DETAILED 3-D FINITE ELEMENT ANALYSIS OF A BRIDGE SECTION CRUSH ACCIDENT

David C. Harding
Sandia National Laboratories*
P.O. Box 5800
Albuquerque, New Mexico 871085-0405, USA, 505/845-9421

Yasushi Miura, Yuichiro Ouchi, Kiyoaki Yamamoto, and Takafumi Kitamura
Japan Nuclear Cycle Development Institute (JNC)
4-33, Muramatsu, Tokai-Mura
Naka-Gun, Ibaraki, 319-1194 JAPAN

ABSTRACT

This report describes the extension of the validation of previous plane strain finite element analysis (FEA) model results presented in another paper. Japan Nuclear Cycle Development Institute (JNC), in cooperation with Sandia National Laboratories, has performed detailed three-dimensional FEAs of a severe hypothetical bridge section crush transportation accident using the ABAQUS/Explicit FEA code. A 50,000+ element model (shells and solids) with 35 inter-surface contact pairs was used to model the approximately quarter-symmetric package and its fresh fuel assembly contents. This model was validated by comparing analytical deformations and accelerations with empirical data obtained during regulatory 9-m side drop tests onto an unyielding target. Bridge crush analysis results show that the primary containment vessel (including its bolted closure) retained its structural integrity, despite large deformations due to extreme crush and bending loads. Fuel assembly crush forces obtained in this analysis were subsequently applied to another detailed FEA model of the fuel pins and their individual claddings, which showed sufficient secondary containment boundary integrity to avoid cladding leakage. These results show that the safety margin against failure for the package is indeed large, as was intended by IAEA regulations. Although this hypothetical accident condition would be of extremely low probability, it is of scientific interest to investigate the dynamic cask response to such postulated conditions, and such analyses should be used within the framework of a land transport risk assessment.

INTRODUCTION AND MODELING BACKGROUND

Type-B RAM transportation packages are designed to withstand severe accident conditions, based on the stringent IAEA regulations (IAEA 1985). Regulatory impacts onto unyielding targets represent environments as severe as impacts into "real" deformable targets at much higher velocities. Additionally, large factors of safety are "built in" to the regulations due to the fact that they require containment boundaries to remain essentially undeformed throughout the sequential accident environment tests, even though package boundary materials are highly ductile austenitic stainless steels. Transportation risk assessments include the evaluation of even low-probability, potentially high-consequence scenarios. One such accident was postulated from the aftermath of the Hanshin earthquake that hit Kobe, Japan in 1995. That particular magnitude 6.7 (Richter scale) earthquake

---

* Sandia is a multi-program laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy under Contract DE-AC04-94AL85000.

caused large sections of concrete bridges to either buckle or slip off of their support columns and fall onto lower-level roadways. There is a very small chance that a truck transporting RAM packages could be traveling on that lower-level roadway during such an earthquake and could experience a crush environment between a falling concrete bridge section and the asphalt roadway below it. The 1,500 metric ton mass of the bridge section and approximate 10.4 meter height above the roadway were estimated from typical bridge sections in Japan (Nittsu 1997).

Realistically, a bridge section usually falls in an earthquake due to only one end dislodging from its support structure. Using conservation of energy principles, the impact velocity is reduced slightly in this case from about 12.8 m/sec down to about 11.6 m/sec, and the effective mass is only half of the entire bridge section's mass. In order to slightly simplify the analysis boundary condition, the entire bridge can be assumed to fall horizontally onto the package with the slightly reduced velocity, and an effective mass of about one-half that of the original bridge. Accounting for the additional rotational inertia of an assumed rectangular-section bridge, results in adding 22% to this mass. Since the Monju-F package is transported as a group of 3 packages on the back of a flat-bed truck, when modeling only one package in the impact/crush accident, the effective bridge mass must again be reduced by a factor of 3 to account for the kinetic energy absorbed by the other packages.

Given the extremely large kinetic energy of the bridge section (primarily due to the large mass), large deformations were anticipated in the package, truck structure, and asphalt roadway below the package. Based on previous modeling experience, numerous contact surfaces, including self-contact for severe buckling, would be needed to accurately represent the behavior of the package in the crush environment. Due to the transient dynamic nature of the impact and crush events, as well as the need to accurately model numerous contact surfaces and large deformations, the explicit integration ABAQUS/Explicit finite element analysis code (HKS 1998) was used to perform the analyses.

