Static and Dynamic Calculation Approaches for Mechanical Design Assessment of Type B Packages for Radioactive Material Transport - 10193

Steffen Komann, Martin Neumann, Viktor Ballheimer,
Frank Wille, Mike Weber, Linan Qiao, Bernhard Droste

BAM Federal Institute for Materials Research and Testing
Berlin, Germany

ABSTRACT

Demonstration of compliance with the IAEA regulatory requirements [1] for Type B packages can be accomplished by experimental tests with prototypes or small scale models, by reference to previous satisfactory demonstrations of a sufficiently similar nature as well as by calculation, when the calculation procedures and parameters are generally agreed to be reliable or conservative. In practice a combination of these methods is usually applied. This paper shows exemplarily how numerical and experimental approaches can be combined reasonably in the assessment of mechanical package integrity. The paper will also concentrate on how static mechanical approaches can be applied and what their problems related to dynamic calculation approaches are.

INTRODUCTION

According to the IAEA regulations [1] Type B(U) packages for the transport of radioactive material have to withstand accident transport conditions impact loading resulting from a 9m free drop onto an unyielding target in sequence with a 1m puncture drop test in a most damaging attitude. In this paper some aspects of computational methods in combination with experimental investigations will be discussed on the basis of BAM experience in the approval assessment of transport packages.

Concerning the 1m puncture drop test the development of the dynamic finite element (FE) model, the verification with results from the test with a half-scale HLW cask conducted at the BAM test facility, and the steps required for the transformation of this numerical model to apply for the prototype cask will be briefly explained.

Concerning the 9m drop test, a simplified method for consideration the dynamic behaviour of the package during a 9m side drop will be compared with the solution of a dynamic calculation and the advantages and disadvantages of each method will be described.

DYNAMIC CALCULATION OF A 1 METER PUNCTURE DROP TEST

Among other tests a 1m puncture drop test onto a steel bar has to be considered for accident transport conditions of Type B(U) packages according to IAEA regulations [1]. Compared to a 9m drop test onto a flat unyielding surface the interaction between package and target is largely localized. The impact interaction between the bar and cask components (e.g. cask body) leads to high plastic deformations with high strain rates concentrated in the local contact zone. An adequate description of such problems with a static approach is not possible. On the other hand the local character of the interaction makes a dynamic FE model with a necessary mesh density in the area of interest feasible.

The transport and storage packages for radioactive materials from the German manufacturer GNS have often a cask body made of ductile cast iron. The puncture bar has to be made of ”mild” steel according to the IAEA requirements. The behavior of both materials has been extensively investigated by static and dynamic tension and compression tests carried out by the applicant GNS. Suitable material models for metal plasticity exist in several finite element codes (e.g. ABAQUS/Explicit, LS-DYNA). This is an additional reason why the 1m puncture bar drop test is well suited for the application of the dynamic finite element method.

In an approval procedure for the new design of a German HLW cask (GNS-CASTOR HAW28M) BAM conducted among others a 1m drop test onto a puncture bar with a half-scale model of the cask. The package was equipped with several strain gauges and accelerometers at points of interest. Some finite element calculations were additionally carried out by BAM for independent assessment and to control the safety analysis presented by the applicant. The main steps of development and verification of the finite element model will be summarized using the example of the puncture bar impact onto the centre of the cask body in the following.

The correct estimation of the puncture bar deformation is a first indication of the quality of the finite element model that has to be developed. Here, a simplified FE model is generated. The cask is modeled by a rigid body. The material behavior of the puncture bar has to be modeled as realistic as possible and should be verified in this first step. Its elastic behavior is described by Young’s modulus and Poisson’s ratio, plastic behavior by strain rate dependent strain-stress-curves. The density of rigid body segment is modified in order to match the true mass of the package. In addition to the puncture bar’s material law, the friction coefficient between cask and bar has to be investigated in detail. The remaining height of the bar after drop test is nearly independent of the chosen friction coefficient. Important is the comparison of the diameters at the contact surface. The higher the friction coefficient, the higher is the effect of “barrel-shaping”. The friction coefficient, that causes the highest similarity between calculated and real bar shape after drop test is chosen for the following analysis.

