COMPUTATIONAL DYNAMIC FRACTURE ANALYSIS FOR DROP TESTS OF SHIPPING CASKS

M. Enderlein [1], W. Völzer [2], A. Ricoeur [1], M. Kuna [1]
[1] Technische Universität Bergakademie Freiberg, Institute of Mechanics and Fluid Dynamics, 09596 Freiberg, Germany
[2] Gesellschaft für Nuklear-Behälter mbH, 45127 Essen, Germany

ABSTRACT

For the development of nuclear shipping casks the integrity of the structure has to be guaranteed under all operational and accident conditions. Among others this includes dynamic loading conditions such as drop tests. In order to determine the safety margin against brittle fracture numerical fracture mechanical methods are used. This paper deals with the numerical implementation of the dynamic J-integral as one method to compute the dynamic stress intensity factor for mode-I loading. To demonstrate the applicability of this method, a simulation of free drop test of a cask with an artificial flaw is performed using a submodelling technique as well. The results of the simulation are contrasted with the outcome of the experiment.

INTRODUCTION

Since the failure of nuclear transport shipping casks may have serious consequences, the integrity of the structure has to be guaranteed under severe test conditions, regulated by the guidelines given by the IAEA [1]. Among others this includes impact conditions. To analyse the safety margin against brittle failure, fracture mechanical methods are applied. To assess the behaviour of a crack usually stress intensity factors are calculated, which in general are a function of time. For a dynamic mode I load this is the Dynamic Stress Intensity Factor (DSIF) K_I^d. For the calculation of the DSIF analytical methods can rarely be used. To analyse cracks in structures of arbitrary shape under dynamic loading mostly numerical techniques, e.g. the Finite Element Method (FEM) have to be applied, which gives an approximate solution of the boundary value problem with crack. For the numerical computation of the DSIF several methods have been developed in the past. One of the most useful techniques is the J-integral. Its applicability for the fracture mechanical evaluation of technical structures is demonstrated in this paper.

THE DYNAMIC J-INTEGRAL

The J-integral is based on theoretical articles of Atkinson and Eshelby [2] and Kostrov and Nikitin [3]. It can be derived from an energy balance within a domain V, see Fig. 1. With S_Γ as a surface enclosing a crack front segment Δs and assuming linear elastic material the J-integral vector J_k^d is given by:

$$J_k^d = \lim_{S_\Gamma \to 0} \int_{S_\Gamma} \left[\left(U + T \right) \delta_{kj} - \sigma_{ij} u_{i,k} \right] n_j dS_\Gamma \ .S \tag{1}$$

The quantities σ_{ij} and u_i are the Cartesian components of the stress tensor and the displacement vector, n_j is the unit normal vector pointing outward of the enclosed domain and δ_{kj} is the unit tensor. U denotes the specific elastic energy stored in a volume element and T means the kinetic energy density. For an efficient numerical evaluation Eq. (1) is transformed into an equivalent domain integral [4]. For a stationary crack with traction-free crack surfaces and without body forces this yields:

$$\bar{J}^d = \int_V \left(\sigma_{ij} u_{i,k} - U \delta_{kj} \right) q_{k,j} \, dV + \rho \int_V \ddot{u}_i u_{i,k} q_k \, dV \ . \tag{2}$$

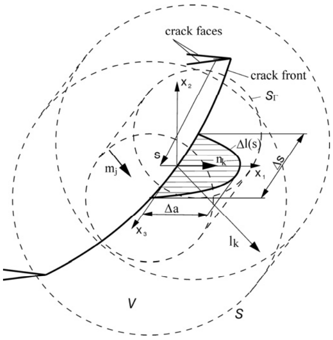

Figure 1: Integration region and virtual crack extension

Therein $\overline{J}^d$ depicts an average value along the crack front segment Δs and q_k denotes a vectorial weighting function (Moran and Shih [4]). The local value of the J-integral at the crack front position s can be calculated from:

$$J^d(s) = \frac{\overline{J}^d}{\int\limits_{\Delta s} l_k n_k \, ds}, \tag{3}$$

where l_k is a dimensionless vector, which defines the area of virtual crack extension in the x_1-x_3-plane (e.g. Helen [5]), see Fig. 1. To compute the Dynamic Stress Intensity Factor K_I^d (DSIF) from the J-integral, the plane strain relation

$$K_I^d = \sqrt{\frac{EJ^d}{1-\nu^2}} \tag{4}$$

is used, where E is the elastic modulus and ν is Poisson's ratio.

To calculate J^d in terms of a finite element analysis, the integral in Eq. (2) has to be computed numerically, see for instance [4]. In order to validate the implementation of the dynamic J-integral, several numerical examples have been analysed, leading to very accurate results. Detailed information about the implementation and the validation are given in [6].

