# FROM EXPERIMENT TO AN APPROPRIATE FINITE ELEMENT MODEL-SAFETY ASSESSMENT FOR DUCTILE CAST IRON CASKS DEMONSTRATED BY MEANS OF IAEA PUNCTURE DROP TEST

**Mike Weber** **Frank Wille**

**Viktor Ballheimer** **Andre Musolff**

BAM Federal Institute for Materials Research and Testing
Berlin, Germany

## ABSTRACT

In the approval procedure of transport packages for radioactive materials, the competent authority mechanical and thermal safety assessment is carried out in Germany by BAM Federal Institute for Materials Research and Testing.

The combination of experimental investigations and numerical calculations in conjunction with materials and components testing is the basis of the safety assessment concept of the BAM.

Among other mechanical test scenarios a 1 meter drop test onto a steel bar has to be considered for hypothetical accident conditions of Type B packages according to IAEA regulations.

Within the approval procedure for the new German package design of the HLW cask CASTOR® HAW 28M, designed by GNS Gesellschaft für Nuklear-Service Germany, a puncture drop test was performed with a half-scale model of the cask at -40°C.

For independent assessment and to control the safety analysis presented by applicant, BAM developed a complex finite element model for a dynamical ABAQUS/Explicit™ analysis. This paper describes in detail the use of the finite element (FE) method for modeling the puncture drop test within an actual assessment strategy.

At first investigations of the behaviour of the steel bar are carried out. Different friction coefficients and the material law of the bar are analysed by using a “rigid-body" approximation for the cask body.

In the next step a more detailed FE model with a more realistic material definition for the cask body is developed. Strain verification is possible by results of the strain gauges located at the relevant points of the cask model. The influence of the finite element meshing is described.

Finally, the verified FE half-scale model is expanded to full-scale dimension. Scaling effects are analysed. The model is used for safety assessment of the package to be approved.

## INTRODUCTION

According to IAEA regulations [1], § 727, Type B(U) packages for the transport of radioactive material must withstand accident transport conditions resulting from a 9 m free drop onto an unyielding target in sequence with a 1 m puncture bar drop in a most damaging attitude. These mechanical tests are supplemented by a thermal (30 minutes at an average temperature of 800 °C) and a water immersion test. The presented paper focuses on the mechanical 1m puncture bar drop test.

Fig. 1 gives an overview of the drop test program performed by BAM during the licensing procedure. The drop test positions and sequences were derived from the regulatory requirements. A combination of experimental investigations and numerical calculations in connection with material and component testing is the basis of the safety assessment concept applied.
One of the goals of the drop tests performed during the approval procedure was to demonstrate that impact limiters remain joined to the cask body during and after impact. Furthermore the drop test had to ensure the compliance with regulatory requirements [1] concerning leak tightness. For this purpose leakage tests has been performed before and after tests [1].
Numerical calculations are necessary to consider additional drop positions and to investigate the influence of modified impact angles. It gives possibility to perform calculations by using material properties at various temperatures and to identify stresses and strains at positions in the inner cask structure which cannot be instrumented my measurement equipment.
Additionally the experimental drop tests give the possibility to verify the finite element models. Hereby the measured values of strains and decelerations were compared with calculated values of specific points of interest. To describe the stepwise development and the use of a finite element model in detail, just the 1 m puncture bar drop test has been selected. All calculations have been done with the dynamical explicit finite element code ABAQUS™ [7].

## DROP TEST

The concept fulfilling regulatory requirements based on performing experimental drop testing, Fig. 1 shows the comprehensive drop test program for the licensing procedure. All tests were performed with a half-scale drop test model, manufactured just for this specific purpose. All drop tests were carried out at the BAM Test Site Technical Safety nearby Berlin.
It was decided to cool down the drop test model to the lowest regulatory required temperature of -40 °C for one drop test sequence. The 1 m puncture bar drop test considered in this paper was part of this sequence. The main goal for this specific drop test was to get information about stress maximization in the cask wall, values for the appropriate analysis of fracture mechanics and material behaviour at lowest temperatures in general. The drop height of 1 m, measured between outer surface of the cask and the top of the puncture bar, causes a velocity of 4.43 m/s at the moment of first contact of cask and bar.