In the next step, a more detailed finite element model with a more detailed material definition for the cask body is developed. The comparison of measured and calculated accelerations and strains is the most important proof for verification, in addition to the control of global characteristics as a reaction force and deformations of puncture bar. In the case of the 1m puncture bar drop test, especially the maximum values detected from strain gauges in the area of impact have to be considered. The results of measured accelerations and strains all over the package are important for an assessment of the impact process. Figure 1 shows the comparison of measured and calculated strains at the inner surface of the cask, opposite of the contact area. To extract the calculated strain at the surface, truss elements have been joined to the nodes of element surface segments. The strain curves resulting from both integration points of solid and truss elements show a good correlation with the measured strain curve.

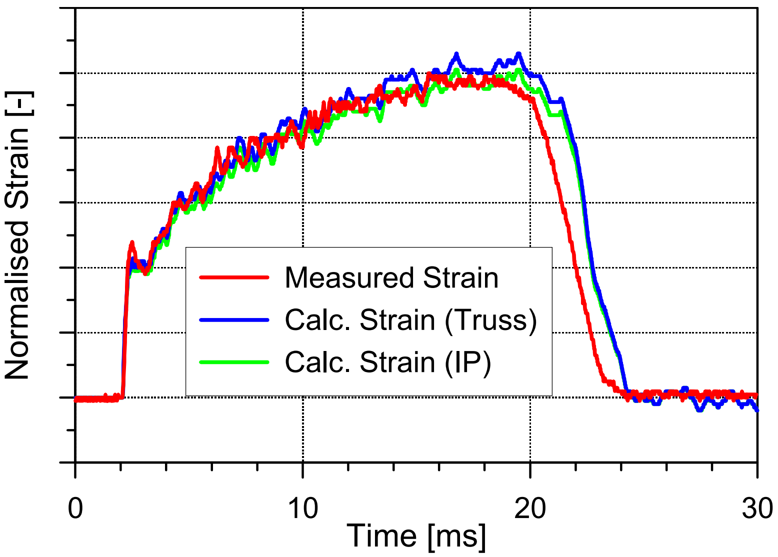


Fig. 1: Comparison of measured and calculated strains (cask body inside, opposite to the puncture bar impact, GNS-CASTOR HAW28M/TB2 drop test horizontally onto a steel bar-75 mm diameter [2])

If the model is by that proof considered to be suitable, it can be adapted to the original full scale package by changing all node coordinates in the input file depending on the scaling factor. At first initial and boundary conditions are not changed. Applied material laws and densities are identical to the scaled model according to similarity laws. That means that strains and stresses should be constant compared to the results of scaled model, time is doubled and forces should be four times greater if scaling factor is 1:2.

Finally the full scale finite element model has to be modified for application in the safety assessment of the original package design. For this purpose the foundation is to be modeled as a rigid body. Material models are adapted to properties according to the material specifications. Calculations are carried out using material properties at lower and upper operational temperatures. The model can also be used to investigate package performance under other safety relevant conditions, which cannot be covered within experimental drop tests, e.g. variation of cask body impact onto the puncture steel bar.

The calculation results can then be applied for assessment of stresses. This includes stresses in parts of the package where measurements during experimental investigations are impossible, or for fracture mechanics investigations with consideration of an artificial flaw in the cask body.

DYNAMIC AND QUASI-STATIC APPROACHES FOR 9 METER HORIZONTAL DROP TEST

General Remarks

Impact limiters are attached to the package in order to reduce forces applied on the cask components due to hypothetical mechanical accident conditions to be evaluated by drop testing. Impact limiters consist typically of an inner metallic structure, an outer metallic casing and the impact energy absorbing material in-between. For most German package designs wood is the major energy absorber.

