9 - Inspection Last Stage: RT2 Retainer Top Contact

In the original ADM design, the RT2 retainer's OD was set as 93.98 mm. During a fuel channel inspection campaign at a CANDU nuclear power station in 2000, it was noticed that the retainer had bad scratches and galling on its surface. Therefore, the retainer's OD was reduced to 86.36 mm in the newer design. In this study, both sizes are modelled and simulated. Figures 10 and 11 show the comparisons. It is found that by trimming down the retainer's OD, the maximum stress on the RT assembly or the PT is also reduced. Meanwhile, a smaller OD can delay or eliminate the occurrence of contact Stage 5, a stage that causes high stresses on both the PT and RTs. Therefore, a design improvement is recommended to further reduce the size of retainer's OD, given that there is no other consideration to prevent this reduction.

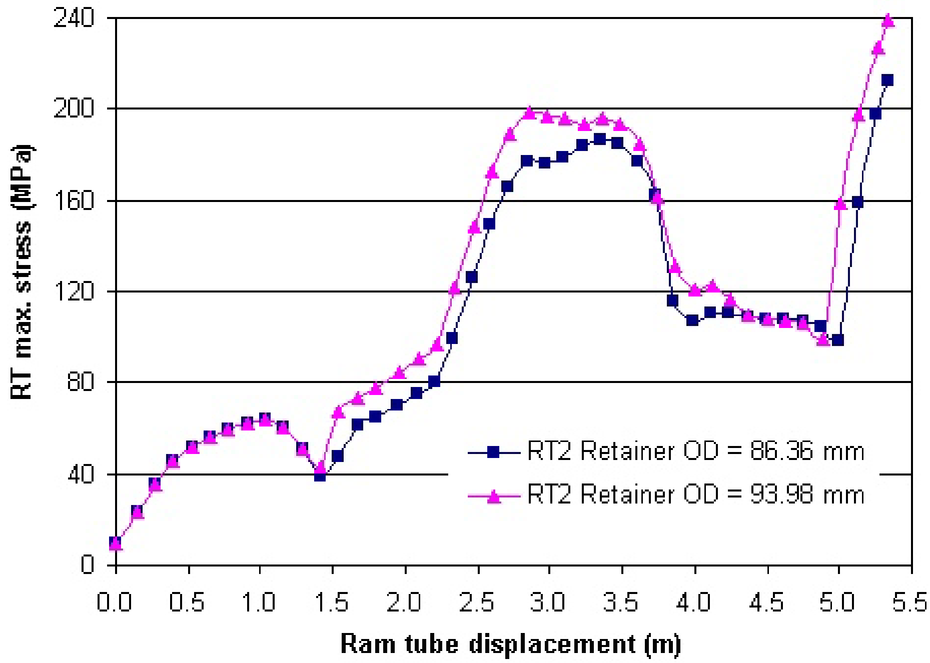

Figure 10 - RT Maximum Stress during Inspection

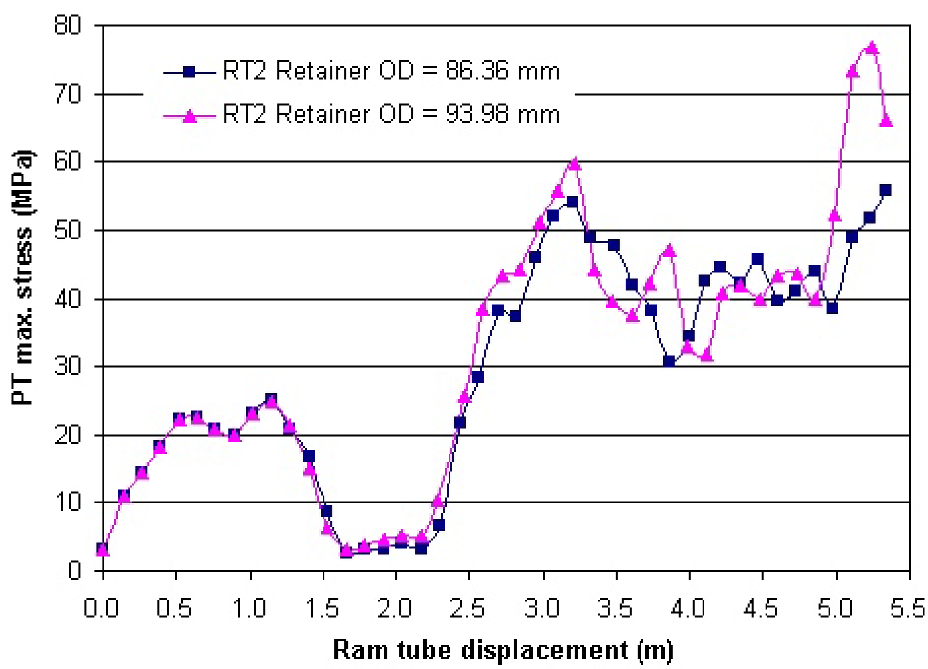

Figure 11 - PT Maximum Stress during Inspection

Recommendations for Tests and Comparisons with Test Data

As the second part of the ADM qualification work, actual tests on a full-scale highly sagged fuel channel were conducted at AECL Chalk River Laboratories in mid June, 2001. The fuel channel in the test rig was manufactured to replicate the sagged PT profile at its end-of-life configuration.

