

MODELING STRATEGIES FOR DYNAMIC FINITE ELEMENT CASK ANALYSES

Uwe Zencker

Günter Wieser

Linan Qiao

Christian Protz

BAM Federal Institute for Materials Research and Testing
12200 Berlin, Germany

ABSTRACT

The safety of transport packages may be demonstrated by numerical calculation of load scenarios defined in the IAEA regulations. Possible handling accidents of casks at interim storage sites or in a final repository are typically analyzed by dynamic finite element computations. In each case the investigated load scenario must be transferred into a mathematical model. Secondly the mathematical model must be transferred into a numerical model. Reliable finite element models should be developed by assembling verified sub-models of components. The finite element mesh, material modeling, initial and boundary conditions, contact definitions, and time integration as well as the benefit of pre- and post-calculations are discussed. The paper presents lessons learnt from modeling dynamic test scenarios for finite element analyses over the years.

INTRODUCTION

The mechanical requirements of casks for radioactive materials are commonly defined by conservative load scenarios. The proof of safety of transport and storage casks may be conducted by means of experimental prototype testing or numerical analysis of the cask in the given load scenario. Additionally analogies to comparable load scenarios can be used. An analysis of cask models with reduced scales accompanied by transferability relations is also possible. The whole load scenario must be scaled suitably in this case. The investigation of load scenarios is performed more and more with numerical simulations instead of experimental testing especially in case of an only modified but not completely new cask design or when the load scenario cannot be easily reproduced by an appropriate experimental scenario. Besides, test results of scaled model casks can be transformed numerically to original size casks. Not all parts of a cask can be accessed directly to measure local stresses or strains (e.g. moderator holes in a cask wall). In addition, effects from altered boundary conditions (e.g. temperature) can be comparatively simply examined in numerical calculations. However, such calculations need reliable finite element (FE) models and especially material models. An adequate material characterization must be implemented in the FE code used, if necessary. Suitable material parameters for time-dependent scenarios are often not found in literature and must be identified in laboratory tests. Pre-calculations with possibly simplified models are sometimes meaningful for the clarification of the initial or boundary conditions.

Quite often the accuracy of the calculation results is not exactly known. But knowledge about the accuracy of the numerical procedure is necessary to be able to define and apply safety factors. When cask tests are planned, pre-calculations of the test scenario should be carried out. Then, minimum safety factors can be derived from a comparison of numerical and test results. Probably effects will be found which were not considered in the pre-calculations because a real test is normally more complex than assumed. Analysis and modeling of these additional effects are part of the post-calculations. With a better knowledge of the physical behavior and relationships, the safety factors can be possibly reduced for the investigated scenario.

Frequently experimental results from neither component tests nor tests with original size casks are available. Then, safety factors must be derived from similar analyses. Maybe there are technical standards defining minimum safety factors. Such standards are regularly applicable to normal but not accident loading conditions. The use of safety factors for normal conditions under accident conditions would typically lead to a very conservative cask design.

Calculation results are also influenced by various numerical effects. The meshing of the cask and all other parts of the load scenario with finite elements and particularly the modeling of physical contacts between individual components or numerical contacts between global model and sub-models should be mentioned here.

In general, investigated load scenarios are very complex because of the quantity of components and their interactions. Ideally only a reduced number of components (two components as a rule) including their mechanical and thermal interactions are analyzed as a start. Correctness and plausibility of sub-models and calculation results for simpler load cases are assessed on this basis. Parameter studies might be helpful at this step. The comparison of calculation results with expected results, e.g. from analytical calculations, tests, or literature, determines the design of sub-models, their interactions and parameters. Finally the whole model is assembled from the sub-models.

BAM (Federal Institute for Materials Research and Testing) has published guidelines for preparation, calculation and evaluation of finite element computations [1] in which procedures for reliable dynamic numerical calculations, their documentation, and assessment of safety analysis reports are described.