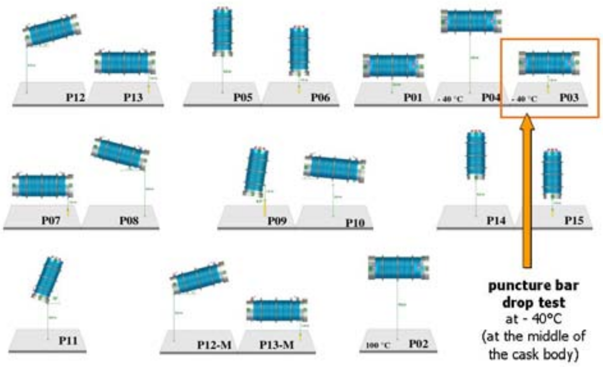


**Figure 1. Drop test program**

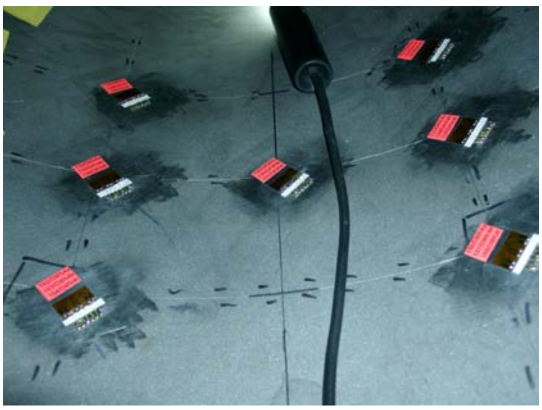

**Figure 2. Strain gauges at the inner wall side of the cask**

For purpose of verifying the FE model that has to be generated, the cask is equipped with strain gauges and accelerometers. The instrumentation in the inner of the cask above impact area of the puncture bar is shown in Fig. 2. The large strains in this area are of particular importance during the later verification process. It is important to consider other measurement results for instance from strain gauges in bottom or lid area of the cask as well.

## FINITE ELEMENT MODEL

### Verification of Puncture Bar’s Behaviour

At first a simplified finite element model was generated (Fig. 3). The cask was reduced to a rigid body because up to 98 % of the energy is dissipated in plastic deformation of the puncture bar. Therefore this simplification is acceptable and causes only a small deviation concerning the puncture bar’s deformation behaviour. The outer diameter of the cylinder and the mass is equal to the values of the drop test model. The comparison of measured and calculated puncture bar height after impact gives information on the quality of finite element material model. Based on comprehensive material investigations, the plastic material behaviour of the puncture bar is represented by strain rate dependent true stress versus logarithmic strain curves, the elastic material behaviour is characterized by Young’s modulus and Poisson’s ratio. The chosen friction coefficient affects the expansion of puncture bars diameter at the contact surface to the cask. Different results of calculated puncture bar deformation compared to deformed bar after drop test are shown in Fig. 4. The friction coefficient that leads to the “correct barrel shaping” is used with the detailed half-scale model calculation, described in the following.

