CONF-890855--35

IMPACT ANALYSIS OF SHIPPING CASKS

Phillip A. Pfeiffer and James M. Kennedy
Reactor Analysis and Safety Division
Argonne National Laboratory
Argonne, Illinois 60439

CONF-890855--35

DE89 012423

Received by OSTI

JUN 02 1989

INTRODUCTION

Shipping casks are being used in the United States Department of Energy to transport irradiated experiments, reactor fuel, radioactive waste, etc. As a shipping container, the cask must meet the requirements of the code of federal regulations, 10 CFR-71, for shipments of fissile material and large quantities of radioactive material. One of the critical requirements in shipping cask analysis is the necessity to withstand severe impact environments. It is still conventional to develop the design and to verify the design requirements by hand calculations (Fromm and Balmert, 1978). Full three dimensional computations of impact scenarios have been performed (Miller, 1978), but they are too expensive and time consuming for design purposes. Typically, on the order of more than an hour of CRAY time is required for a detailed, three dimensional analysis. The paper describes how simpler two- and three-dimensional models can be used to provide an intermediate level of detail between full three dimensional finite element calculations and hand calculations.

The RAS/TREAT shielded shipping cask, which is shown in Figure 1, is composed of an outer shell and an inner cask liner separated by sleeves of depleted uranium (DU). The outer shell is welded to a head forging to which two lifting trunnions are attached. A stepped shielding plug (closure head) is bolted to the head forging. The closure head has 16 energy absorbing fins to protect the top end of the cask during a hypothetical accident. A bottom end cap is welded to both the inner cask liner and outer shell. Both the plug and end cap have uranium inserts. The cask has three alignment pads located 120 degrees apart to facilitate handling procedures. There are also four support feet which provide the cask with a fixed orientation.

The regulation that is examined here is: 10 CFR-71.73 hypothetical accident conditions, free drop. Free drop for an accident condition of a Class I package (approximate weight of 22,000 lb) is defined as a 30 foot drop onto a flat, essentially unyielding, horizontal surface, striking the surface in a position for which maximum damage is expected.

Three free drop scenarios are analyzed to assess the integrity of the cask when subjected to large bending and axial stresses. These three drop scenarios are: (1) a thirty foot axial drop on either end, (2) a thirty foot oblique angle drop with the cask having several different orientations from the vertical with impact on the top end cask corner, and (3) a thirty foot side drop with simultaneous impact on one of the lifting trunnions and the bottom end. Prevention of damage hinges on the strength of the various components that comprise the cask. The predicted levels of deformation and stresses in the cask will be used to assess the potential damage level.

The finite element computer codes, WHAMS-2D (Belytschko and Mullen, 1978) and WHAMS-3D (Belytschko and Tsay, 1982) were used to model the cask and containment vessel response for integrity assessment.

TWO-DIMENSIONAL FINITE ELEMENT MODELS

The mesh constructed for the RAS/TREAT shipping cask (weight of 21,550 lb) for the bottom end drop case is shown in Figure 2. The top end drop case mesh is very similar. Shell elements are only shown as a line. The shaded elements in Figure 2 are special impact elements, which allow only compressive forces in the vertical axis. Cask lift-off and sliding of the cask normal to the vertical axis is possible with the impact element. Slidelines are used between the cask walls (inner and outer) and the DU to allow for slumping of the DU. This feature was needed in order to address possible buckling of the cask, which might occur due to the high axial loads and DU slumping from the end impacts. The steel material properties in the end cap, head forging and cask walls were assumed to be the same, and it is assumed that it is subjected to high strain-rates. The static yield and ultimate stresses are 30 ksi and 55 ksi, respectively, while the corresponding dynamic values are 51.5 ksi and 82.7 ksi. The DU was modeled as a hydrodynamic fluid in order to capture the slump effects. The initial velocity direction for the bottom end drop mesh is given in Figure 2. All the translational (except rotational nodes of shell elements and impacting nodes) have an initial velocity of V_o, which for a 30 foot accident drop is 43.95 ft/s.

