SMiRT-12 / K. Kussmaul (Editor)

J04/6

ASSESSMENT OF CASK LOADING IN DROP TESTS WITH FINITE ELEMENT METHODS

J. Rückert[1], R. Diersch[2] and P. Marcault[1]

[1]Engineering System International GmbH, Frankfurter Strasse 13-15, D-6236 Eschborn, Germany
[2]Gesellschaft für Nucklear-Service mbH, Zweigertstrasse 28-30, D-4300 Essen 1, Germany

In the design of nuclear spent fuel casks it is desirable to assess the structural loading of safety critical components in prescribed drop tests, without performing immensely expensive experiments. With modern explicit Finite Element Codes, specialized for crashworthiness analysis, it is possible to efficiently simulate such extremely nonlinear and dynamic impact events, even on high performance workstations. Here, a Finite Element model for the simulation of a standardized 9m - fall onto the edge of the shock absorbers that protect the container is presented. In addition the evaluation of a material model for wood, which is used in the shock absorbers is described together with the calibration of this model against impact tests with wooden specimens. It is shown that a relatively simple approach, assuming isotropy and neglecting shear failure, can yield satisfactory results in assessing the relevant material behaviour.

1. INTRODUCTION

During the past few years Finite Element methods and in particular those based on the explicit solution strategy are gaining popularity for crashworthiness simulation in the automotive and aerospace industries. In this paper the application of such a program for the simulation of the impact of a nuclear spent fuel cask is presented. The goal of the simulation is to predict the loading of the container i.e. to get stress and/or strain information at any location in the structure.

Besides the geometry and material of the cask, the mechanical behaviour of the shock absorbers is essential in obtaining this information. Shock absorbers are usually comprised of a steel-structure that is filled with wood. This package heavily deforms under an impact and thereby limits the force transmitted to the container. As the wood is most important in absorbing the kinetic energy, its material behaviour needs to be treated sufficiently. Here it is shown how a material model for wood was developed and calibrated, that is considered to be satisfactory for the shown drop test. The calibration is based on experimental results from drop tests with wooden specimens.

Prior to the presentation of the wood model a brief introduction to the explicit PAM-CRASH™ code which is used for the simulations is given. Finally the Finite Element model for the container is described in detail, together with the relevant simulation results.

2. THE PAM-CRASH™ CODE

PAM-CRASH™ is a 3D-Lagrangian explicit Finite Element code for analyzing the nonlinear dynamic response of structures. It is specifically designed for crashworthiness analyses in transportation, aerospace and nuclear industries [1]. Spatial discretization may be achieved by using shell, beam or solid elements. The discretized equations of motion resulting from the virtual work principle are expressed as

$$[M]\{\ddot{u}\} = \{F_{ext}\} - \{F_{int}\}$$

where: $\{\ddot{u}\}$ is the acceleration vector of all nodes,
$[M]$ is the diagonal mass matrix,
$\{F_{ext}\}$ is the vector of externally applied loads and body forces consistent with the spatial discretization, evaluated in the current configuration and
$\{F_{int}\}$ is the vector of internal forces.

The equations of motion are integrated by the method of central differences. In this method time discretization is accomplished by the central difference operator, which expresses velocities at time $t + \Delta t/2$ and displacements at time $t + \Delta t$ explicitly in terms of present accelerations, velocities and displacements.

A variety of material laws are available to model nonlinear, anisotropic, strain rate dependant and failure material behaviour. The contact/impact algorithm permits gaps and sliding contact along element interfaces.

The program has successfully passed through a series of standard benchmark tests which are generally used in the nuclear industry for qualification of Finite Element codes.

3. EVALUATION OF A MATERIAL MODEL FOR WOOD

For the fir-wood which is used in the absorbers, force-deflection curves derived from experiments were available. In these experiments specimens of 50 mm height and 100 mm diameter are hit by a stiff impactor (m= 103 kg) with an initial velocity of 13,29 m/s. It can be observed that the results depend heavily on the grain direction of the wood relative to the crushing direction. For crushing perpendicular to the grains (curve A), the wood passes through an elastic phase which is followed by a phase at a constant force level until a point is reached after which the force increases rapidly. Upon unloading it can be seen, that most of the deformation is permanent. This behaviour can be explained by the internal structure of wood, which is essentially a "composite" material with stiff fibers surrounded by a rather soft matrix. As the wood crushes, the fibers, depending on their loading direction, buckle, break or simply are compressed, whereby the matrix material is reduced in volume drastically.