REQUIREMENTS

BAM assesses the design of casks for transport as well as storage of radioactive materials. The assessment procedures for transport casks are based on recommendations of the IAEA (International Atomic Energy Agency) [2] and their appropriate conversion into national and international (e.g. European) regulations. Requirements for casks for interim storage are fixed in technical acceptance criteria of the individual storage site on the basis of guidelines for the storage of radioactive waste with negligible heat generation [3] and for dry cask storage of spent fuel and heat-generating waste [4]. Casks for final disposal are certified for storage at the German Konrad site according to waste acceptance requirements [5, 6]. Compliance of a cask with appropriate national and international regulations must be demonstrated.

Accident conditions are given by idealized load cases (e.g. the drop of a Type B(U) package from 9 m height onto an essentially unyielding target, or the drop of a storage container from 5 m height onto the hard rock ground of the storage facility). The possible load scenarios for interim storage of a cask are analyzed individually for each site. Typically drop or crash scenarios are derived. Hence, conservative load scenarios without additional safety factors or representative load scenarios with appropriate safety factors are considered.

In reality there are concurrent initial and boundary conditions, i.e. it does not have to be obvious which combination of initial and boundary conditions leads to the most damaging scenario. It is generally expected that a conservative hard and low energy absorbing impact limiter causes

highest stress and also maximum damage in the cask body. However, during a slap-down drop test with softer shock absorber a trunnion could hit the target with even higher local stresses which perhaps would not happen with a harder shock absorber for the same impact angle, cf. Figure 1. As another example, welding seams could be loaded higher despite the softer shock absorber. The damage of a trunnion or failure of a welding seam could be safety critical events. Similar questions arise for the temperature of the load scenario. Critical temperatures of the individual components might be different. Minimum temperature causes conservatively high stress in the cask body, but concerning leak tightness the conservative maximum plastic deformation of the lid system would occur at maximum temperature. As a result, material parameters chosen conservatively do not guarantee that the load scenario altogether is conservative.

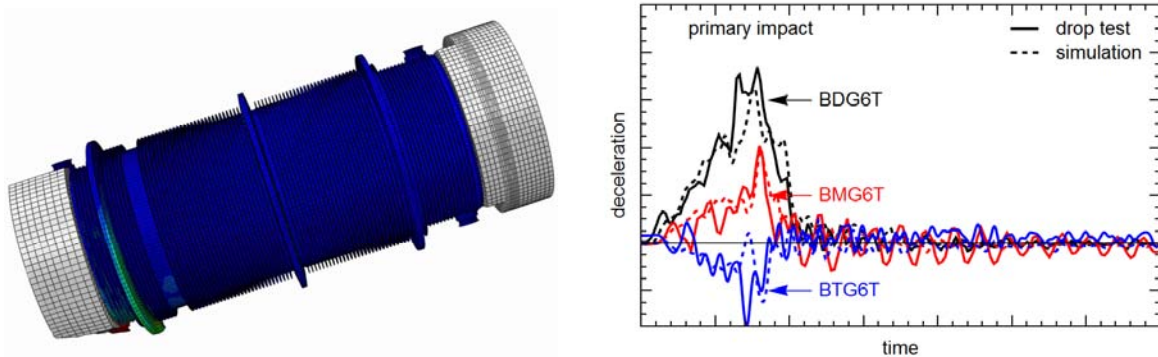


Figure 1. Slap-down drop test with an impact angle of 20° [7]

Conservative boundary conditions should also be chosen physically meaningful and realistic. Conservative but non-realistic boundary conditions could result in erroneous cask loading. As a rule, the boundary conditions are chosen so that the cask load is maximized. However, impact duration could be so much shortened in dynamic load scenarios by conservative conditions, that the load type could change, what is not intended.

A cask drop onto a mathematically unyielding foundation is regarded as an example for a drop onto a conservatively hard target. Through this, the impact duration is shortened in combination with an increase of the stress or strain resp. in the cask, cf. Figure 2. While for a real (or yielding) target a tension stress state at inner side of cask bottom is dominating for long time, the load type changes for the unyielding target between pressure and tension. Hence, a representative load case together with an appropriate safety factor can potentially result in a more realistic cask load.