THREE-DIMENSIONAL FINITE ELEMENT MODELS

Schematics of the two finite element models of the cask are shown in Figure 3 with node and element numbering. The principal finite element employed is an elastic-plastic beam element with a hollow cylindrical cross-section. Twelve integration points are employed in the circumferential direction. Two different models were used; they differ in the modeling of the DU. The first model only represents the mass of the DU, not the stiffness. The mass is distributed equally to the outer and inner shell elements. The second model has elements for the DU with full mass and stiffness representation. The two models have been used because the DU is in 20 inch sleeves, which are lapped together and closely fitted between the outer shell and inner cask liner. It is impossible to represent these details so the two models provide bounds on the response based on zero stiffness and full stiffness for the DU. Note that although the components are shown separate in the figure, the midlines of the components physically coincide due to the common node numbering. A total of 73 elements and 40 nodes were employed in the model with only mass representation for the DU; 107 elements and 40 nodes were used in the model for the mass and stiffness representation of the DU. Element cross-section properties are listed in Table 1.

Table 1. Element Cross-Section Properties

Elements	Outside Diameter (in)	Thickness (in)	Area (in^2)	Section Modulus (in^4)	Material
1 to 33	22.0	1.5	96.60	5100.0	A333-1
34 to 35	24.0	2.5	168.86	9755.0	A350 LF-1
36	24.0	5.625	324.71	13700.0	A350 LF-1
37 to 39	24.0	3.625	232.04	12040.0	A350 LF-1
40 to 73	12.75	0.68	25.76	470.0	A106-B
74 to 107	19.0	3.0	150.80	4995.0	DU

The lifting trunnion is modeled by simple rod elements with crushing steel properties and a tension cutoff to allow free upward bounce. The lifting trunnion has a 6 inch diameter cross-section with a depth of 7 inches. The impact of the cask on the bottom end and top corner was modeled by an element which represents an ungula deformation of a cylinder. This accounted for an increasing cross-sectional area due to the cylinder deformation (crushing). The element deformation was equated to the height of the ungula.

FINITE ELEMENT SIMULATIONS

One finite element analysis (axisymmetric with WHAMS-2D) addresses the accident where the cask is dropped from a height of thirty feet onto either of its ends. For each load case, two different simulations were made; the first was without imperfections in the cask walls, the second with imperfections. The reason for the imperfections is to trigger a buckling mode, if it is possible.

A second finite element analysis (with WHAMS-3D) pertains to an accident where the cask is free dropped from a height of thirty feet with impact on the top end cask corner. To provide a simple yet conservative model, the energy absorbing fins are neglected in both models (modeling variation in DU stiffness). Strain-rate effects on the stress-strain properties were not included in the simulation. The impact occurs at node 40 and is described by an element representing the ungula deformation of a cylinder. This allows the impact to initially act on a small area and then spread out as deformation (crushing) of the element occurs. Element 39 is considered rigid in this simulation. Four different drop orientations are considered; initial cask orientations were 0.5°, 10°, 20° and 30° from the vertical.

A third finite element analysis (with WHAMS-3D) addresses the free side drop of an accident where the cask is dropped from a height of thirty feet with simultaneous impact onto one of the lifting trunnions and the bottom end. The lifting trunnion is represented by simple rod elements acting on nodes 35, 36 and 37. An element (at node 1) which represents an ungula deformation of a cylinder is used to describe the bottom end impact.

SIMPLIFIED ANALYSIS AND HAND CALCULATIONS FOR CASK IMPACT

An alternative to using finite element codes (2D axisymmetric continuum and/or shell, 3D continuum and/or shell, etc.) for impact problems, are hand calculations based on conversion of potential energy into strain energy. By approximating the instantaneous impact force on the cask, the resulting axial and shear forces and moments in the cask can be calculated.