DETAILED 3-D MODEL DEVELOPMENT

Although the initial results from previous plane strain analyses (see companion paper) indicate that primary containment vessel (PCV) and fuel pin cladding integrity is maintained during the bridge crush accident despite large deformations in the PCV wall, Model 1 was relatively simplified and had limitations. Out-of-plane bending and shear forces which might be expected near the relatively stiff fuel holders could not be determined with the plane strain Model 1 of the package. A highly detailed, three-dimensional model was necessary to capture these kinds of package deformations, as well as the performance of the lid and bolted closure to the PCV. It was anticipated that this 3-D model would provide a much more accurate prediction of the primary containment boundary's integrity, and provide better localized fuel assembly compressive loading in order to re-run the detailed plane strain Model 2 analysis and ascertain fuel cladding integrity as well. An IAEA regulatory 9-m side impact onto an unyielding target (used to validate the 3-D model with acceleration and deformation empirical results {Harding 1999}), and the bridge crush accident conditions both allow for taking advantage of one actual and one approximate symmetry plane in modeling the package. Although the Monju-F package is not longitudinally symmetric (it has a lid on only one end), modeling it as such greatly reduces the total number of finite elements, allowing for greater model detail in the primary containment boundary (see Figure 1).

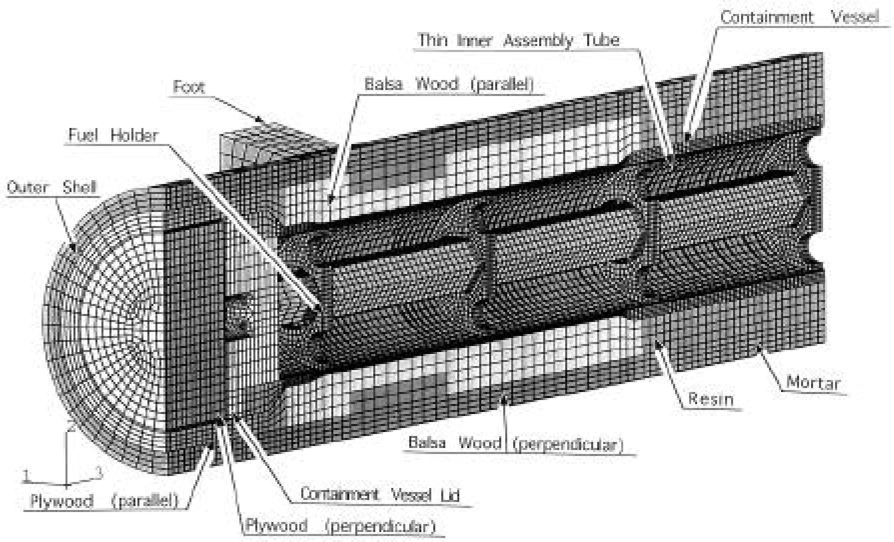

Figure 1. Quarter-symmetric 3-D finite element model.

The 304, 316 and 630 stainless steel materials of the outer shell, primary containment vessel and lid, bolts, inner assembly tube, fuel holders, and fuel assembly were all simulated using elastic-plastic (four noded) shell or (eight noded) solid elements with isotropic strain hardening. The fuel assembly was modeled in a lumped fashion, as an elastic-perfectly plastic material, instead of modeling individual fuel pins, which would have been computationally expensive, and yet produces similar results in terms of package deformations. Crushable overpack materials, such as the grain oriented balsa wood, plywood, mortar, and resin were modeled using 8-noded solid brick elements with crushable foam plasticity constitutive models, each with a pressure-dependent yield surface to produce effective hardening under compression. As with the previous plane strain models, all package material properties match test data at respective elevated temperatures associated with normal transport, due to radioactive decay heat from the fuel assemblies.

The primary containment boundary, including the lid, bolts and PCV, was the critical structure in the 3-D model. A minimum of 4 elements through the thickness was necessary to accurately simulate the bending and shear response of the primary containment vessel, even in areas where its thickness necks down to 8 mm, between the 1st and 3rd fuel holders. A high degree of detail was also included in each of the fuel holders, to accurately model deformations and interactions with both the PCV (via the thin fuel holder tube) and the fuel assembly. Welds between the inner shell and the PCV, as well as smaller machine screws bonding fuel holders within the inner assembly tube, were modeled with simple truss elements using appropriate cross-sectional areas. The bolted closure lid and flange are the thickest and strongest portions of the Monju-F package. The lid was also modeled in sufficient detail to determine its response to the large bridge crush loads. The lid bolts were modeled in sufficient detail to obtain axial stress/strain behavior during the impact. Each bolt was modeled using 4 eight-noded brick elements, consisting of 630 stainless steel. The four nodes on each end were equivalenced with respective nodes on the 304 SS lid and 304 SS PCV flange. Lateral contacts between the bolts and lid or flange were neglected to avoid transverse shear induced hourglassing in the bolt elements (shear instead taken by rigid flange barrel design). Bolt preload was applied via analytical thermal contraction of the solid bolt elements, matching the initial bolt torque-induced axial force.