If crushing takes place parallel to the grain direction (curve B), the behaviour is quite different. The initial stiffness is much higher, and the load reaches a maximum in the early compression phase. Once the fibers buckle, the impactor force reduces to a lower level ending up in a higher total deformation.

Because layers with grain directions perpendicular to each other are used in the shock absorbers of the container, another specimen representing this mixture was tested. The resulting force deflection curves are shown for all three cases in figure 1. The behaviour of the mixture (curve C) is indeed a combination of the previously described curves. The first peak value is lower than for parallel loading and the highest peak at the end of the process is lower than for the perpendicular case.

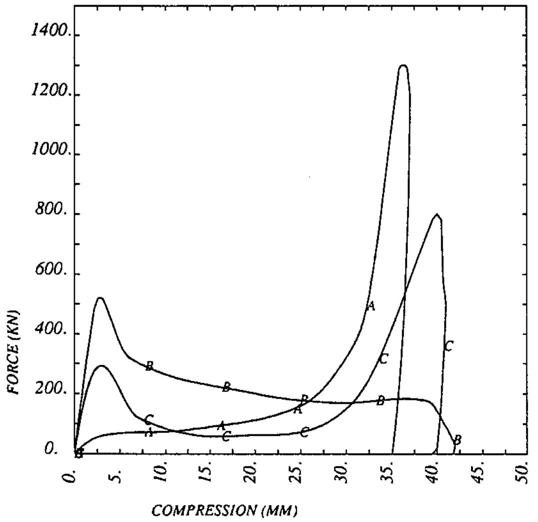

Fig. 1 : Deformation Behaviour of Wood

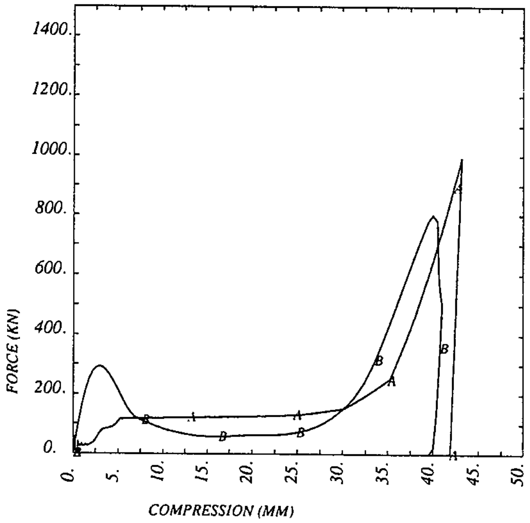

Fig. 2 : Comparison Experiment - Simulation

Common to all experiments is the observation that the wood shows little lateral extension which means that the developed cross strains are relatively small.

It is clear that the available tests are insufficient to evaluate a general material model for wood. Nevertheless an engineering solution needed to be attempted. Therefore the following assumptions were made:

- Cross strains are neglected, which means that Poisson's ratio is zero at any state in the crushing process. An existing material model was modifed according to this assumption by establishing the appropriate relationship between the instantaneous Bulk modulus K and Shear Modulus G.

- The crushing is a purely volumetric process i.e. only the hydrostatic part of the stress tensor causes plastic deformation. Plastic shear deformation is neglected.

- As in the shock absorbers layers with mixed grain directions are used, this mixture is assumed to behave isotropically. This seemed reasonable, taking into account that the wood is surrounded by steel plates so that orientation dependent deformation mechanisms like splitting are unlikely to occur.

Based on these assumptions a FE-model of the drop test was set up. A user defined pressure - volumetric strain relationship was found, such that the force-deflection behaviour of the mixed specimen is approximated. In doing so the first unloading phase seen in figure 1 was not taken into account because this is known to lead to problems in a numerical treatment. Instead the aim was to produce a curve which represents the same amount of energy absorption i.e. the areas under both the experimental (curve B) and the calculated curve (curve A) are the same (see figure 2).

4. EDGE IMPACT SIMULATION

The load case under consideration is a standardized 9 m fall of the cask onto the edge of the protecting shock absorbers with the ground assumed to be rigid. The initial position is such that the center of gravity of the container is located directly above the impacted edge.