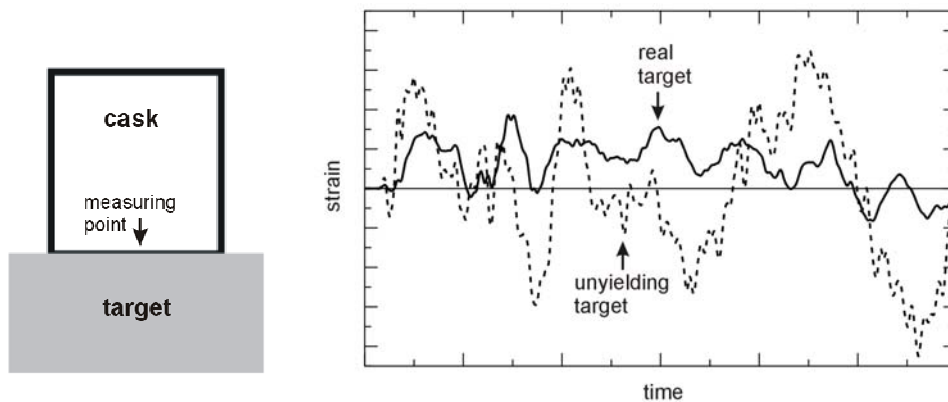


Figure 2. Strain at inner side of cask bottom for 5 m drop onto an unyielding or real target

RELIABLE FINITE ELEMENT MODELS

Reliable FE models are essential for numerical simulations. The individual components of the load scenario need a partial verification. At first, the components have to be defined, e.g. cask body, basket, fuel assemblies, lids, impact limiters, puncture bar, foundation. Basis for a reliable FE model is the successive assembling of the verified sub-models for the individual components including their interactions and parameters.

In each case the investigated load scenario must be transferred into a mathematical model with specification of geometrical and physical properties:

- Idealization of drop boundary conditions and target foundations
- Selection and idealization of all relevant cask components
- Adequate material formulations
- Adequate contact definitions
- Specification of loads (e.g. seal force, bolt preloads)
- Applicable symmetry conditions

Secondly the mathematical model must be transferred into a numerical model concerning among others:

- Discretization
 - Element types (geometry and interpolation approach)
 - Size and shape of elements (mesh)
 - Loads (single loads, distributed loads, ...)
- Solver
 - Solver type
 - Timestep
 - Solver control parameters
- Output
 - Variables
 - Frequency

FINITE ELEMENT MESH

The transfer of the mathematical model into a finite element model requires the meshing with finite elements. For this, the element geometry (volume, plate, shell, beam, etc.) and interpolation approach (first order, second order) must be fixed. Elements with reduced integration (with only one integration point in the middle of an element with linear interpolation) are typical for dynamic simulations. They are sufficiently exact for the calculation of the propagation of stress waves and can be calculated efficiently for large models. Element formulations which suppress zero-energy modes by so-called hourglass control (e.g. by addition of artificial strain energy) should be preferred.

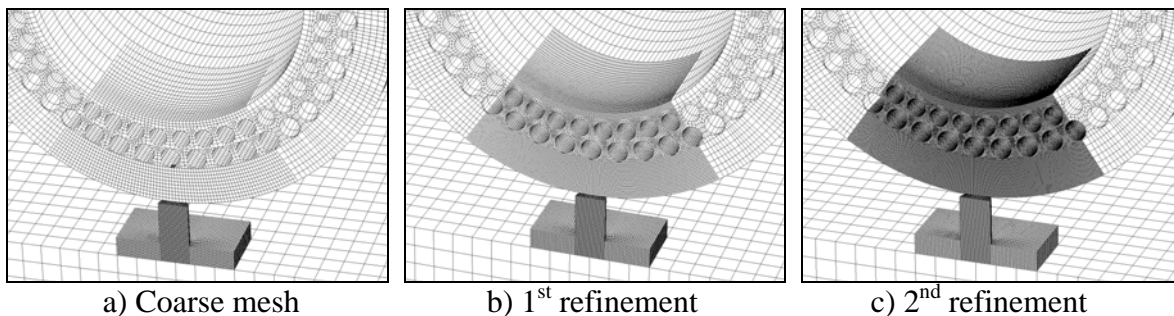


Figure 3. Finite element mesh with refined sub-model [8]