Shell elements were used extensively in areas where thicknesses were relatively small. Accurate bending response can be maintained when sufficient numbers of elements are used, and ABAQUS calculates this response using 5 integration points through the thickness in its shell element formulations. Shell elements in the Monju-F model include the hexagonal fuel assembly shell, the inner assembly tube, the outer and inner shells (contiguous at the lid end), and the end plug shells (confining the 90° plywood).

Large deformations of various components, including welded or screwed joints were anticipated in the extremely severe bridge crush accident analysis. In anticipation of these large strains and deformations, the fuel holder-to-tube machine screws were allowed to "fail" when and if they reached a prescribed 60% plastic strain, using a failure model in ABAQUS. Similarly, the truss elements simulating the weld bead between the inner shell and the flange end of the PCV were defined with the failure model. In the case of the potential weld failure, additional contacts were defined in order to properly track force transmission between newly adjacent sliding surfaces if the inner shell stretched away from the PCV due to weld failure. Additional contact controls were necessary to keep the ABAQUS code from halting due to local or global tracking errors, and subsequent excessive element deformations if contact surfaces inadvertently crossed each other. Artificial "edges" had to be defined on many contacts so that the boundaries of contact surfaces couldn't slip behind adjacent surfaces and cause large node jumps after the code tried to correct for such cases. Special hourglass controls were sometimes necessary where structures consisting of few elements through a thickness underwent large deformations.

In order to accurately transmit stress waves and forces through the elements of the crushable overpack materials after they reached "lock-up", the elastic moduli of these materials was artificially increased. This avoided wave speed-to-deformation rate errors that could cause premature halting of the code. Because the bridge crush analysis is crush dominated (and deformations are large) as opposed to impact force dominated, increasing the elastic moduli of overpack materials would not affect the PCV response. The perpendicular or 90° balsa wood was crushed beyond its lockup point so quickly that a new analysis step was necessary, in which the material was removed from the analysis after it was fully crushed, and new contacts were defined to continue the analysis. An additional analysis step became necessary to redefine local and global tracking algorithms and accurately track the rapidly changing contacts between the fuel assembly and the 4 fuel holders. Because of the contact difficulties experienced in the model development, many contacts had to be redefined using a "balanced" master/slave algorithm that required additional computation time. Others were better defined using node sets instead of surface definitions. In default time stepping mode, a single analysis would have taken almost 4 weeks of CPU time. The total analysis time was decreased to a more reasonable level by using fixed time stepping through most of the analysis, based on the internal initial stable time step calculations before bridge crush began. Also, the bridge (and rigid mass) velocity was increased by a factor of 2.5, from 11.6 up to 29 m/sec, which allowed the bridge bottom surface to reach the roadway's upper surface more quickly and thus required less CPU time. The factor 2.5 was chosen based on experiments with different values, in an attempt to minimize additional inertial forces on the package derived from the impact, as opposed to the original crush-dominated event. All of these model adjustments were necessary because this bridge crush analysis was difficult, based not only on its material nonlinearities and geometric nonlinearities, but also the rapidly changing inter-surface contact definitions. The forces involved are so large that materials are pushed to their deformation limits and beyond. It is this "beyond" region where localized failure of overpack materials often occurs and modeling such phenomena becomes difficult.

The actual transport condition includes 3 Monju-F packages within a lifting cradle, all on a flat-bed truck. Just as the fuel holders could produce shearing loads to the inside of the PCV, various truck and cradle beams and axles could impart undesirable bending and shear loads to the containment boundary. A full, highly detailed model of the truck and cradle would be computationally prohibitive, especially in conjunction with the already computationally expensive package model. Thus, a simplified model of the critical truck structures was developed, which captured the bending and shear behavior of applicable cross beams, the cradle beam and the rear axle. The tires, wheels, suspension, and wooden truck bed planks were all neglected due to their location away from the path of the modeled central package. The effective stiffness of these components is also insignificant compared with the stiff cross beams, axle, and cradle beam.