For axial loads only, the plasticity effects on strain energy is easily handled because the stress is constant across the cross-section. A closed form solution is available and can be obtained by hand calculations. When plasticity occurs due to moment or a combination of axial load and moment, the calculation of strain energy becomes complicated because the axial stress is no longer constant across the cross-section. One method to solve this type of problem is a layered-iterative technique. An iterative scheme is also employed to solve the energy balance. The details of this simplified analysis approach are described in a companion paper (Pfeiffer and Kennedy, 1989).

Solutions are obtained for the thirty foot end drop, oblique angle drop and the side drop. The resulting stresses and decelerations for drop angles of 0° (end impact), 0.5° and 5° thru 85° by 5° increments are sought. The influence of with and without the depleted uranium (DU) stiffness (area and moment of inertia) and strain rate effects (static vs. dynamic) on the stress-strain response are included. The mass or weight of the DU is always taken into account. Both the outer and inner steel shells are assumed to have the same stress-strain response as an approximation. Another approximation and simplification was the assumption of the DU stiffness having the same stress-strain response as the steel.

At an angle of 90° the cask will most likely impact as a simply supported beam. A similar type of simplified analysis can be performed by assuming that the cask will impact simultaneously on one of the lifting trunnions and the bottom end. By assuming a simply supported beam with the mass (or weight) distributed along the length and with equal reaction force on each end, a solution can be obtained.

COMPARISON OF ANALYSES AND CONCLUSIONS

The end drop analyses considered both material failure and buckling of the cask for the finite element (2-D) simulations and the hand calculations. These simulations were made without and with imperfections to ascertain whether the cask is safe from failure due to buckling or exceeding allowable stress values. All the simulations and hand calculations indicate the cask will be able to survive the thirty foot drop.

The finite element results indicate that the imperfections do not influence the response of this cask very much, and the bottom and top end drops result in almost the same stress levels in the cask shells. Hand calculations were made for the bottom end drop for both static material properties (with and without DU stiffness) and for dynamic properties (with and without DU stiffness). The static yield and ultimate stresses are 30 ksi and 55 ksi, respectively; the dynamic values are 51.5 ksi and 82.7, respectively. Comparing the hand calculations with a maximum stress of 58 ksi and 62 ksi and the finite element simulations where the stresses are from 55 ksi to 60 ksi (which are based on dynamic material properties for the end drop) shows a difference of around 4%. This indicates a reasonable estimate of maximum stress in the cask end drop can be made with just simple hand calculations.

Buckling was also addressed in both hand calculations and finite element simulations. The hand calculations investigated the crimping behavior of the cask shells under axial loads. In addition, the finite element analysis also included the DU slump, which would augment the buckling of the cask shells. Buckling was not observed in either analysis. The hand calculations indicated that inelastic buckling would occur at axial stress levels of 122 ksi, which is about twice the amount of stresses developed under end impact loads for both hand calculations and finite element analyses.

The oblique angle and side drops were investigated by both finite element (3D) and simplified analyses. A simplified analysis was developed to analyze instantaneous impact of cylindrical shells under elastic-plastic loading conditions. The finite element simulations performed for the side and oblique angle drops indicate that the static ultimate stress of 55 ksi would not be exceeded in bending. Results were presented for two models: one which included only the mass of the lapped sleeves of the DU and one which included the mass and stiffness of the DU. These two models were studied because it is difficult to ascertain whether the bending stiffness of the DU would actually come into play. The oblique angle drop simulations with a near vertical, 0.5°, initial drop angle orientation yielded the maximum bending stress of 51.1 ksi (Figure 4). Simulations were performed for a sufficient range of drop angles (0.5°, 10°, 20°, 30° and side drop) to guarantee that the maximum damage is sufficiently captured. The results of the simplified analysis in Figure 5 with dynamic properties and without DU stiffness indicated that the critical angle was 0° with a maximum stress of 67 ksi. The results of the simplified analysis indicate that reasonable solutions can be obtained when compared to the finite element results. Both methods indicated that the maximum stresses are obtained when the cask is near vertical. As the drop angle is increased (towards the side drop of simultaneous impact on the trunnion and bottom end), the maximum stresses are reduced to just above yielding. In the side drop analysis the finite element solution gave a maximum stress for the mid-span of 30.6 ksi with DU stiffness and 32.8 ksi without DU stiffness. The simplified analysis gave a maximum stress for the mid-span of 36.2 ksi with DU stiffness and 39.9 ksi without DU stiffness. The simplified analyses resulted in stress levels about 20% higher than the finite element analyses. The conservative result of the simplified analysis is mainly due to the assumed prismatic beam solution in which the mass and stiffness is evenly distributed along the span.