The finite element theory states that the numerical approximation of a body by finite elements converges to the correct analytical solution with mesh refinement in the absence of geometrical or physical singularities. Contrary, a mesh refinement leads to an increase of needed main memory and computation time non-proportionally with the number of nodes. Unfortunately, there is no generally valid rule for construction of a finite element mesh that ensures a numerically stable sufficiently accurate dynamic finite element computation.

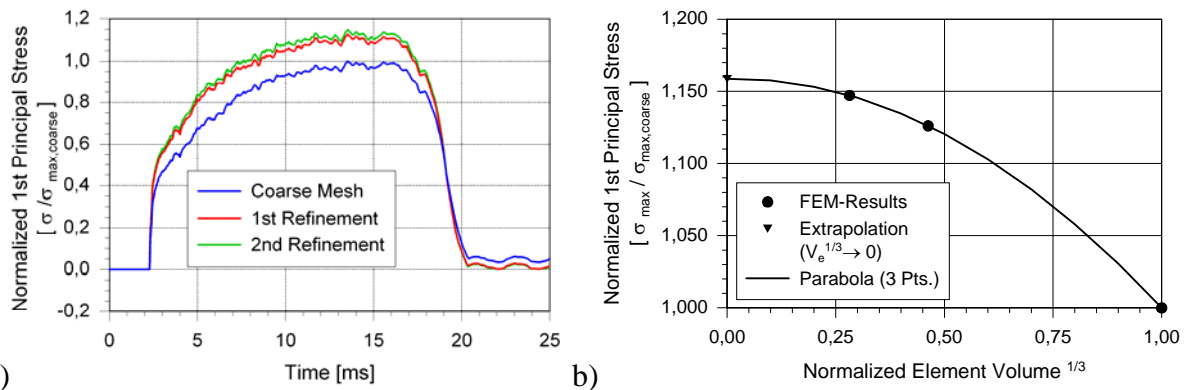


Figure 4. History (a) and convergence (b) of normalized first principal stress at borehole [8]

The question, how fine a FE mesh should be, was examined at the example of the 1 m puncture drop test of a cylindrical cask with center of the cask wall onto a steel puncture bar [8]. The cask contained two rows of bore holes for moderator material located near the cavity. Figure 3 illustrates the step-by-step mesh refinement at the impact zone. A coarse mesh was created first, which then was refined in two steps. The element size was bisected in each step. The puncture bar was not refined. A remarkable influence of the element size on the calculation result was found. Figure 4a shows the first principal stress at a borehole. The first refinement step leads to a noticeable stress increase. Further stress increase at the second refinement step is smaller. The solution is numerically stable as it approaches a limit curve. Similar effects can be found also for von Mises effective stress. Strains at body surface for model validation based on experimental results can be found by extrapolation towards infinitely small elements, cf. Figure 4b.

A strong mesh refinement is recommended only in areas of special interest because numerical effort increases. On the other hand, the mesh must not be too coarse in dynamic calculations because the mesh has the effect of a numerical low-pass filter for the stress waves. The cut-off-frequency of this filter decreases with increasing element size. High-frequency stress waves will not be transmitted correctly and stress peaks smear. The refined mesh was constructed by incorporation of a sub-model into the global model using tied contact conditions. At such inner contact surfaces partial transmission and reflection of stress waves is possible because differently fine meshes have different transmission behaviors.

TIME INTEGRATION

The time integration can be implicit or explicit. An explicit time integration procedure is only conditionally stable, that is, the maximum timestep is limited. A finer mesh with smaller finite elements results in a reduction of the maximum timestep and in an increase of the total number of timesteps needed to solve the numerical problem. The naturally limited small explicit timestep ensures that the numerical solution is not only stable but also sufficiently accurate. For implicit time integration also unconditionally stable time integration procedures can be constructed (e.g. Euler backward), which allow considerably bigger timesteps than explicit procedures. However, the accuracy of the solution reduces with extension of the implicit timestep, and the computation

of an implicit timestep is numerically more effortful than an explicit timestep due to the necessary inversion of the stiffness matrix. Therefore, for implicit time integration a compromise must be found between efficiency (timestep as big as possible) and accuracy of the solution. In practice implicit procedures are often not suitable for dynamic calculation of large models (particularly in connection with many numerical contacts).