In order to fully benefit from the tests and to reduce any unnecessary workload, optimal set-up of the measurements is essential, such as the number of sensors/gauges and their locations. Note that only the PT is available for measurements, since strain gauges cannot be placed on the RT assembly and there are no other accessible means to monitor the behaviour of the RT assembly. Therefore, based on the FEA results and ANSYS animations of the RT-PT stresses and displacements during an inspection, recommendations were made for the set-up of test rig. There recommendations included what should be measured, how many sensors/gauges were needed and where to place them. Details are not included in this paper. The simulation results also provided a baseline for the tests so that the sensors/gauges could be selected with proper ranges.

Figure 12 gives one of the comparisons between the analytical results from ANSYS and the experimental data from the tests. As seen, the PT vertical displacements are in good agreement at the beginning and the end of the inspection. The overall trends have some similarities. However, a considerable discrepancy still exists, especially in the middle section where the difference reaches its peak value of 2.5 mm. This phenomenon can be understood well by largely the gravity effects that are not included in the ANSYS simulations, as mentioned in Section *FEA Model Development*. At the beginning or the end of the inspection, the relatively heavy inspection head, which is about 1702 mm long, 99 mm in diameter and almost a solid piece, is mainly supported by the PT end fittings. Therefore, the gravity effects do not appear at these stages in the PT deflection. When the inspection head moves into the PT middle section, all its weight sits on the PT, and thus causes the maximum discrepancy between simulation results and test data.

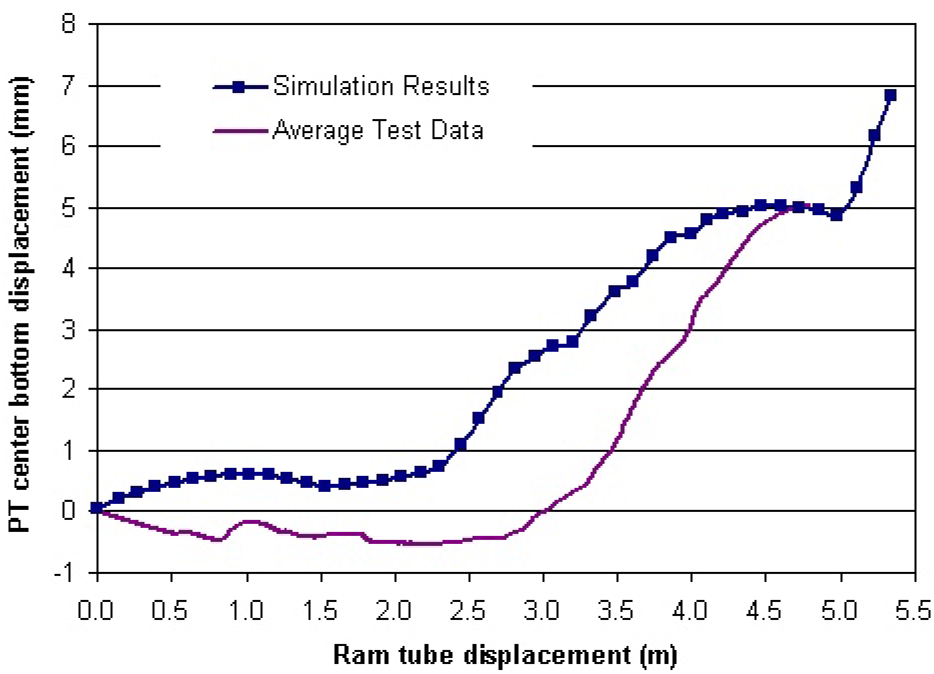

Figure 12 - Comparison with Test Data: PT Vertical Displacement

Summary of FEA Results

Based on the FEA results, it is confirmed that the stresses induced on both the RT assembly and the sagged PT are acceptable during ADM extension/retraction processes:

- The yield strength of the RT material (SA564 Type 630 Condition H1100) is 930 MPa [3], whereas the maximum stress shown in Figure 10 is 240 MPa.
- The yield strength of the PT material (Zr-2.5Nb) is 330 MPa [1], whereas the maximum stress given in Figure 11 is 76 MPa. (Note that this yield strength is for unirradiated tubes; the value can be considerably higher once being irradiated.)

Therefore, for both the RT assembly and the PT, the maximum stresses are about a quarter of their yield strengths, which is well within the safety margin even with the consideration of gravity effects.

Conclusion

In this paper, the FEA study of the ADM delivering fuel channel inspection tools in a highly sagged PT is presented. It shows that ANSYS has great capacity of simulating 3-dimensional contact problems with large rigid-body motions. It also shows the benefits of an FEA study to the qualification or improvement of mechanical designs, as well as the assistances to the set-up of test apparatus on an actual system.

References

[1] *Zirconium Alloy Design Data,* National Standard of Canada, CAN/CSA-N285.6.7-88, March 1988.

[2] Mott, R.L., *Applied Strength of Materials*, Prentice-Hall, 1978.

[3] Atlas Specialty Steels - S17400 Precipitation Hardening Stainless, www.atlassteels.com/17-4.html.