The main objective of this study was to find the orientation of the cask which gave the maximum damage. A large number of finite element simulations are needed because of the multitude of drop angles that are to be analyzed. Therefore a simplified analysis for impact of a shipping cask was developed in order to efficiently locate the approximate orientation and level of maximum damage. The results of this study indicates that the simplified analysis can be used to reduce the number of cases that need to be analyzed by the finite element simulations. However, the finite element simulations are still needed in order to fully capture the important details of the impact of the cask. Details, such as change in cross-section, different materials, DU slump, strain-rate, etc., cannot be easily implemented in hand calculations or simplified analyses.

The two-dimensional finite element calculations described took approximately 2.5 minutes of computer time per simulation. The three-dimensional finite element calculations described took approximately 30-60 minutes of computer time per simulation on an IBM 3033 computer. When these simulations are run on a CRAY, the times are reduced by approximately 4-5 times. Compared to the full three dimensional computations (continuum) the computer time is about an order of magnitude less per simulation. It is our opinion that this intermediate level of finite element analysis is sufficient for designing a cask for safe operation. This type of analysis is less costly in dollars to run and also less costly in time to develop the models as compared to full 3-D continuum models. The models described here capture the main features such as bending and axial stresses that are most important in design.

ACKNOWLEDGEMENT

This work was performed under the auspices of the U. S. Department of Energy, Office of Technology Support Programs, under contract W-31-109-Eng-38.

REFEREMCES

Belytschko, T. and Mullen, R. (1978). WHAMS: A Program for Transient Analysis of Structures and Continua. Structural Mechanics Software Series, II, pp. 151-212.

Belytschko, T. and Tsay, C. S. (1982). WHAMSE: A Program for Three-Dimensional Nonlinear Structural Dynamics. Report EPRI NP-2550, Electric Power Research Institute.

Fromm, E. O. and Balmert, M. E. (1978). Safety Analysis Report on the Shipment of Irradiated Fissile Materials in the ANL-402-SPM and ANL-403-SPM Shipping Casks. Battelle Columbus Laboratories.

Miller, J. B. (1985). Three-Dimensional Finite Element Impact Analysis of a Nuclear Waste Truck Cask, SAND84-1899, Sandia National Laboratories.

Pfeiffer, P. A. and Kennedy, J. M. (1989). Free Drop Impact Analysis of Shipping Cask. Nuclear Engineering and Design, to be published.