BASES OF THE DROP TEST SIMULATION, FINITE ELEMENT MODEL

The simulation presented here is based on the CRIEPI study [7], which describes a 9 m free drop test of a ductile cast iron cask with an artificial crack at a temperature of -40 degrees Celsius. Fig. 2 shows the main dimensions of the cask and the crack.

For the simulation of the drop test the finite element solver ABAQUS/Explicit was used. This software works with an explicit time integration scheme, which in dynamic calculations shows clear advantages over implicit methods regarding the computational rate. For the calculation of the DSIF, a postprocessor was developed performing the above mentioned numerical computation of the dynamic J-integral. In Fig. 3 the finite element model of the cask is shown. It mainly consists of the following components: cask body, secondary lid, fuel rod carrier and two shock absorbers. Because of the symmetry with respect to the axial cross section, only one half of the cask was modelled using first-order hexahedral elements. All geometric data and material properties are taken from the CRIEPI study [7].

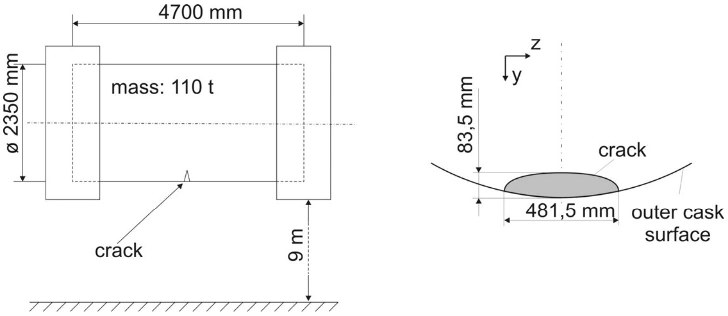

Figure 2: Main dimensions of the cask and the artificial crack

The cask body is made of cast iron and has two rings of axial holes, which host the moderator rods when the cask is in operation. The sections containing the holes were substituted in the model by regions with an equivalent flexural stiffness and a reduced density.

The secondary lid is assembled to the cask with bolds in axial direction. They are realized in the model using spring elements with an equivalent stiffness. In radial direction a clearance of 3.5 mm between the lid and the cask body is assumed. To avoid a penetration of the lid and the cask body during the simulation, all surfaces of the lid and the cask which are or may come in contact are modelled as contact pairs involving friction. The primary lid was not installed during the real experiment and thus has not been considered in the simulation.