The cask itself consists of the actual final disposal container made from an Magnesium alloy. The material behaviour of this alloy is assumed to be elastic - perfectly plastic. The content of transport is modelled with solid elements of a relatively low stiffness. The modelling of the transport container is more complicated. As the sidewall of this part is interrupted by moderator holes filled with absorbing material, it was chosen to use an orthotropic material model instead of meshing these holes in detail. The top of the transport container is in reality fixed by a series of screws. This connection is also not modelled in detail but it is simply assumed that a continuous mesh is sufficient.

An important aspect in the cask FE model is the use of so called slidelines. These are used whenever two or more structural parts are in contact, which is the case here between the content of transport and the final disposal container and between the final disposal container and the transport container. Thereby relative movements between these parts are allowed which can be of importance for the assessment of stresses and strains in the container upon impact. The location of the slidelines is indicated in figure 3, where the complete model of the cask is shown. As can be seen in figure 3, the meshing of the impacted shock absorber is much finer than the mesh of the container itself. This is because it was expected that all plastic deformation will be concentrated in the absorber and the details of theses deformations determine the overall container loading. The modelling strategy was first to generate a shell element model of the steel cage (see figure 4) and then to fill the hollow space with solid elements to represent the wood layers. There use was made of the previously described wood material model.

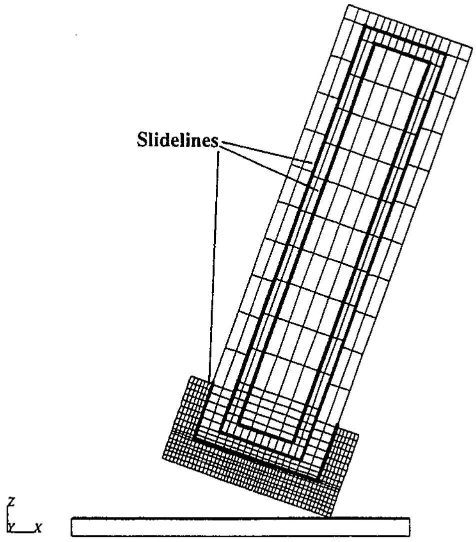

Fig. 3 : FE - Model

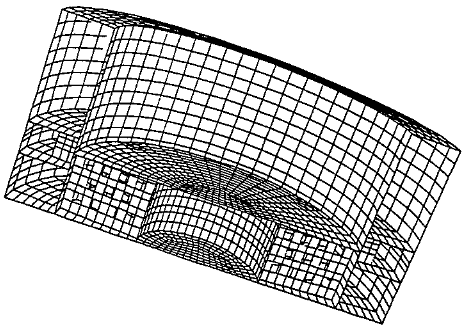

Fig. 4 : Cage - Model

Between the container and the absorber another slideline is used. The mass of the second shock absorber is taken into account by added nodal masses at the top face of the cylinder. Gravity is neglected, because it is expected, that the relevant stresses occur in the first impact phase which is relatively short. The sidewards movement which follows the first impact phase is not considered. In the simulation the container is given an initial velocity of 13.29 m/s which corresponds to the drop height of 9 meters. For the model the single symmetry of the problem is taken into account and only half the structure is meshed. The total mesh consists of 6800 solid elements and 3800 shell elements which leads to a cpu time of 5 to 10 hours on a modern engineering workstation.

As usual for explicit FE simulations, the conservation of energy and momentum is used to check the quality of the calculation. A time-history plot of the impact force (figure 5) shows that the impact lasts approximately 50 ms with a maximum value of 44000 kN at 30 ms. A view of the deformed structure at this time is given in figure 6. The shock absorber is heavily deformed, but the lower edge of the cask is still well above the ground.

