# PLANE STRAIN MODELING OF A SEVERE 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

Japan Nuclear Cycle Development Institute (JNC) is leading Japan's efforts in fast breeder reactor research and development. As a part of its comprehensive safety assessments, JNC has been ensuring safe fuel transport by performing regulatory tests and analyses on its various Type B packages. In addition, the severe earthquake in Kobe Japan in 1995 motivated JNC to estimate package performance in such severe accident environments as one of the phenomena to consider for emergency preparedness, despite their ultra-low probabilities of occurrence. Although an experimental test would be difficult to implement, numerical simulation is an effective alternative, particularly with respect to cost and time. For that purpose, large deformation and highly non-linear explicit integration analytical methods was investigated. The postulated accident environment was that of a 1500 metric ton concrete bridge section falling 10.4 meters onto a fresh fuel package laden truck traveling on an asphalt roadway below it. As a first stage of this severe accident assessments, JNC and the author from Sandia National Laboratories working as a JNC International Fellow, performed a series of plane strain large-deformation finite element analyses (FEA) of this hypothetical bridge crush (extra-regulatory) accident condition using the ABAQUS/Explicit finite element analysis code. A wide range of material constitutive models was required to accurately model the behavior of various ductile metals, brittle concrete, crushable woods and foams, and pressure-dependent yield materials such as soil and asphalt. Fuel assembly loading results obtained from an overall bridge/package/roadway crush analysis were subsequently applied to a highly detailed FEA of the fuel assembly's individual fuel pins and their cladding. Analysis results showed that although deformations were relatively large compared with regulatory accidents, the two containment boundaries provided by the primary containment vessel and individual fuel pin claddings retained their structural integrity. Because of the simplified nature of the two-dimensional plane strain analyses (appropriate for such a long, slender package), these results should be verified by full three-dimensional analyses, which include out-of-plane bending and shear forces as well. These calculations are presented in another paper in these proceedings (Harding 2001). Preliminary conclusions about the ABAQUS/Explicit code's ability to simulate package performance during even severe accidents can be made. These could be utilized for a comprehensive land transport risk assessment that evaluates additional postulated events and their probabilities of occurrence.

---

[*] 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.

## INTRODUCTION

The ability to analytically determine a radioactive material (RAM) transport package's performance under severe accident conditions is virtually a prerequisite to performing RAM transportation risk assessments. Analytical tools, such as powerful workstations and finite element analysis software, are constantly improving and are making it easier to accurately predict large deformations of structures with multiple contact surfaces. Japan Nuclear Fuel Cycle Development Institute (JNC) is using such tools to quantify the safety of its transportation operations, in which packages have already been experimentally proven to be safe. The same analytical models can then be used to evaluate extremely low-probability extra-regulatory accident conditions as needed under the overall risk assessment framework.

The extensive damage to buildings, roadways, and bridges during the 1995 Hanshin earthquake in Japan presented one very severe potential accident phenomenon which JNC could take into consideration for emergency preparedness. Large concrete bridge sections fell from their supporting pillars and came crashing to the ground below. Due to the design of some roadways in Japan, these bridge sections occasionally fell onto asphalt roadways below, and thus there existed the very slight chance that a truck-transported RAM package could be crushed between a falling bridge section and the truck bed or roadway below it. This is the accident condition evaluated in this paper, using the Monju-F package as a representative package for the analysis.

The Monju-F fresh fuel package was designed by JNC and was certified in 1990 to meet IAEA regulations for Type B nuclear material transport containers. The 5-m long by 0.63-m diameter package has a mass of 2630 kg when fully loaded with two fuel assemblies, as shown in Figure 1. The 304 stainless steel containment vessel is significantly thicker in areas adjacent to the mixed oxide (MOX) fuel and at the bolted flange/lid closure. Cup-shaped heat dissipating copper rings are embedded within the resin-based neutron shield. A relatively rigid mortar material surrounds the neutron shield and functions as a load spreader and thermal shield. Energy absorbing grain-oriented plywood and balsa wood, surrounded by a stainless steel outer shell, protect the containment boundary in impact accident environments.