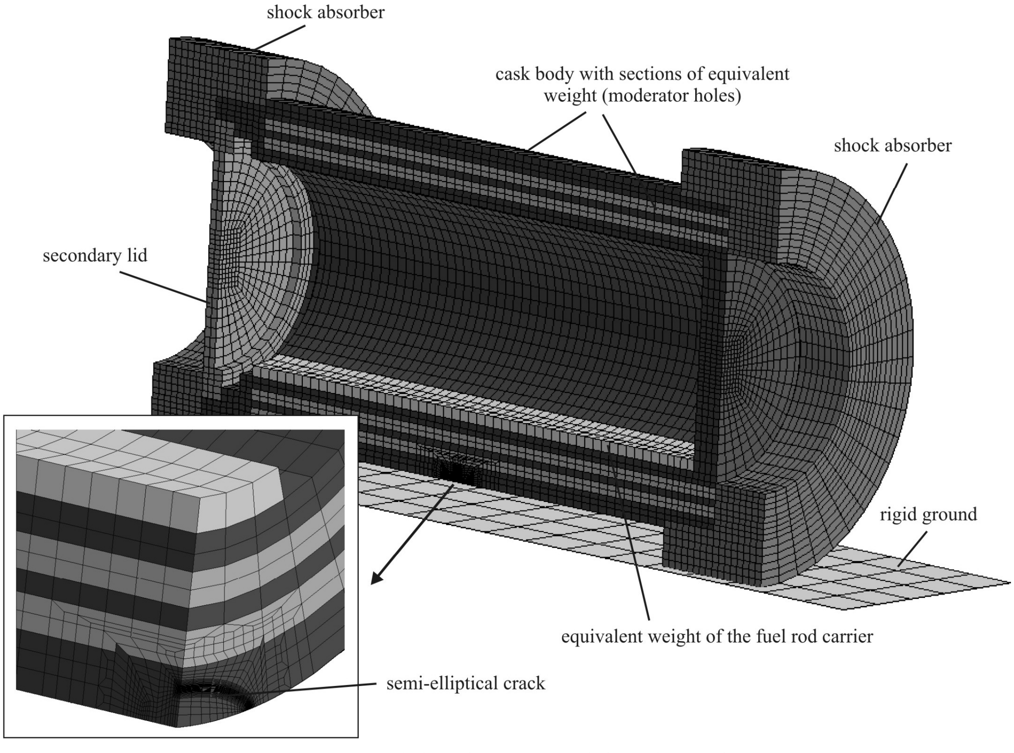

Figure 3: Finite element model of the cask

During the drop test the cask was loaded with a dummy of the carrier of the fuel rods. Because of the clearance to the inner face of the cask, this dummy has no influence on the cask stiffness. In the finite element model the dummy is realized by one additional layer of elements on the inside base of the cask, see Fig. 3. These elements have a high density to simulate the weight of the dummy and a very low elastic modulus to avoid an increase in stiffness.

On both ends the cask is equipped with shock absorbers made of ply wood with a carbon steel liner. The difficult material behaviour of the absorbers was modelled in homogeneous form as elastic-plastic material using the stress-strain-curve given in the report. At all surfaces of the absorbers adjacent to the cask body, to the secondary lid and to the ground, respectively, a contact algorithm involving friction is defined.

RESULTS OF THE SIMULATION

During the real drop test a number of strain gauges and accelerometers were applied to the cask, measuring the axial and the circumferential strains and the acceleration. Based on the recorded data [7] the quality of the numerical simulation has been assessed, comparing global strain values and corresponding time intervals in the simulation and the drop test. As a result a good agreement was found, especially concerning the axial strain values, which are most important for the fracture mechanical assessment due to the orientation of the crack faces. For details see [8].

To assess the behaviour of the crack K_I^d is computed as a function of the time t and the angle ϕ, which defines the position at the crack front. At $\phi = 84°$ the crack front meets the outer surface of the cask. The results of the calculation are given in Figs. 4 and 5. In the first graph K_I^d is plotted depending on the time for three selected angles ϕ. As can be seen, all curves have an absolute maximum at $t^* = 16$ ms. The highest value of K_I^d is reached by the curve for $\phi = 0°$. This fact is shown more clearly in Fig. 5, where K_I^d is plotted along the crack front for the time t^*. The maximum of K_I^d is 53 MPam$^{1/2}$. It occurs at the minor axis of the semi-elliptical crack, which lies in the symmetry plane of the model. Following the curve in the direction to the outer surface of the cask, K_I^d decreases and has a minimum of about 33 MPam$^{1/2}$. The low increase after $\phi = 81°$ is caused by numerical effects.

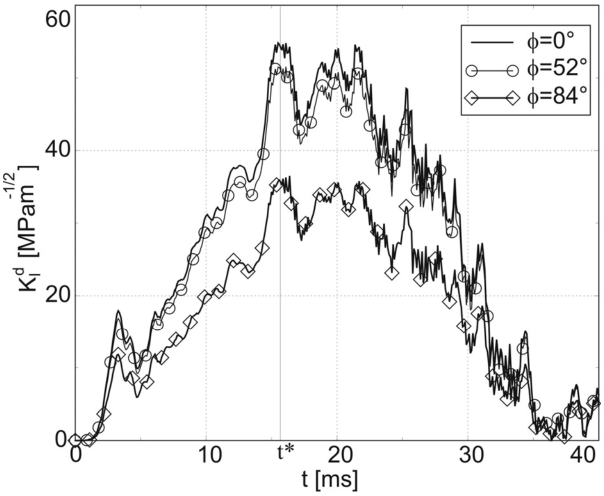

Figure 4: Results for the DSIF as a function of time

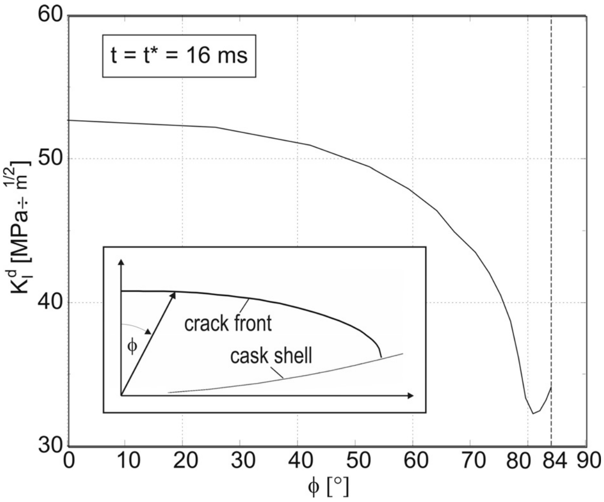

Figure 5: Results for the DSIF as a function of the crack front position

In the CRIEPI study a Dynamic Fracture Toughness of K_{Id}= 69 MPam$^{1/2}$ is given for the cask material at a temperature of -40 degrees Celsius. This is about 30% higher than the calculated DSIF and supports the observed results of the CRIEPI drop test that no crack initiation was detected.