In order to reduce the travel distance from the package to the roadway (and thus reduce total computation time since the analysis may run until the package is pressed into the roadway), the pair of main cross beams, axle, 2nd cross beam, cradle beam, and reinforcement beams were modeled using thick shell elements. Contacts between these shell elements and both the asphalt and outer shell of the package do not need to account for the shell thickness, so the package can appear to be resting on the asphalt before the analysis begins. But the bending and shear strength of these cross beams is taken into account by defining area moments of inertia (from the beam cross section) identical to those of the original I-section, C-section, and square-section geometries. The original I-, C-, and square-section beams were each modeled as a solid beam with the original width or footprint (to accurately transmit force through the roadway) and a thickness determined from its equivalent area moment. Boundary conditions applied to the beams were determined from their geometry. Beams such as the axle, cradle, and the two reinforcement beams are long enough to be influenced by the adjacent two packages on the truck, so symmetry boundary conditions (no x-axis displacement of nodes allowed) apply. The main cross beam and the 2nd cross beam, however, end at the bolted joint with the longitudinal truck beam, so that end is "free".

In the overall 3-D bridge crush model, coulomb frictional contact was defined between 127 separate active contact surfaces (defined in 63 contact pairs and 7 contact node sets), including self contacts for the inner and outer shells. The complete model includes over 76,440 nodes, 60,987 elements, and 252,867 degrees of freedom and requires approximately 4 days of CPU time per 24 milliseconds of crush time to run on a Sun Ultra 1 workstation (rated at 300 SPECfp_92, with 256 MB RAM). Only 24 msec was necessary to model the 60 msec actual crush event, due to the artificially increased bridge velocity, by a factor of 2.5.

3-D BRIDGE CRUSH MODEL RESULTS

The overall PCV and all internal structures deformation results are shown in Figure 2, along with the lifting cradle and truck structure beams. Although the deformations in the PCV were large, maximum equivalent plastic strain (PEEQ) levels were well below failure levels for the material. Note that some of the fuel holders appear to be folding over, under extreme crush loading from the bridge section. Plastic strains throughout the PCV are contoured in Figure 3. The apparent misprint that is shown in this figure is the 2010% peak PEEQ levels at the top and bottom surfaces of the PCV, where it is compressing the 2nd fuel holder. Close-up views of these apparent high-strain areas would show that the excessive PEEQ levels are only on the outer surface elements. One defect of using fixed time stepping to insure a more reasonable total CPU usage is that when any element deforms to a level where a stress wave traverses it after the next time step begins, inaccurate deformations arise in the next time step for that particular element. This phenomenon is the cause of the unrealistic plastic strain level jump in these outer elements, and thus these results should be neglected as a defect of the specialized modeling technique. The maximum actual through-thickness PEEQ levels above the 2nd fuel holder are less than 37%, well below the 90% ductile failure level of the material. Very slight deformations to the previously-flat sealing surface on the end of the PCV are almost visible in this figure, however, a more detailed model with a large number of elements along this sealing surface, near the bolts, and in the lid's O-ring groove would be necessary to assess seal integrity. PEEQ levels in the lid near the O-ring seal (between bolts and barrel) were less than 18%, and deformation along this previously flat surface is only slightly visible. Plastic strains in the PCV bolts peaked at only 3.6%, well below their failure level. Thus, the primary containment boundary remained largely intact, although highly deformed. Although this model was not detailed enough to verify leak-tightness, the bolted closure maintained its structural integrity throughout this analysis.

