RAI 2-2

The applicant provided the response to RAI 2-3 as following:

"The weld and the accompanying heat-affected zone in the finite element model are modeled as shell elements as well as the rest of the cladding. The end plugs are modeled as solids."

"The strain at the weld corresponds to nodal strain at the top boundary of the end plug that is reported with the strain contour of the cladding of the heat-affected zone as shown in..."

"The maximum inelastic deformation of the weld and the heat-affected zone in the cladding is 2.6% as shown in the figure above."

The staff understands that the weld between the cladding and the end plug creates a moment connection between the cladding and the plug. It is expected that the bending strain at this location will be high. Each node in a shell element has translational stiffness as well as bending stiffness. The nodes of a solid element have translational stiffness but no bending stiffness. Therefore, when a shell element is connected to a solid element, there is no moment transfer because it is a pin connection. So the 2.6% strain reported is a membrane strain because the bending strain does not exist due to the pin connection.

Based on the staff's discussion above, the staff needs the following information to complete the review, and determine the acceptability of the applicant's computational modeling and its analysis results:

(a) Provide analysis that calculates the maximum bending strain in the cladding weld in the heat-affected zone.
(b) Is reduced integration used in the cladding shell elements or are the shell elements fully integrated? How many integration points are used through the shell thickness? Is reduced integration used in the plug solid elements or are the solid elements fully integrated?
(c) Is the 2.6% nodal strain and average of the nodal strains from the 4 elements connected to the node? What is the maximum strain in each of the 4 elements connected to the node?
(d) Provide a plot of the shell and solid finite element mesh in the heat-affected zone.
(e) Provide the LS-DYNA output files (i.e., d3plot files, etc.) for the fuel rod analysis.

This information is needed to determine compliance with 10 CFR 71.41(a), and 71.73(c)(1).

AREVA Response

a) Near the heat affected zone, the effective plastic strain of all shell elements adjacent to the lower plug is small, 0.84%. As the value is less than 1% it is not formally presented in the analytic report but the value can determined by reviewing the output files. The plastic strain of the cladding increases as the vertical distance between the shell elements and the lower plug increases due to bucking effects of the impact. The plastic strain in the cladding reaches the maximum of 2.6% at about 9 inches from the lower plug. For the heat affected zone, the bending stress consists of less than 5% of total combined stress. At the weld, the plastic strain of the cladding is at its minimum along the entire section of cladding. The local bending stress is very small, and the margin of safety is large (compared to 14% maximum plastic strain). Therefore the bending strain or stress of the weld near the heat affected zone is not reported.

RAI 2-2

The minimum ultimate strain of the material Zirconium used for the end plug and the cladding is 14% per reference No. 13 of ATKINS-NS-DAC-ARV-15-02 , Page 23, Figure 16, [Review of Zircaloy-2 and Zircaloy-4 Properties Relevant to N.S. Savannah Reactor Design, Whitmorsh, July 9, 1962, ORNL-3281, UC-80-Reactor Technology, TID-4500, 17th Ed. C.L.]. The result of the current drop analysis shows the maximum plastic strain in the fuel cladding is 2.6% as shown in page B7, Figure B.5.1-2 of main report ”*AREVA TN-B1 ATRIUM-11 Fuel Assembly Shipping Container Drop Analyses*, ATKINS-NS-DAC-ARV-15-02, December 18, 2015” (FS1-0025122-1.0). The maximum strain of 2.6% in the fuel rod is much less than the ultimate strain of 14% for the material Zirconium.

In addition, through welding qualification, including lot testing, the weld is ensured to be structurally stronger than the fuel rod cladding. This testing is burst testing of cladding tubes with welded end caps, for which failure is observed to occur in the base metal rather than the weld or adjacent heat-affected zone. This indicates that the weld and heat-affected zone is stronger than the base metal. Since the analysis indicates that the fuel rod will not fail if the weld is treated as base metal, failure would not occur if the actual weld and heat-affected zone material properties were explicitly represented.

b) The shell element formula in the original analysis was performed using the constant Belytschko-Tsay shell formulation. The reduced integration formulation is not used. The shell element is not fully integrated. Four integration points are used throughout the shell element thickness. Constant stress solid element formulation is used for the solids. Reduced integration is not used in the plug solid elements. The solid elements are not fully integrated. This element formulation is used because the plastic strain is small and the element formulation results in slightly more conservative results.

c) The strain reported is taken from the element not from the nodes. The LSDYNA program does not report nodal strain in the simple element output. However, the nodal displacement and other scalar results are reported as the average from the nodal displacement of 4 adjacent elements. The nodal averaging is an option, which can be turned off. Because the strain is small and the margin of safety is large, the elemental stresses provide an adequate estimate of the strain.

d) A plot of the refined shell and solid finite element mesh is presented in Figure B.5.1-1 of page B7 of Reference ATKINS-NS-DAC-ARV-15-02, AREVA TN-B1 ATRIUM-11 Fuel Assembly Shipping Container Drop Analyses. Because the stress in the HAZ is small, less than 1%, it was not presented in the original report. The existing reports provided plots identifying the maximum stress in the rod.

e) Relevant analysis results were previously provided in reports NSA-DAC-AREVA-14-01 “Structural Analyses of the AREVA Atrium-11 LTA Fuel Assembly in the RAJ-II Container during Normal and Accident Transport Conditions”, and ATKINS-NS-DAC-ARV-15-02 AREVA TN-B1 ATRIUM-11 Fuel Assembly Shipping Container Drop Analyses. The raw data contained in the d3plot and associated ASCII output requires the LSPrePost software, and a detailed knowledge of the finite element model used in analysis to obtain targeted information.