MATERIAL MODELING

The relevant mechanical, thermal and chemical properties of the materials used must be known for a reliable FE analysis of a load scenario. These properties must be translated into a FE material model with material parameters. It is not always possible to find suitable material parameters in literature. In the context of dynamic calculations this concerns primarily strain-rate dependent stress-strain curves (if necessary also dependent upon the temperature). If material data are found in literature, it must be clarified under which conditions they were measured (whether e.g. the appropriate technical standards have been met). If non-standard materials are needed, the material properties (yield stress, ultimate strength, fracture toughness, etc.) must be measured independently in laboratory tests. The simulation of the laboratory test itself is sometimes necessary especially in case of dynamic investigations to separate specimen behavior and influences from the test equipment. For example, standard testing machines usually begin to vibrate at dynamic tensile tests with strain-rates above some 100/s and generate oscillations in measured stress-strain curves independently of the specimen behavior. In such case, other test equipment suitable for highly dynamic tests should be used. Otherwise, the material behavior of the specimen can be separated from non-perfect measurement results by simulation of the test.

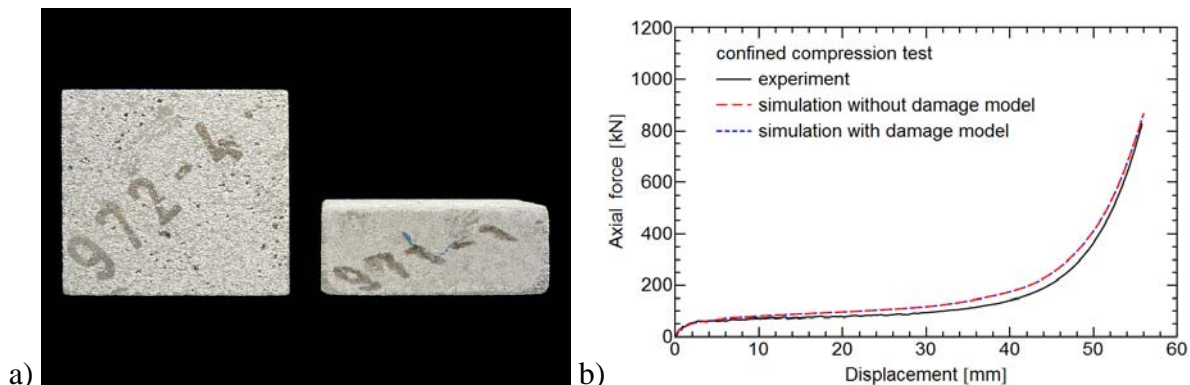


Figure 5. Damping concrete specimen before/after test (a) and measured as well as calculated force-displacement curves for constant loading rates (b) [10]

Currently BAM investigates the mechanical behavior of damping materials like polyurethane foam, damping concrete and wood as part of the research project ENREA (Development of numerical methods for analyzing impact limiters subjected to impact or drop scenarios) [9]. Damping concrete is a patented concrete using polystyrene balls as aggregates. Density is 1/3 of standard concrete with essentially lower yield stress. The material behavior is described by crushable foam with isotropic hardening. A damage criterion for ductile damage of the polystyrene balls as well as a shear fracture criterion for failure of the cement matrix can be added. The material model was calibrated by simulation of displacement-controlled compression tests with different temperatures and loading rates [10]. Figure 5a shows a damping concrete specimen before and after test. Measured and calculated force-displacement curves for constant loading rates are given exemplarily in Figure 5b.