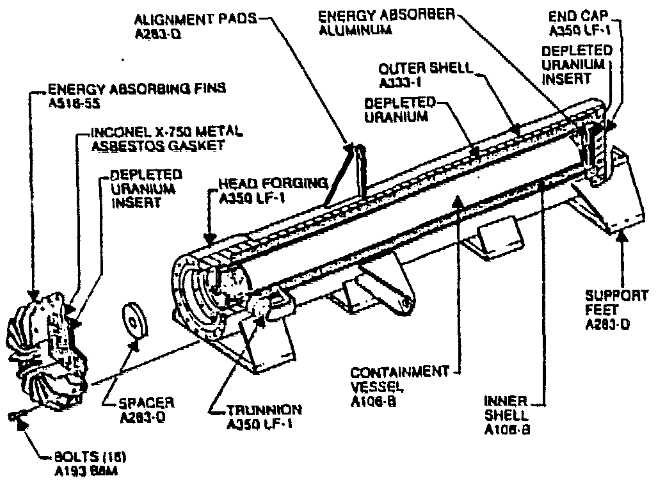

Figure 1. TREAT Shipping Cask

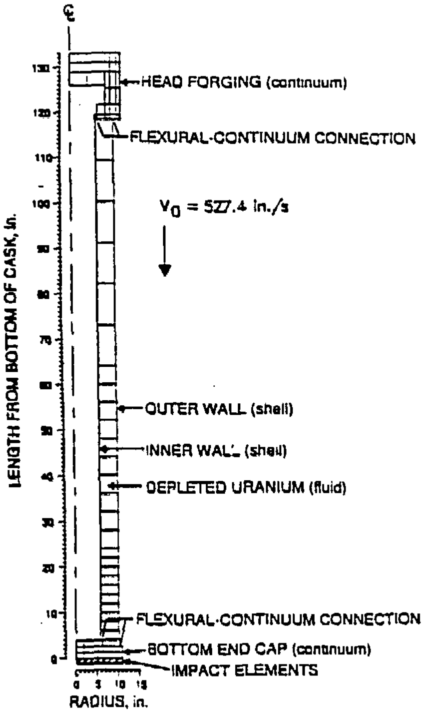

Figure 2. 2-D Mesh for End Impact

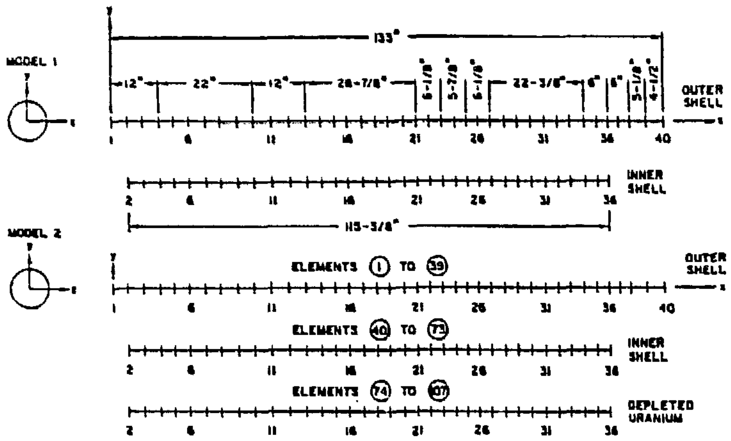

Figure 3. Finite Element Models of TREAT Cask

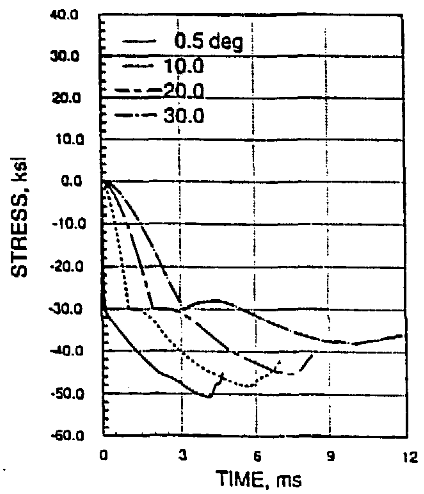

Figure 4. Stress-Time Histories at Node 38 (w/DU m & K)

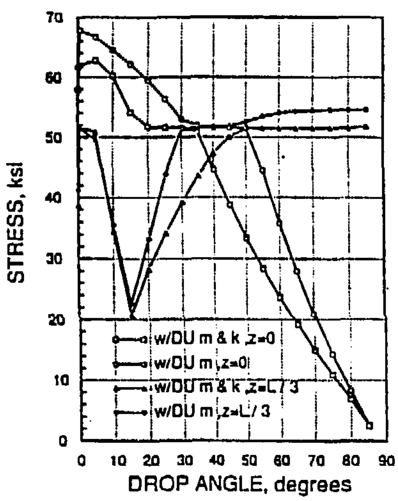

Figure 5. Axial Stresses with Dynamic Properties