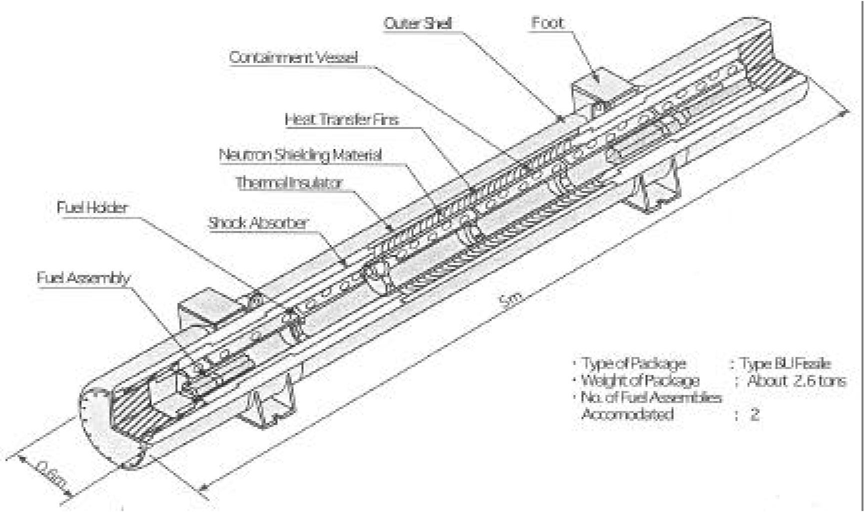

**Figure 1**. Monju-F Fresh Fuel Type B Package

Due to the long and slender overall shape of the Monju-F package, a 2-D plane strain model of the package was initially used to simulate a side-on package crush environment. Subsequently, a highly-detailed 3-D finite element model was developed to provide a more accurate representation of package deformations, the results of which are presented in the following paper. A detailed plane strain model of individual fuel pins within the fuel assembly was used to assess confinement at the fuel cladding boundary. The results of the 2-D and detailed 3-D models under the bridge crush accident condition, are the primary subject of this and the follow-on paper. Similar analytical models may be used in the future to assess package performance under other low-probability extra-regulatory conditions.

## MATERIAL MODELING

There are many different materials associated with this bridge crush problem, each with different behaviors under stress. Developing accurate constitutive models for each of these materials is crucial for obtaining an accurate overall package response to the severe accident conditions. ABAQUS contains a library of material constitutive models that can approximate the behavior of ductile metals, brittle concrete, crushable packaging overpack materials, and granular soils making up the roadway. These existing models were tuned with the appropriate material parameters from test data in order to accurately predict the package's accident response. Additionally, package material properties matched those for the steady-state elevated temperatures observed during transport of fresh fuel.

All metallic structures of the Monju-F package, including the fuel cladding, hexagonal fuel assembly casing, fuel holders, fuel holder tubing, primary containment vessel, lid, bolts, and outer shell, utilized classical metal plasticity models with isotropic yield and either perfect plasticity or isotropic strain hardening behavior. Because the MOX fuel pellets have much stronger compressive strength than tensile, a pressure-dependent yield formulation of a crushable foam model was used only for this (porous sintered) metallic material.

The moderate strain rates associated with the 11.6 m/sec bridge impact do not necessitate the use of strain rate-dependent material properties. Apparent hardening above static stress-strain relations in the crushable overpack materials can be observed merely due to inertial effects. These crushable materials, including the grain oriented balsa wood, plywood, mortar, and resin were modeled using a crushable foam plasticity constitutive model, with the yield surface's deviatoric stress dependent on pressure to effectively produce hardening under compaction while maintaining minimal tensile strength (Krieg 1978). This constitutive model allows the materials to deform volumetrically in compression due to cell wall buckling processes, as occurs in both foam materials and other cellular solids like most kinds of wood.