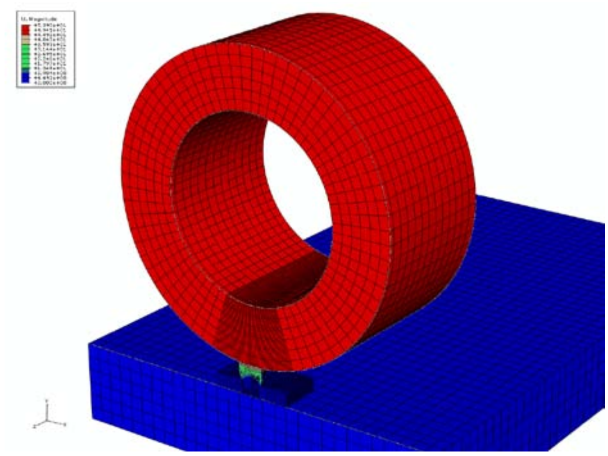

**Figure 3. Rigid body model**

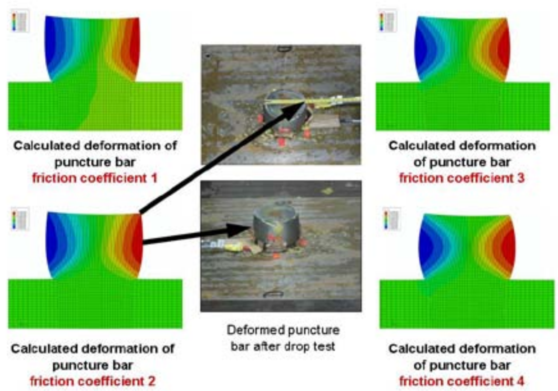


**Figure 4. Comparison of puncture bar deformation**

Half-Scale Model

In a next step, the rigid body approximation of the cask body was replaced by a more detailed modeling of its geometry (Fig. 5) and material behaviour. The mass of cooling fins and all other neglected small parts of the construction were considered by an increased density of the cask body. The material behaviour of the cask body was described with strain rate dependent true stress versus logarithmic strain curves. Dummy mass components to represent both the bottom- and lid-side impact limiters and the content (canisters with vitrified waste) guarantee a realistic distribution of mass along the cask length. The Young's Modulus of the simplified content modeling was chosen very small to avoid an unintended increase of cask body stiffness.

According to IAEA regulation [1] the minimum initial length of the puncture bar is 20 cm (Ø 15 cm). Considering the scale factor of 2, the length of the bar was 10 cm (Ø 7.5 cm) for drop testing and for modeling as well. In this case, initial length of puncture bar was defined with the objective of no contact of cooling fins to mounting plate during impact. Unintentional dissipating of energy can be avoided with this boundary condition.

An appropriate mesh density in the contact area was found by a convergence study, performed with a finite element model, reduced to the cask body. Beginning with a coarse mesh, the size of elements was bisected in two steps. It can be seen that the normalized maximum principal stress at the outer borehole approaches a limit curve (Fig. 6). Considering both precision and required computing time, density of 1$^{st}$ refinement step is chosen for the following verification of the finite element model. A detailed description of the complete convergence study is given in [6].