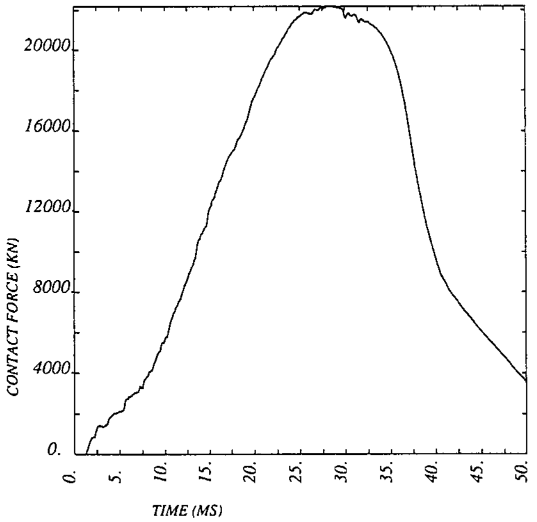

Fig. 5 : Impact Force

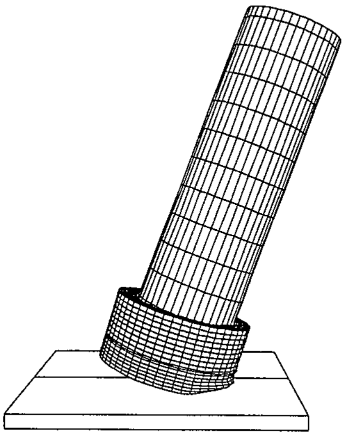

Fig. 6 : Deformed Structure

A closer view of the deformations of the cage is given in figure 7. To investigate the overall deceleration of the structure, an average of the deceleration time-histories of a number of nodes at the top of the container is used, where a maximum of 65 g is reached at about 32 ms.

Besides these more global results, details of the stress distribution in the container are of particular interest in such a simulation. Therefore both contour plots of the stress in the structure at a certain time, as well as time-histories of stress at certain locations can be used. As an example the contours of the maximum principle stresses are shown in figure 8 for the bottom of the container. It can be seen that the highest stresses occur approximately in the middle of the bottom face of the transport container. In this way any location in the structure can be investigated and the stress values found can be used e.g. for fracture mechanics considerations.

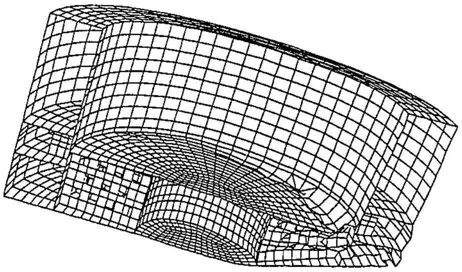

Fig. 7 : Deformations of the Cage

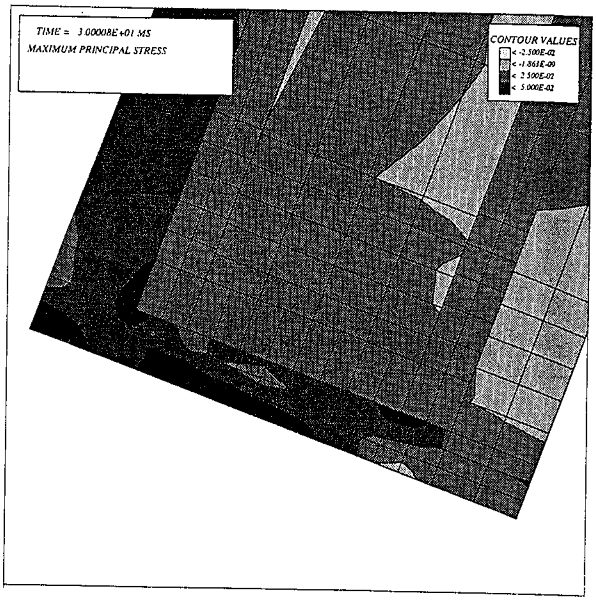

Fig. 8 : Stress Contours

5. CONCLUSIONS

In this paper a Finite Element model suited for a drop test of a nuclear spent fuel cask has been presented. An essential part of this model is an approach to model the material behaviour of the wood which is used in the shock absorbers of the investigated package. Based on impact tests with wooden specimens, material parameters for the wood model were found which allowed to capture the relevant properties of the wood which is in this case the ability to absorb energy by volumetric plasticity. The correct functioning of this model has been demonstrated in numerical simulations with a full model of the container. The results allowed to investigate the structural loading in such an impact situation and to provide data to assess the safety of the container construction.

6. REFERENCES

[1] Haug, E., Ulrich, D., The PAM-CRASH Code as an efficient Tool for Crashworthiness Simulation and Design, The Second European Cars/Trucks Simulation Symposium, Schliersee (Munich), 1989