The properties of the asphalt roadbed and its underlying layers were defined from empirical data (Akamatsu 1997). The Mohr-Coulomb yield surfaces for each layer include granular cohesion and internal friction parameters, and a logarithmic stress-strain relation given by: $\varepsilon = -\ln(1-P/K)$, where $\varepsilon$=true volume strain, P=hydrostatic pressure, and K=bulk modulus. The Modified Drucker-Prager/Cap Plasticity constitutive model in ABAQUS/Explicit is intended to model cohesive geological materials that exhibit pressure-dependent yield, such as soils and rocks (HKS 1998).

The cap portion of the constitutive model serves two purposes: it bounds the yield surface in hydrostatic compression, thus providing an inelastic hardening mechanism to represent plastic compaction; and it helps to control volume dilation when the material yields in shear, due to its lower tensile strength.

Because detailed information on the bridge composition was not known, a simple Mises-based elastic/plastic isotropic material model was used, with a compression strength of typical high-strength concrete. Potential localized brittle fracture of this material can be inferred via resultant plastic strains, although the high degrees of reinforcement typically seen in bridge structures would provide an additional basis for bridge integrity.

## 2-D PLANE STRAIN MODELING

Initially the Monju-F package was modeled with a simplified 2-D plane strain model, to take advantage of the package's long and narrow shape. A plane strain model assumes that all stresses and deformations remain in the plane of interest, which would only be true in the case of a uniformly loaded (in transverse crush) infinitely long, uniform package. Aside from the obvious limitations of such a model, for example capturing localized deformations along the length near the fuel holders, such a simplified model can provide a good estimate of the package deformations, fuel assembly loads, and bridge and roadway behavior in this severe accident environment.

The initial 2-D plane strain models neglected the complicated truck structure, and instead simply assumed that the 3 packages rested on the asphalt roadway and were impacted and crushed by the falling concrete bridge section. The effective mass of the original 1,500,000-kg bridge section was reduced by 1/2 for only one end falling, by 3 since only the central package of 3 truck-loaded packages was modeled, by 1/2 since only 1/2 of the package was modeled (due to symmetry), and increased by 22% to convert the rotational kinetic energy of a tilting bridge back into an assumed horizontally falling bridge (to simplify the model and avoid a sharp corner impact by the bridge). This resulted in an effective bridge mass of 152,500 kg.

## MODEL 1: PACKAGE AND FUEL

A representative 2-D plane strain "slice" was taken of the bridge, asphalt roadway, and Monju-F package through the middle of the package, where the MOX fuel is located, between the fuel holders. Symmetry about a vertical axis between the two fuel assemblies allows for modeling only 1/2 of the package. Instead of modeling a very wide section of roadway and bridge, a 0.5-meter wide section of bridge and roadbed were initially assumed as the width that would be influenced by the package crush. About 2.6 meters of total roadbed depth were assumed necessary to capture roadway response to the load, and a 0.5-meter thick section of reinforced concrete, with a rigid mass simulating the rest of the bridge, was used to simulate the bridge.

The 11.6 m/sec initial velocity of the rigid mass and concrete section were derived from the 8.4-m height of the tilting bridge end above the package. Along the symmetry line, only vertical displacements were allowed at the nodes of elements, and the slope of shell elements at that boundary was preserved throughout the analysis. This is a typical boundary condition for symmetry. Soil nodes at the roadbed's bottom surface were pinned, simulating bedrock or infinite soil below. Another rigid boundary was used to define a frictional slide line between the roadbed modeled and shearing forces from the surrounding roadbed. The vertical shear force at this boundary depended on the product of the normal stress and the friction coefficient (Gonzales 1987), which was derived from the internal friction angle as: $\mu=\tan\beta$. Thus, the shear force along this boundary varied between the different layers partly due to their respective internal friction angles. Without making such a modeling simplification, excessive element deformation can occur near the object deforming the soil.

The concrete bridge was modeled with 100 continuum plane strain 4-noded reduced integration elastic-plastic quadrilateral elements, having properties previously defined. Reduced integration elements are more stable and accurate in ABAQUS. The roadway consisted of three layers: 100-mm thick asphalt using 20 CPE4R elements, 500-mm thick rocky soil using 100 CPE4R elements, and the 2000-mm thick loose sandy soil layer using 300 CPE4R elements, all with pressure-dependent deviatoric yield strength properties.