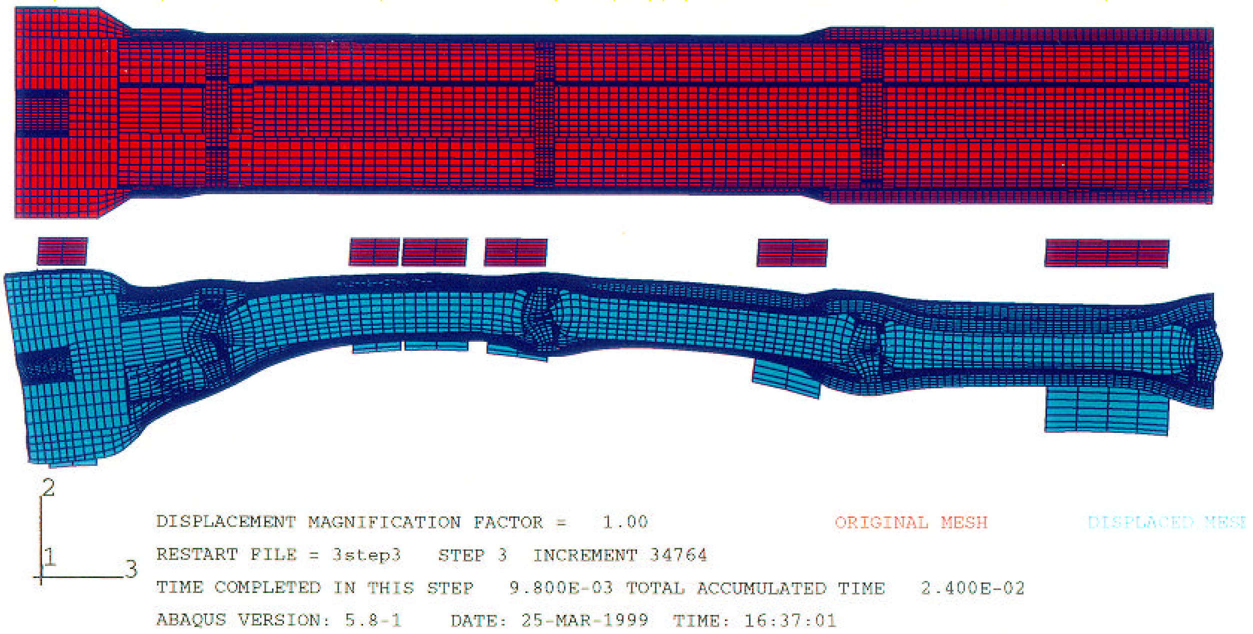

Figure 2. Deformations of the primary containment vessel and its contents

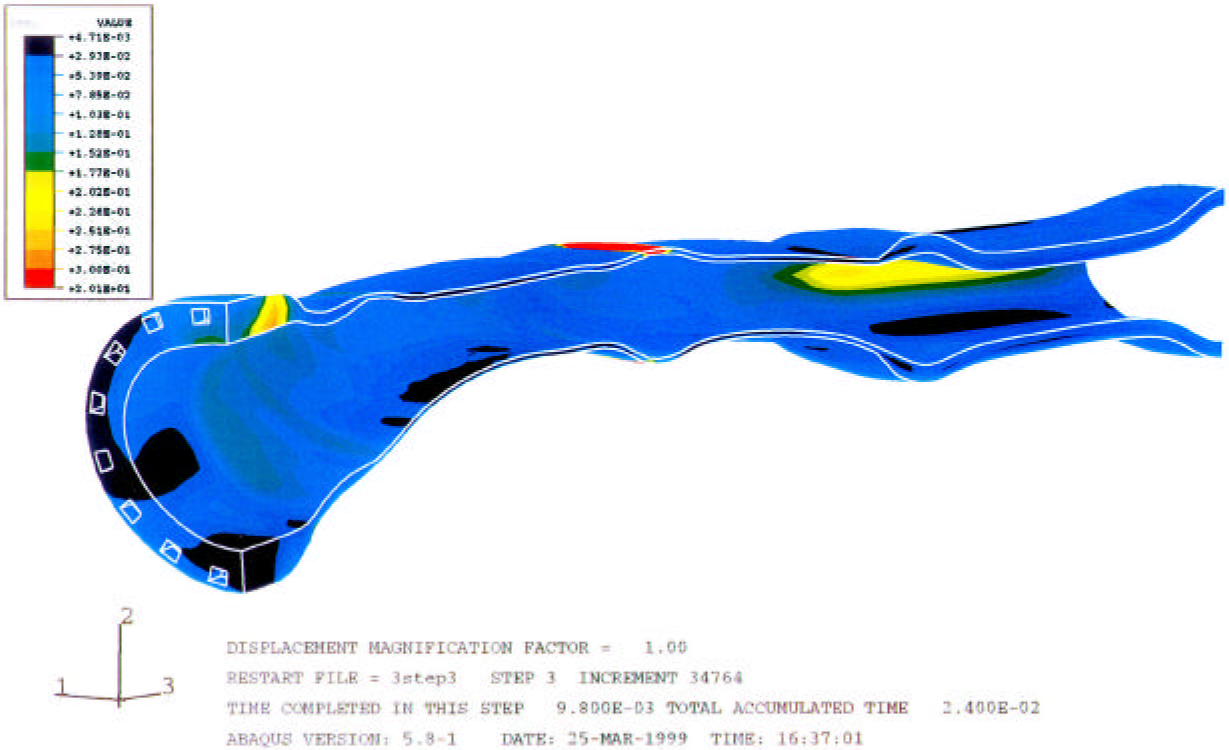

Figure 3. PCV Equivalent Plastic Strain Contours

Compressive loads were transmitted from the primary containment boundary (the PCV) to the fuel assembly via the fuel holders, which also underwent severe deformations. The largest deformation occurred in the second fuel holder, which should be intuitive because the PCV is only 8 mm thick in this area. The large deformations seen in the interior holes of fuel holders meant identical larger deformations occurred in the fuel assembly. Vertical loads on the fuel should be applied to the highly detailed 2-D plane strain fuel pin model to determine individual fuel pin responses. Although surface stress levels reached about 250 MPa at the second fuel holder and 300 MPa at the 4th fuel holder, the through-thickness vertical stress (S22) levels in the fuel reached about 200 MPa at the 4th fuel holder.

The highly detailed 2-D plane strain Model 2 was used to assess the integrity of the fuel pin cladding under the more severe fuel assembly loading results obtained from the 3-D bridge crush model. The analysis was run until the vertical stress in the hex wall section and MOX fuel pellets reached approximately 200 MPa, or about 0.9 milliseconds. Although bending of the hexagonal fuel assembly shell occurred and the fuel cladding was slightly deformed, the cladding peak equivalent plastic strain of about 23% was well below the failure level for the material. Through-thickness values were 12% or less. Thus, integrity of the fuel pin cladding was maintained in this analysis, which simulated the compressive fuel assembly loads resulting from the 3-D bridge crush analysis. As was done previously with the 3-D model, the computer simulation soundness was verified by comparing model energy parameters, and detrimental artificial strain energy was insignificant compared with the total energy.

SUMMARY AND CONCLUSIONS

A series of detailed 3-D and 2-D plane strain FEAs were used to determine containment boundary responses to a severe bridge crush accident condition. Analysis results showed that although deformations in the PCV were relatively large, the primary containment boundary including the bolted closure maintained its integrity throughout the bridge crush accident. Equivalent plastic strain levels are an excellent indicator of ductile material integrity during large deformation, and these plastic strain levels for the lid, PCV, and lid bolts were all well below ductile failure