In order to reduce the effort concerning modelling and computational costs the applicability of a submodel technique for dynamic load cases was examined. The principle of this technique is that first a simulation is performed using a global model without crack. After that the local model of the crack and its environment is analysed, using the time dependent displacements from the global analysis as boundary conditions for all edge nodes of the submodel. Fig. 6 shows a detail of the finite element model together with crack submodels of different radial dimension. The results of the calculations are given in Fig. 7. It can be seen, that the solutions approach the reference solution with increasing size of the submodel. A further enlargement in circumferential direction leads to a nearly exact correlation with the reference solution.

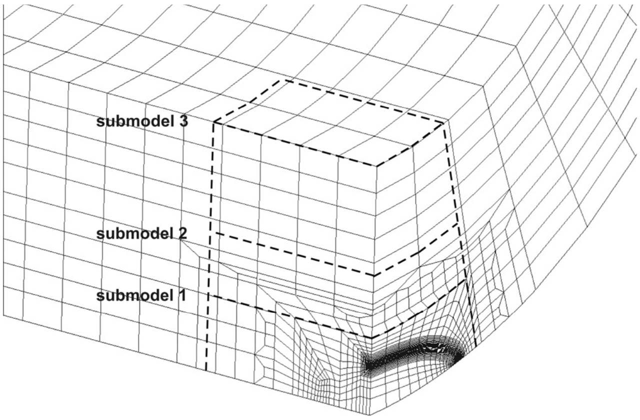

Figure 6: Size of the examined submodels

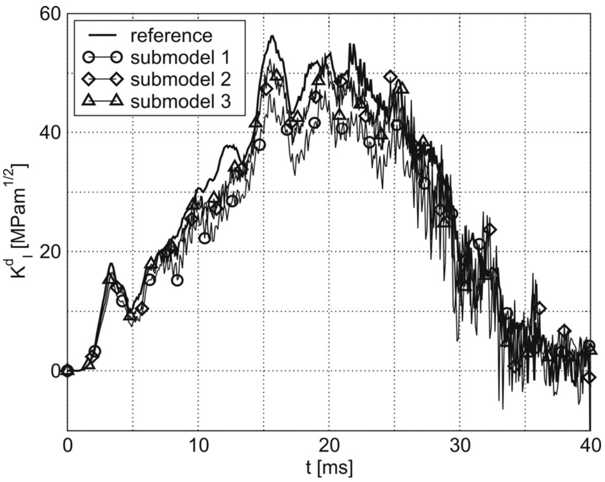

Figure 7: Influence of the submodel-size on the DSIF

CONCLUSIONS

A dynamic finite element simulation of the free drop test of a ductile cast iron shipping cask has been performed. For the fracture mechanical assessment of the crack, the dynamic J-integral was evaluated. As a result it turned out that the maximum dynamic stress intensity factor amounts approximately 30% less than the fracture initiation toughness. This confirms the results from the real drop test, where no crack propagation was found. It has been shown by the simulation that the J-Integral method is a very useful technique to compute the stress intensity factor for cracked structures under dynamic loading conditions. In combination with a submodel technique the computational effort can be limited without loss of quality of the results.

REFERENCES

[1] International Atomic Energy Agency (IAEA), Regulations for the Safe Transport of Radiaoactive Material, Edition (Revised), Regulations No. TS-R-1 (ST-1, Revised), IAEA, Vienna, 2000.

[2] Atkinson, C., Eshelby, J.D., Flow of energy into the tip of a moving crack. J. Fracture Mech., 4, 1968, 3-8.

[3] Kostrov, B.V., Nikitin, L.V., Some general problems of mechanics of brittle fracture. Archiwum Mech. Stosow., 22, 1970, 749-775.

[4] Moran, B., Shih, C.F., Crack tip and associated domain integrals from momentum and energy balance. Engng Fracture Mech., 27, 1987, 615-642.

[5] Hellen, T.K., On the method of virtual crack extension. Int. J. numer. Meth. Engng, 9, 1975, 187-207.

[6] Enderlein, M., Ricoeur, A., Kuna, M., Comparison of finite element techniques for 2D and 3D crack analysis under impact loading. Int. J. of Solids and Structures, 40, 2003, 3425-3437.

[7] CRIEPI Report, Integrity of cast-iron cask against free drop test, Part III: Verification of brittle failure, design criterium. Abiko Research Laboratory, 1990.

[8] Enderlein, M., Klein, K., Kuna, M., Ricoeur, A., Numerical fracture analysis for the structural design of CASTOR casks. SMIRT 17, 2003.