The package components in this 2-D slice included the 3-mm thick outer shell (50 beam elements), 50-mm thick mortar thermal shield (200 quad elements), 80-mm thick resin neutron shield (350 quad elements), 25-mm thick PCV (200 quad elements), 3-mm thick hexagonal fuel assembly shell (48 truss elements), and lumped hexagonal-shaped fuel assembly (128 CPE4R elements), as shown in Figure 2. The outer shell was modeled with 2-D beam elements that account for bending, and the fuel assembly shell was modeled with truss elements to model its tensile strength in the event of fuel crush. Note that all solid element layers have at least 4 elements through the thickness to accurately respond to bending and shear.

Besides the initial velocity conditions and boundary conditions previously mentioned, inter-surface contacts were specified to properly transmit forces through the various layers and components of the model. Tied contacts were defined between the top of the bridge and its rigid mass element, the package outer shell and the mortar layer, the fuel assembly shell and its lumped fuel, and between each of the three roadway layers. Non-tied frictional contacts (with $\mu=0.1$ assumed, except where noted) were defined between the lower bridge surface and the outer shell of the package ($\mu=0.2$), mortar and resin, resin and PCV, PCV and itself, PCV and fuel assembly shell, and between the outer shell and the top asphalt surface ($\mu=0.3$). These contacts were free to separate, mate, or slide throughout the analysis, as dictated by the mechanics of the analysis. The entire model consisted of 3402 degrees of freedom, with 1500 elements, 1674 nodes, and 11 active contact pairs. Using 7.5 E-7 sec. time steps, the analysis took 40 min. total CPU time for 60 msec of crush time.

## 2-D PLANE STRAIN MODEL 1 RESULTS

The simulation was run until the lower bridge surface reached the original roadway top surface, or about 60 milliseconds. Deformation of the package and roadway absorbed part of the bridge's kinetic energy. Only 17% of the initial bridge kinetic energy was removed up to this point. However, its downward progress would certainly be stopped as it contacted the surrounding roadway surface. Details of the package and roadway deformations are shown in Figure 3. The package was actually pressed downward into the roadway, and the outer mortar layer is significantly crushed. Internal fracturing of this and the brittle resin layers is likely. Significant bending of all package layers, including the PCV, was ultimately restricted by the fuel assembly, which rotated from its initial position due to its unrestrained boundary conditions. Although significant bending of the PCV occurred, the peak equivalent plastic strain of almost 25% is well below the failure level for the material, which is about 90%. Ductile metallic materials fail at plastic strain levels corresponding to their reduction in area (q) at failure in a tensile test by the following relation (Dieter 1976): $PEEQ_{fail} = \ln[1/(1-q)]$.

Reduction in area values for 304 and 316 SS generally range from about 50% to 78% ($69\% < PEEQ < 150\%$), but vary widely based on heat treatment. Thus, integrity of the primary containment boundary was maintained. In order to assess the integrity of the secondary containment boundary, which is the hermetically sealed fuel pin cladding, a more detailed model was necessary, especially given the difficulty in approximating the overall fuel assembly response via a single isotropic material with an outer shell. But the loading from this current model should be applied to the more detailed fuel pin model. The average (compressive) vertical stress in the lumped fuel and adjacent PCV walls was between 10 and 50 MPa.