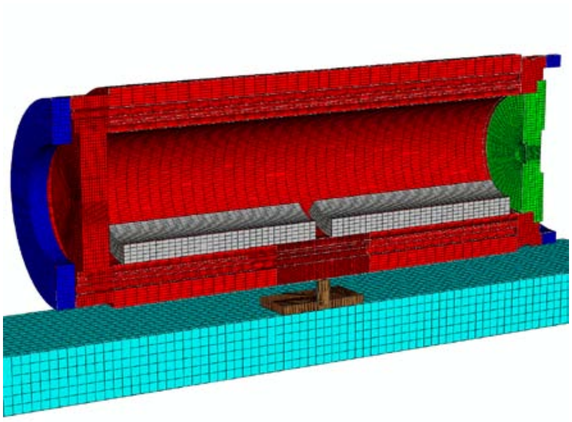

**Figure 5. Cross section through FE model**

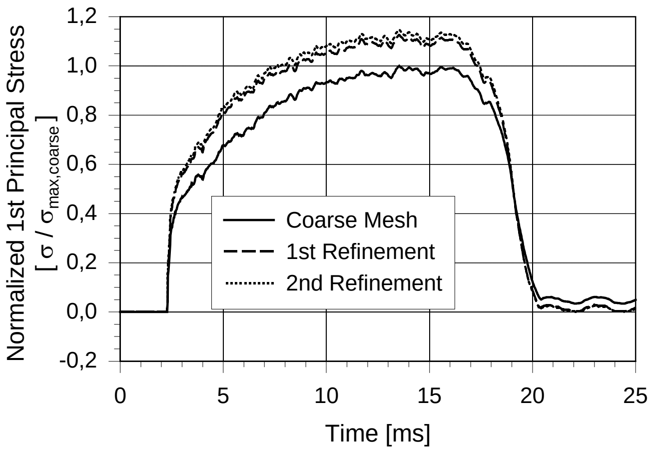


**Figure 6. Convergence during mesh refinement**

Before the verification process was started, the global kinematic behaviour of the finite element model had to be checked. The energy balance was plausible. The rate of artificial energy, caused by "hourglassing", remained very small (<<1 % referred to the entire model). The measured strains were compared to calculated strains. The calculation was analyzed at integration points and at surface of the elements assigned to position of strain gauges during drop test. For this purpose, one-dimensional truss elements were joined with the element nodes at the cask body surface. So it is possible to measure the strains directly, independent of their inclination in the global coordinate system. As an example Fig. 8 shows the comparison of values at a position nearby the intersection point of puncture bar's centre line and the inner surface of the cask. Maximum value, duration of impact and the whole curve shape match well. The predominant state of bending stress causes slightly higher calculated strains in the truss elements. Compared to Fig. 6 the duration of impact is approximately 3 ms longer. As mentioned, the convergence study was performed using a model reduced to cask body and not with the whole package. Drop height remained unchanged and therefore the difference in mass generates difference in impact duration.

Despite of a considerably lower level of strains the measurement points away from the impact area were considered too. The values fit well together. As a result of verification a correction factor had to be defined. It is a criterion of quality of the generated finite element model. The factor had to be considered in the further process of safety assessment.

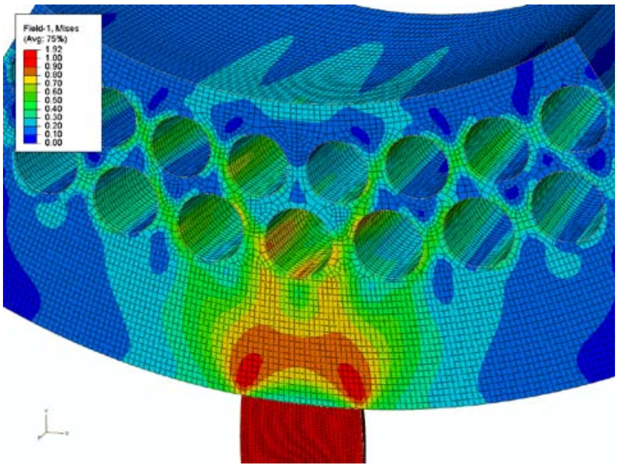

**Figure 7. Normalized effective stress (von-Mises) at time of maximum force**

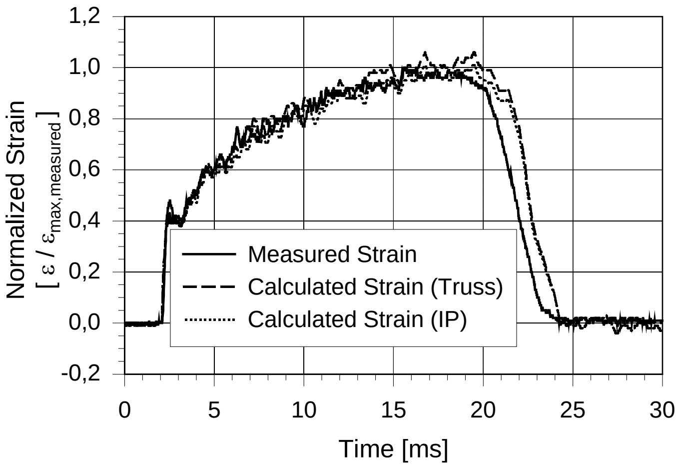


**Figure 8. Comparison of measured and calculated normalized axial strains**

<u>Full-Scale Model</u>

The acceptance of reduced-scale drop testing based on the correct application of appropriate similarity theory. The validity of the similarity considerations respectively their limits had to be checked. For this purpose, the verified finite element model was scaled up by doubling of all node coordinates in the ABAQUS® input file. Initial and boundary conditions were not changed. Applied material laws and densities are identical to the half-scale model. According to similarity laws, stresses and strains should be nearly unchanged, impact duration doubled and forces should be quadruplicated if all lengths are doubled. Fig. 9 shows the run of curves of normalized contact force during impact of both the half-scale and the full-scale model. The diagram axes are adapted to the aforementioned relation of time (1:2) and force (1:4), a direct comparison is possible. The curves match very well with the exception of a small difference in duration of impact. At drop tests, the load of specimen is affected by the release of potential energy. Due to the higher (absolute) deformation of puncture bar, the amount of additional relative drop energy is higher at the full-scale cask and so “scaled” impact time differs slightly. To compensate this effect, a drop height adaption could be necessary where required [4] [5]. A good correlation of stresses and strains at same (scaled) time indicates a slight influence of non-scaled strain rate effects caused by the used scale factor in this case and confirms the use of the half-scale model applied.