Due to the complexity of the interaction between cask body, impact limiters and unyielding target the dynamic simulation of a 9m drop of a package with impact limiters is often much more complicated than the simulation of a 1m drop of the cask body onto the puncture steel bar. In particular, an appropriate description of the main energy absorbing material has to be implemented into the finite element model. The necessity of adequate finite element mesh densities of the impact limiter components as well as of the most loaded regions in the cask can lead to a large dimension model and therefore to some additional difficulties in verification procedure and in further application of the model in the safety analysis.

Taking into consideration that the impact limiters absorb the major part of the kinetic energy (as they are relatively soft compared to the cask) and significantly reduce the intensity of the impulsive loading on the cask, a different method can be applied for an approximate evaluation of the drop process in this case. This method includes a combination of simplified numerical tools together with a quasi-static finite element analysis [3]. In a first step the maximum impact force and rigid body deceleration-time history of the cask during the impact process can be calculated with simplified numerical tools [4]. In a second step this rigid body deceleration can then be applied on a verified static numerical model. Dynamic effects, which can not be covered by the static numerical analysis, have therefore to be considered by using an additional dynamic factor.

Simplified numerical calculation of the rigid body deceleration

Simplified numerical tools such as ImpactCalc [4, 5] are able to calculate the history of rigid body deceleration and shock absorber deformation by modelling it as one dimensional mass – spring – system.

The spring is loaded by the cask mass under the initial velocity resulting from a 9m free drop. The impact limiter compression force is modelled as nonlinear spring force [6]. The current stress in each wooden block is thereby deduced as a function of global impact limiter deformation by using stress strain curves from small scale wood compression tests. Typical stress strain relationship from a small scale compression test can be seen in Figure 2.

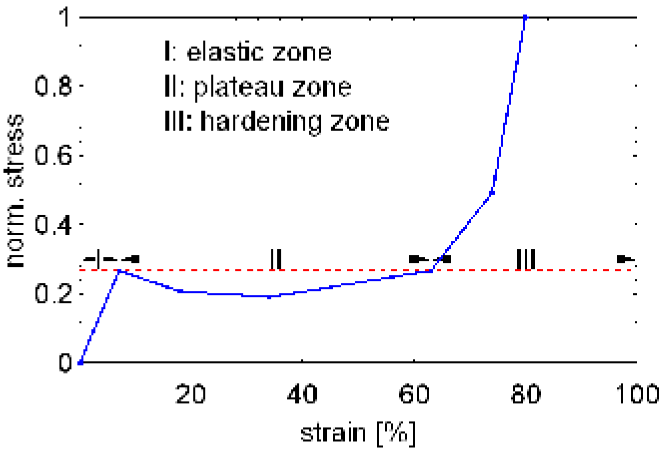


Fig. 2: Yield curve of wood

The impact limiter compression force $F(x)$ is then calculated by multiplying the average stress in the wooden blocks $\overline{\sigma}(x,y)$ with the current contact cross section of the deformed impact limiter with the impact target $A(x)$:

$$F(x) = \frac{A(x)}{y_{max}(x)} \int_0^{y_{max}(x)} \sigma(x,y)\; dy = A(x)\overline{\sigma}(x) \qquad \text{(Eq. 1)}$$

The equation of motion for the cask mass M in form of

$$M\ddot{x} + F(x) = 0 \qquad \text{(Eq. 2)}$$

with following initial conditions $t = 0 \quad x = 0; \quad \dot{x} = v_0$ (where v_0 is the impact velocity of the 9m free fall of approx. 13.3 m/s) can then be solved. Calculation results are deformation over time: $x(t)$ and resistance (impact) force over time: $F(t)$. From the impact force the rigid body deceleration can be derived by:

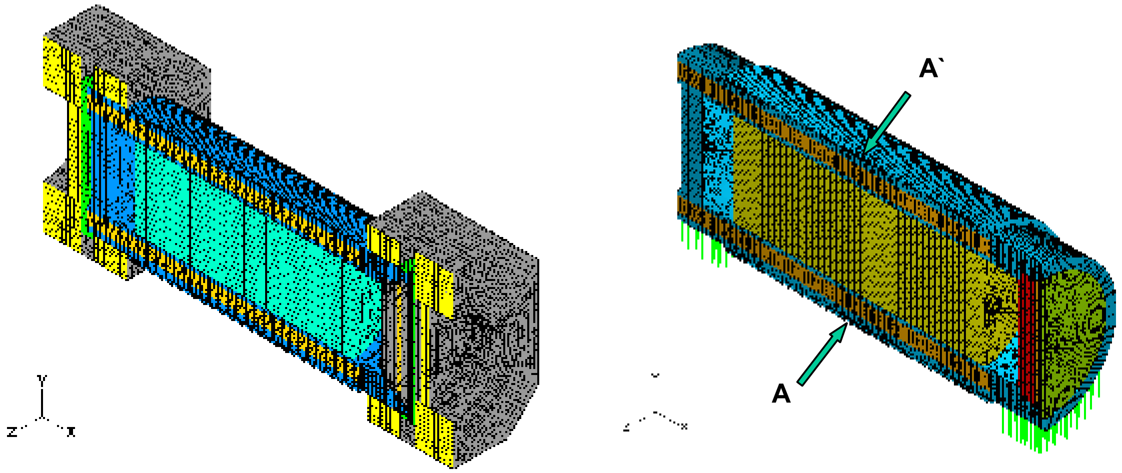
$$G(t) = F(t)/M \qquad \text{(Eq. 3)}$$

The maximum rigid body deceleration $G_{\max}$ can then be used in “quasi-static” FE-calculations for stress and strain analyses of the cask components. In some cases this value has to be increased by an additional factor discussed below to cover the dynamic effects in the response of components under consideration to impulsive loading. A more detailed description of the simplified numerical tool is available at [5].

Quasi-Static Numerical Analysis

The finite element model of the reference cask used for the quasi-static calculation (Figure 3b) consists of cask body, primary and secondary lid, lid bolts and content. Using the available load and geometry symetry, only one half of the cask was modeled. Elastic material properties are considered for some components. For the cask body (ductile cast iron) the elastic-plastic material law is used. The calculations were carried out with the commercial finite element code ABAQUS (Version 6.5) [7].

In the quasi-static model (Figure 3b) the load consists of a gravitational field according to the maximum rigid body deceleration calculated with simplified numerical tools. The impact limiter in the static model was simplified as nonlinear spring with the force-deformation-curve derived from tests with characteristic wood specimens (Figure 2). By using this static finite element model the stresses in the cask body were determined.



a) Model for dynamic calculation

b) Model for quasi-static calculation

Fig. 3: Reference package model for dynamic and quasi-static calculation

Comparison with Dynamic Calculation

The finite element model of the package for the dynamic calculation is presented in Figure 3a. In comparison to the quasi-static finite element analysis the impact limiters have to be included directly in the dynamic model.

The target was modeled as a rigid surface. The impact velocity resulting from 9m drop test is applied on the cask and impact limiters as an initial velocity. All free surfaces (between cask body and content, cask body and impact limiters as well as between impact limiters and target) were defined as contact without friction.

Simplified finite element models of the octahedron impact limiters were used for dynamic calculation. Both impact limiters include an inner plate made of steel, a 5 mm thick steel sheet housing and wood filling of the cavities of the steel housing. The inner steel plate is connected to the cask body by steel bolts. For the purpose of comparison with analytical method, the steel housing and fastening bolts of the impact limiters were not included in the finite element model. The inner plate and cask body is directly connected with "tied" contact.