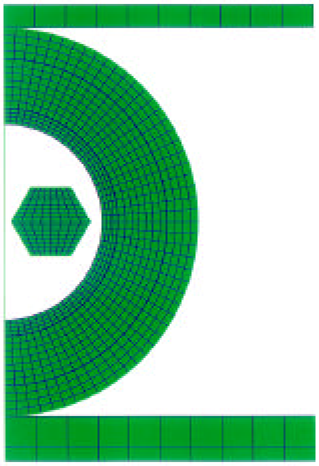

**Figure 2**. Section of Plane Strain Model.

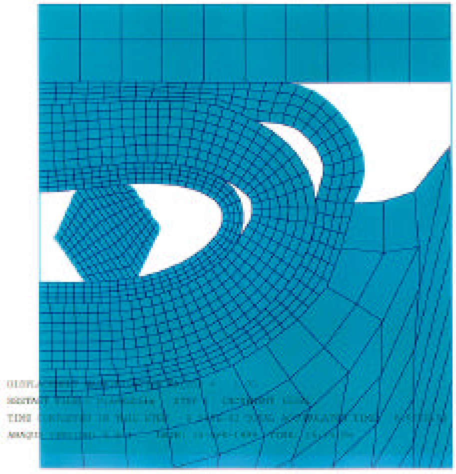

**Figure 3**. Deformed cask and roadway after bridge section crush.

Although some element hourglassing occurred in the lumped fuel, the overall model results appear reasonable. Some hourglass controls within ABAQUS were necessary to maintain element stability. Assessing model energy parameters is one method of insuring model accuracy. Large values of artificial strain energy mean that significant elastic energy is stored in hourglass resistances and that mesh refinement is necessary. Values less than about 4% of total model energy are desirable. The artificial energy was shown to be extremely small, thus increasing confidence in the FEA results.

## MODEL 2: FUEL PINS

Each Monju-F fuel assembly consists of 169 individual fuel rods or pins containing numerous stacked fuel pellets. The central portion of each pin contains the core MOX fuel, and there are blanket materials at each end. The fuel is surrounded by a small 0.08-mm air gap and then sealed by a 0.47-mm thick high-strength 316 SS cladding which serves as a secondary containment boundary. A 5.4-mm diameter wire spacer spirals along the length of each fuel pin to provide an air gap between the 169 pins in the assembly.

In order to assess the integrity of the fuel cladding containment boundary, a highly detailed 2-D plane strain model was developed. The "slice" for the model was again taken from the central portion of the fuel assembly and a small section of the impinging PCV wall, which was shown in the previous model to compress against the hexagonal fuel assembly shell (hex). The overall model is shown in Figure 4, minus the PCV wall above the hex. In order to reduce overall model size, two symmetry planes were assumed (with appropriate boundary conditions), so that only a corner of the actual hexagonal fuel assembly was modeled.

This detailed 2-D model was used to simulate the crush of the fuel assembly between opposite sides of the PCV walls. The same rigid bridge mass element as used in the previous model was tied to the 25-mm thick PCV wall section, modeled with 200 quad elements. The bridge mass and PCV were given the same 11.6 m/sec downward initial velocity. Other fuel assembly components in this 2-D slice included the 3-mm thick hex shell (176 quad elements), 0.47-mm thick fuel pin cladding (2320 quad elements, 320 each pin), 5.4-mm diameter MOX fuel pins (1015 quad elements, 140 each pin), 1.37-mm diameter wire spacers (360 quad elements, 60 each pin), and 6 truss elements to secure each of the 6 wire spacers to its respective fuel pin.

The two symmetry conditions translated to zero horizontal displacements for nodes along the vertical axis and zero vertical displacements along the horizontal axis. Tied contact was defined between the top of the PCV and its rigid mass element. Non-tied frictional contacts (with $\mu$=0.1 assumed everywhere) were defined between: the lower PCV surface and the outer shell of the hex, hex and numerous claddings, hex and numerous wire spacers, hex and itself, many claddings and themselves (self contact), and between fuel pins and respective claddings. These contacts were free to separate, mate, or slide throughout the analysis, as dictated by the mechanics of the analysis. The entire model consisted of 10,059 degrees of freedom, with 4081 elements, 5028 nodes, and 87 active contact pairs. Using 1.5 E-8 sec. time steps, the analysis took 3 hours and 6 min. total CPU time for 1.2 msec of crush time. Self contact is computationally expensive because of the necessity to increase the frequency of contacts checks between facets of element surfaces.