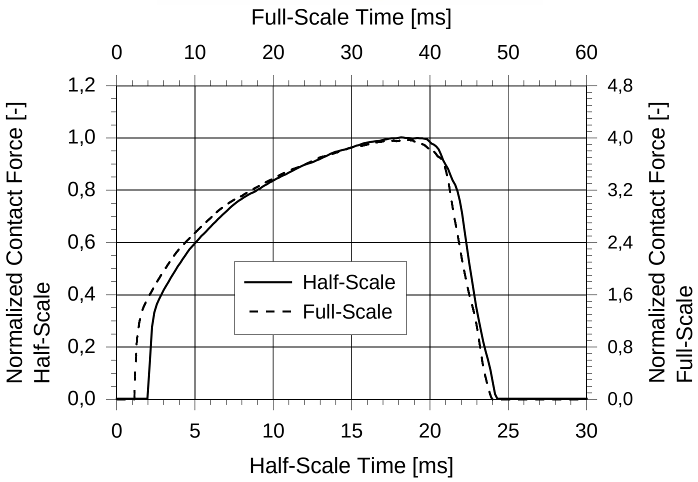


**Figure 9. Normalized puncture bar force**

<u>Finite Element Model for Safety Analysis</u>

Finally, the up-scaled finite element model has to be adapted to the requirements of safety assessment itself.

According to IAEA regulation [1], the target has to be modeled as unyielding, a “rigid body” modeling was applied. Finite element material model of the cask body was adapted to properties defined at the valid material specification. Calculations were carried out using material properties at temperatures of -40 °C and 150 °C. While the calculated stresses were analyzed for both cases, the load factor of strains was considered only at 150 °C. According to IAEA regulations, initial length of puncture bar has to be chosen in the way of causing the maximum damage. It is conservatively assumed that both the impact limiters at bottom- and lid-side and the three aluminium rings around the cask body do not remain at the package after 9 m (+0.3 m) drop test(s) belonging to the sequence. The initial length of the puncture bar has to be minimized (≥20 cm, [1]) but the cooling fins must just not touch the target when the puncture bar deforms during impact. While the material model of puncture bar at simulation of drop test with the half-scale model corresponded to the real material used, now, it describes an “artificial” material. The “mild” steel matches the requirements of IAEA advisory material article 727.13 (150 MPa $\leq R_e \leq$ 280 MPa, $R_e / R_m \leq 0.6$).