levels. Slight deformations of the O-ring surface, which was not modeled in detail, indicate the possibility that leak tightness may not be maintained, however, more detailed modeling of this area is necessary to accurately assess seal integrity. Remember however, that despite the extra-regulatory nature of this bridge crush accident, regulatory acceptance of a post-accident package is based on a leak rate of less than A_2 per week, not leak tightness, and by this definition the Monju-F package would likely "survive" this extra-regulatory accident.

Compression of the PCV by the bridge, led to large deformations in the fuel holders inside the PCV, and subsequent local deformations in the fuel assembly. Compressive reaction forces in the fuel assembly at these high-stress areas were applied to a highly-detailed 2-D plane strain finite element model of individual fuel pins and their cladding, in order to assess cladding response and integrity. Results from the fuel pin model showed that despite large deformations, equivalent plastic strain levels would not likely exceed failure levels in the secondary containment boundary defined by the cladding, even under loads exceeding those predicted by the 3-D model.

In conclusion, results from these analyses graphically show that the safety margin against failure for the Monju-F package is indeed large, as was intended for all packages during development of the IAEA regulations. The bridge crush accident investigated in this report represents an extremely low-probability severe loading condition, and yet the package's containment boundaries would likely remain essentially intact. In order to fully assess the package's design and safety margin, as well as gain even greater public confidence in nuclear transport safety and the nuclear industry as a whole, additional analyses (and possibly tests) of other postulated severe accident conditions could be performed within the framework of a comprehensive land transport risk assessment.

REFERENCES

Harding, D., Miura, Y., Ouchi, Y., Yamamoto, K., and Kitamura, T., 1999, “Detailed Finite Element Analysis of a Regulatory Impact Accident,” Proceedings of 1999 ASME Pressure Vessels and Piping Conference, Boston, Massachusetts.

Hibbit, Karlsson and Sorensen, Inc. (HKS) 1998, ABAQUS/Explicit Version 5.8-1.

International Atomic Energy Agency (IAEA), 1985, "Regulations for the Safe Transport of Radioactive Material," Safety Series No. 6, Vienna, Austria.

Nittsu Research Center, Inc., March 1997, "Data Research Report for the Evaluation of Safety Margins in a Nuclear Fuel Package Transport Accident."