## 2-D PLANE STRAIN MODEL 2 RESULTS

The analysis was initially run until the vertical stress in the PCV wall section reached the approximate level shown in the Model 1 results, or about 0.55 milliseconds. The overall deformations to the hex, cladding, fuel pin, and wire spacer structures are shown in Figure 5. Although the deformations appear slightly larger than those for the lumped fuel assembly in the Model 1 results, the actual fuel assembly contains numerous air gaps that must be closed before significant stresses build up. Compressive vertical stress levels in the PCV range from about -10 to -50 MPa, similar to the Model 1 results. Although some bending of the hexagonal fuel assembly shell occurred and the fuel cladding was slightly deformed, the cladding peak equivalent plastic strain of almost 11% is well below the failure level for the material. Thus, integrity of the fuel pin cladding was maintained in this analysis. Model soundness was verified by plotting model energy parameters, and detrimental artificial strain energy was shown to be insignificant.

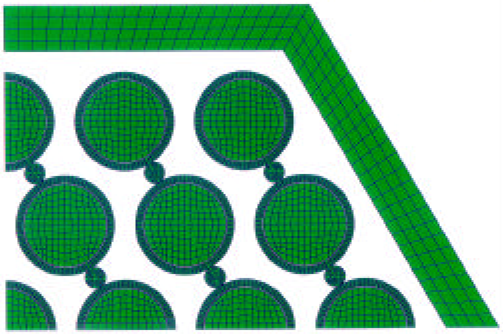

**Figure 4**. Detailed 2-D fuel pin model.

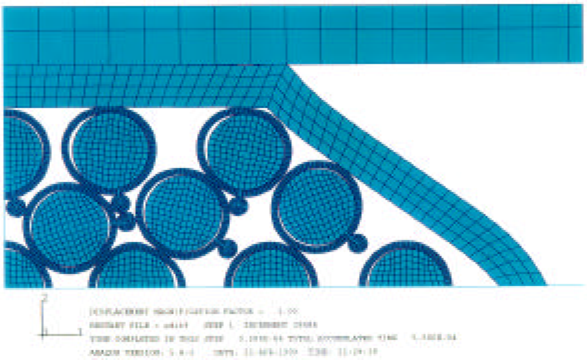

**Figure 5**. Deformed fuel assembly.

## SUMMARY

A series of 2-D plane strain FEAs have been performed to initially assess the response of a Monju-F package to an extremely severe, low-probability earthquake accident environment defined by a truck-loaded package being crushed between a falling 1,500-ton concrete bridge section and an asphalt roadway below. Analysis results showed that although deformations in the primary containment vessel and hexagonal fuel assemblies were relatively large, the PCV and fuel cladding maintained their 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 were all well below ductile failure levels. A full 3-D model is necessary, however, to capture out-of-plane bending and shear loadings on the fuel assemblies, as well as to better understand the performance of the bolted flange closure of the primary containment vessel, which is the subject of the follow-on paper. These FEA simulations of package response to extreme accident conditions could be used as part of a comprehensive land transport risk assessment that evaluates additional postulated events and their respective probabilities of occurrence.

## REFERENCES

Akamatsu, H., Taniuchi, H., Fujimoto, T., and Ouchi, M., 1997, "The Analysis of Drop Accident on to Real Target," 4th International Conference on Transportation of the Nuclear Industry, UK.

Dieter, G. E., 1976, Mechanical Metallurgy, McGraw-Hill, New York.

Gonzales, A., 1987, "Target Effects on Package Response: An Experimental and Analytical Evaluation," SAND86-2275, Sandia National Laboratories.

Harding, D. C., Miura, Y., Ouchi, Y., Yamamoto, K., and Kitamura, T., 2001, "Detailed 3-D Finite Element Analysis of a Bridge Section Crush Accident," 13th International Symposium of PATRAM, Chicago, IL.

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

Krieg, R. D., 1978, "A Simple Constitutive Description for Soils and Crushable Foams," SC-DR-72-0883, Sandia National Laboratories, Albuquerque, NM.