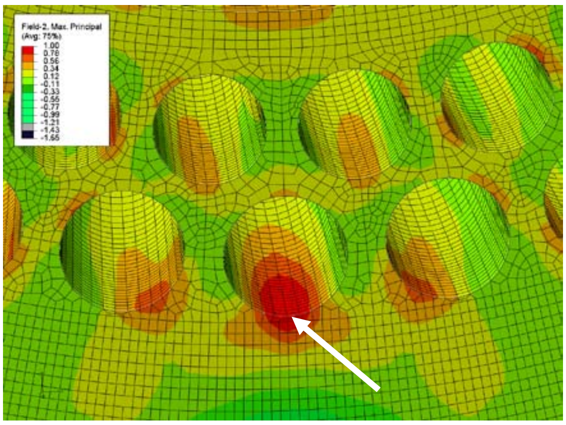

**Figure 10. Distribution of normalized principal stress at 36 ms**

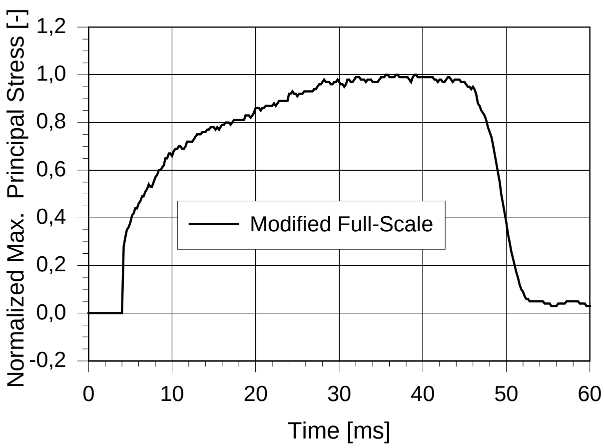


**Figure 11. Normalized principal stress at outer borehole above puncture bar**

Results of this finite element calculation are directly applicable for safety assessment of package design. Fig. 10 shows exemplary the distribution of maximum principal stress at time of highest load. The run of curve for the element, pointed by the arrow, is shown in Fig. 11. Detected loads, such as strains and stresses, have to be compared with design loads, dependent on the chosen assessment concept. The correction factor, defined as a result of verification of half-scale model, has to be considered.
Additionally, this global model is the basis for analysis of fracture mechanics that have to be carried out if ductile cast iron is exposed to dynamic loads at low temperatures. At all positions, where a defined dimension of principal stress is exceeded [2], the fracture mechanical behaviour has to be investigated more extensively. An artificial crack is assumed and simulated within a sub-model. A material failure of the chosen (assumed) size in calculation has to be prevented at the real package by non-destructive testing methods as for instance by ultrasonic analyzers.
A sufficient correlation between the results of applicant and BAM's independent calculations is one of the most important test criterions within the assessment procedure.

## CONCLUSIONS

For independent assessment and to control the safety analyses presented by applicants, there are different possibilities. In the present case of approval procedure for the new German package design of the HLW cask CASTOR® HAW 28M, BAM decided to generate independent finite element models to simulate the drop tests identified as particularly important. To describe the steps of development on the way to a finite element model suitable for assessment of package design, the 1 m puncture bar drop test was chosen as an example. Because of the small number of impact partners, well investigated material behaviour of the components involved and availability of necessary material laws, the puncture bar test is well suited for generating a dynamical finite element model. Due to localization of high plastic deformations in the contact zone between the bar and cask body a differentiated mesh strategy adapted to the load level in the different areas of the cask is possible.
All steps of model development base on generally accepted guidelines and therefore they are very similar to the applicant ones. Otherwise, BAM performed the necessary calculations using the dynamical finite element code ABAQUS/Explicit™ in contrast to LS-DYNA™, used by the applicant. This approach guarantees an independent test procedure and fulfils the requirements of [3].
The measured values, detected during drop testing, give the possibility for verifying the finite element model. After up-scaling and adaption to the attributes of package, a suitable tool for independent assessment of package design is available.

## REFERENCES

[1] IAEA - International Atomic Energy Agency (Ed.): Regulations for the Safe Transport of Radioactive Material, 2009 Edition (Revised). Safety Standard Series, Vienna: IAEA - International Atomic Energy Agency, 2009.

[2] Bundesanstalt für Materialforschung und –prüfung (BAM): BAM–GGR 007–Leitlinie zur Verwendung von Gusseisen mit Kugelgraphit für Transport- und Lagerbehälter für radioaktive Stoffe, Rev.0, Juni 2002

[3] Bundesanstalt für Materialforschung und –prüfung (BAM): BAM–GGR 008–Richtlinie für numerisch geführte Sicherheitsnachweise im Rahmen der Bauartprüfung von Transport- und Lagerbehältern für radioaktive Stoffe, Rev.0, Febr. 2003

[4] Quercetti, T; Müller, K; Schubert, S.: Comparison of experimental Results from Drop testing of a Spent Fuel Package Design Using a Full-scale Prototype Model and a Reduced-scale Model. Packaging, Transport, Storage & Security of Radioactive Material, Vol.19, 2008, No.4, pp. 197-202

[5] Wille, F.; Ballheimer, V.; Droste, B.: Suggestions for the correct Performance of IAEA 1m Penetrator Bar Drop Test with reduced-scale Packages considering Similarity Theory Aspects. Packaging, Transport, Storage & Security of Radioactive Material, Vol.18, 2007, No.2, pp. 111-116

[6] Zencker, U.; Wille, F.; Weber, M.: Finite Element Mesh Design of a Cylindrical Cask under Puncture Drop Test Conditions. PATRAM 2010, London, UK, Paper No. 219

[7] ABAQUS Version 6.8, 2008, Dassault Systèmes Simulia Corp., Providence, RI.