The characteristic yield curve of wood is shown in Figure 2. It can be divided into three zones: elastic zone, plateau zone and hardening zone. Since load increases drastically in the hardening zone, impact limiter should not reach the hardening zone in compression.

This characteristic yield curve for wood was considered in the material definition of impact limiters.

Comparison of Stresses

A comparison of stresses in the middle of the cask body impacted in a horizontal 9m drop test, derived by dynamic and static FE-calculations is shown in Figure 4. The maximum stresses in the cask body calculated by a static numerical model at points A and A` (see Fig. 3b) are – as expected – smaller as the results from dynamic calculation.

That means the dynamic peak impact of the cask body is about 30% larger than the value calculated by quasi-static approach. Therefore an additional factor has to be considered in order to take into account dynamic effects. By applying a suitable factor on the maximum deceleration it is possible to calculate the stresses by using a quasi-static finite element model for mechanical assessment [3].

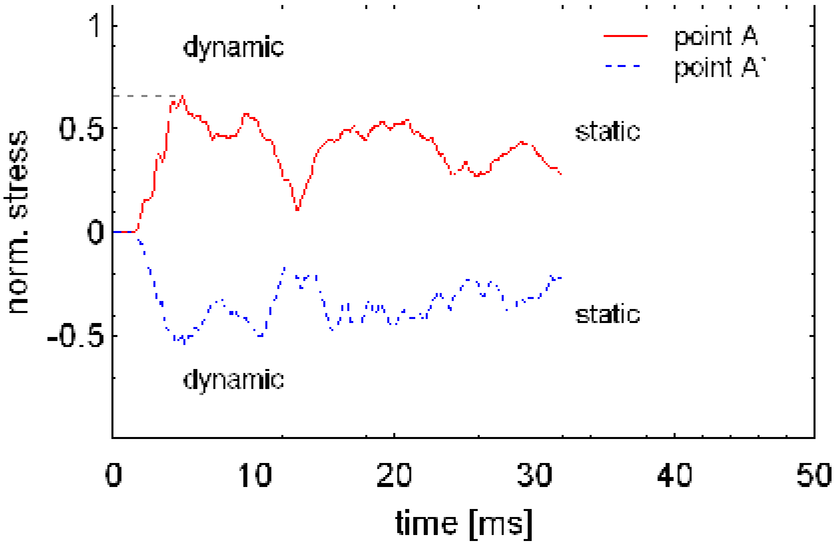


Fig. 4: Normalised stresses at the middle of reference cask body
(at points A, A'-Fig. 3b; 9m horizontal drop test)

CONCLUSIONS

Different numerical methods for structural analysis applied in the approval procedure carried out recently by BAM are presented in this paper. The main characteristics of the problem under investigation such as relative hardness of collision partners, stress level, strain rate, and dimension of the highly loaded package areas in connection with material properties as well as safety margins to the critical load states have to be taken into account in deciding an acceptance of either a dynamic or a simplified quasi-static calculation. The quantity and quality of the experimental information available for the verification of the numerical model is the other important point in this decision. Generally the degree of required experimental verification of an approach depends on the complexity of the question being considered.

It was shown for the case of 1m puncture drop test, that a reliably dynamic calculation is possible by using the experimental small-scale model drop test results for the verification of the numerical model. The developed numerical finite element model can be used for the mechanical assessment during the approval process. The investigation of package performance under conditions, which cannot be covered by experimental drop tests, can be carried out using the verified dynamic numerical model. This includes variations of similar drop impacts, fracture mechanic evaluations or changes in environment conditions, such as temperature. Getting calculation results for assessment package integrity under consideration of dynamic effects without any other simplified assumptions, is an advantage of this method. The effort to develop such a dynamic numerical model is rather large and has a high liability to errors, which can only be avoided by using experimental investigations for verification. Otherwise the local character of the interaction is favourable to a practical realisation of the dynamic approach in this case (e.g. for an appropriate mesh density in the high loaded region).