INITIAL AND BOUNDARY CONDITIONS, CONTACT DEFINITIONS

Often the most damaging test conditions must be determined in pre-calculations. It is well known that the impact angle as initial condition has important influence on the test results in flat bottom drops of cubic containers. As another example, the influence of boundary conditions is illustrated by means of the contact definition between cask and puncture bar at the puncture bar test [11]. The cask, simplified to a rigid ring, hits a deformable bar meshed with continuum elements, cf. Figure 6. The cask may be reduced to a rigid body because up to 98 % of the potential energy is dissipated in plastic deformation of the puncture bar. This simplification is acceptable if the focus is on the deformation behavior of the puncture bar. Its plastic behavior is represented by strain-rate dependent stress-strain curves directly implemented in the material model. Sliding contact with Coulomb's friction is assumed between body and bar surfaces. The friction coefficient affects strongly the deformation of the bar. Measured and calculated deformations of the bar are compared. The friction coefficient that leads to the correct barrel shaping is fixed as contact condition for investigation of the whole test scenario later on.

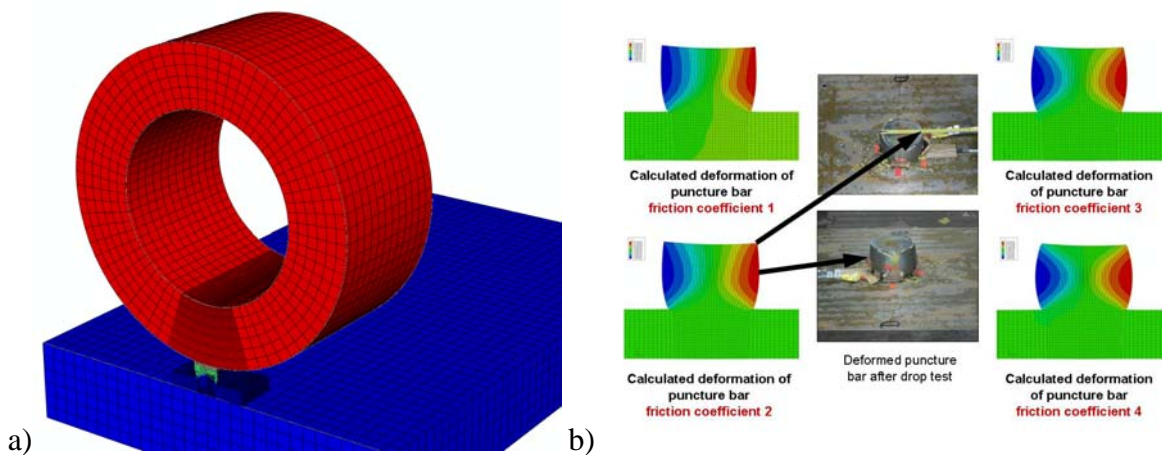


Figure 6. Model of rigid body with deformable puncture bar (a) and comparison of puncture bar deformation (b) [11]

An important test condition is represented by the drop test foundation. An unyielding target according to the IAEA regulations [2] could be modeled as (mathematically unyielding) rigid body or with simple (essentially unyielding) elastic models. A description of the hard but deformable foundation of the German Konrad repository is contained in the waste acceptance requirements [5, 6], but not all parameters are fixed. Therefore, a reference model was developed [12]. The interim storage facility built-in foundation partly consists of damping concrete and steel fiber screed which is currently modeled conservatively within licensing procedures and under ongoing research at BAM [10].

The model of the lid system is important for the safety assessment of the leak tightness of a cask. On the one hand, there are special gasket elements implemented in FE codes which must be completed by constructive and material parameters. On the other hand, the calculated mechanical behavior of the lid system can be correlated with experimental data from mechanically and thermally loaded lid-flange components available at BAM. An exact calculation of deformation and movement of lid and cask body is the decisive issue in this approach. Influence factors are contact modeling (contact type and contact parameters, esp. friction parameters) between lid and cask body, bolt preloads and constructive details like the gap between lid and cask body. These influence factors were recently examined in detail and recommendations for their modeling were given in Ref. [13].