The dynamic simulation of 9m side drop test is due to the complexity of the interaction between collision partners (cask body, impact limiters, unyielding target) rather difficult. However the dynamic finite element analysis has certain advantages in comparison with the combined simplified numerical and quasi-static FE-approach. Different phenomena (like load transfer, impact limiter weak points, influence of the impact limiter hardening, friction, etc.) which are not ascertainable with the simplified method can be examined and explained satisfactorily. Generally, the accuracy of the dynamic FE analysis depends on the detail of the modeling, the selection of material definitions, and the simulation of all relevant conditions of the impact. Dynamic calculations require an extensive verification with experimental results otherwise a reliable calculation and therefore a reliable assessment of the stresses is not possible. The necessity of adequate finite element mesh densities of the impact limiter components as well as of the most loaded regions in the cask can lead to a large dimension model and therefore to some additional difficulties in the practical realisation of this approach.

A different method can be applied for considering the dynamic behaviour of the cask in a 9m side drop test. A simplified numerical tool can be used to calculate the deceleration-time history of the package in a first step. In a second step dynamic effects have to be accounted for by applying a dynamic factor on the gravitational field in the following quasi-static numerical calculation. At least the deceleration-time function calculated by the first step and used for the assessment of the dynamic factor has to be verified appropriately by experimental results. By using the quasi-static numerical model with consideration of the dynamic factor parameter studies can be carried out for investigation of the influence of different conditions such as mesh sensitivity, load application, material characteristics, temperature etc. Following detailed investigations like fracture mechanic evaluations are then possible too. The use of the quasi-static numerical analyses for assessment of the dynamic behaviour of the package is now state-of-the-art analyses. But in the future a lot of work is to be dedicated to the development of dynamic numerical models for complex impact limiter structures and the interaction between the different parts of the package and the unyielding target. In particular, a lot of experimental work is required in order to estimate the real behaviour of the absorber material and to perform an adequate implementation of this in the finite element code by means of a suitable material law of the energy absorbing material.

REFERENCE

[1] IAEA - International Atomic Energy Agency (Ed.): Regulations for the Safe Transport of Radioactive Material, 2005 Edition (Revised). Safety Standard Series No. 113, Vienna: IAEA - International Atomic Energy Agency, 2005.

[2] A. VOßNACKE, R. LIEDTKE, R. NÖRING: CASTOR® HAW28M-Fallprüfungen zur Absicherung des VR-Zulassungsverfahrens, Jahrestagung Kerntechnik 2006, Aachen, Germany

[3] L. QIAO, M. NEUMANN, S. KOMANN, V. BALLHEIMER: Dynamic and quasi-static FE calculation of impact from 9 meter drop of a spent fuel transport cask, Ramtransport09, Radioactive Materials Transport Conference 2009, 13.-14- May 2009, Manchester, UK

[4] M. NEUMANN, F. WILLE, V. BALLHEIMER: Approximation of package deceleration and shock absorber deformation under 9m drop test conditions using simplified numerical tool ImpactCalc, In: Packaging, Transport, Storage and Security of Radioactive Material, Volume 19, Number 1, March 2008, S. 35-40

[5] M. NEUMANN: Untersuchung des Verhaltens stoßdämpfender Bauteile von Transportbehältern für radioaktive Stoffe in Bauartprüfung und Risikoanalyse. BAM-Dissertationsreihe, Band 45, Berlin, 2009 (PhD-Thesis)

[6] M. NEUMANN, F. WILLE, L. QIAO, V. BALLHEIMER, S. KOMANN: Simulation of impact limiter crushing of RAM packages under 9m drop test conditions, Paper ID no. 10090, Waste Management Conference 2010, March 7-11, 2010, Phoenix, Arizona, USA

[7] ABAQUS Analysis User's Manual, Version 6.5, ABAQUS Inc. 2004