CALCULATIONS

If experimental investigations were carried out, the FE model can be checked by comparison of calculation results with test results of components, scaled model casks, or full-scale casks. Corresponding results for thick-walled cast iron casks were presented in Ref. [12].

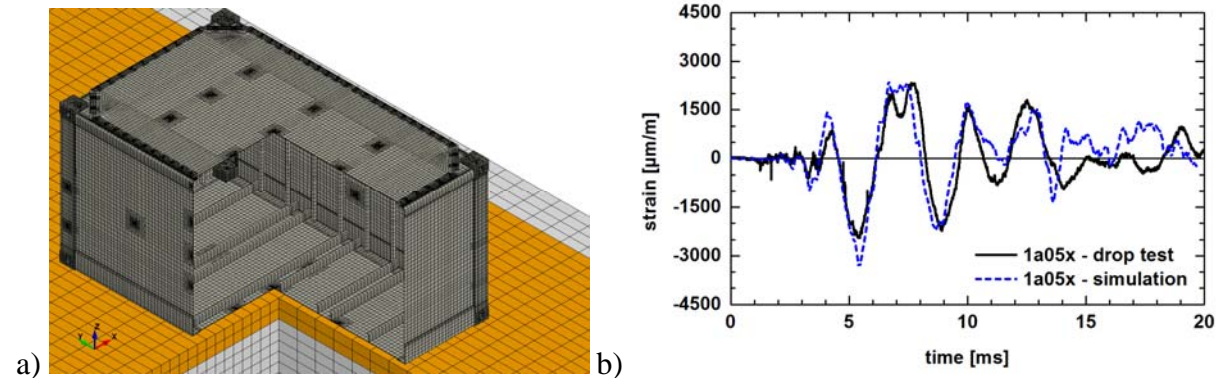


Figure 7. Model of steel sheet container Type V with impact target (a) and measured as well as calculated strains in the middle of the lid in x-direction (b) [14]

Newer investigations were carried out at thin-walled steel sheet containers. The box-shaped container of Konrad Type V [5] is considered as an example [14]. The FE model is shown schematically in Figure 7a. Due to the unsymmetrical construction of the container, the experimental set-up was modeled as full model. The model of the steel sheet container includes all relevant structural parts. Thin-walled components like side walls were modeled with shell elements. Solid elements were used for pillars, corner fittings and lid bolts. The welding seams were simplified to tied contacts. The impact target consists of a steel plate and a concrete block beneath. The foundation was confined by non-reflecting boundaries, while no further boundary conditions had been necessary. As initial condition, the container was placed close above the impact target, having imposed a velocity corresponding to the actual drop height. The mechanical behavior of the container was described with an elastic-plastic material model, whereas the impact target was assumed to be elastic.

In order to obtain comparable numerical results, the simulation of the drop test scenario was carried out with adjusted container orientations taking into account the slight impact angles derived from the measured deceleration signals. Figure 7b shows both the measured and the calculated strains in the middle of the lid in x-direction (cf. Figure 7a) during the 5 m drop. The strain history has been approximated sufficiently exactly in the considered measurement point. However, evaluation of single signals does not provide an adequate overview of the impact scenario and the following free wall-bending vibrations. Therefore calculation results have to be assessed at different positions of the container, particularly at highly stressed areas in the structure. Altogether the comparison of the numerical calculations with the test results showed that the developed FE model is suitable to describe the mechanical behavior of a box-shaped steel sheet container during a flat bottom drop test.

CONCLUSIONS

The examined load scenarios often seem to be simple. However, their numerical modeling is frequently effortful. Primarily the material behavior and the boundary conditions are often not exactly known. Therefore, in practice the application of regulations requires interpretations, additional pre-calculations or experimental investigation of material behavior.

Even simple load scenarios could require very fine meshes with many elements. Complex load scenarios can be simplified by sub-models. There are no general rules about the number of elements needed over a given wall thickness of a cask or the element size. Finally, the practical experience of the engineer is of decisive importance.

A reliable modeling strategy for dynamic cask analyses is the identification of components, the creation and verification of FE sub-models for the individual components and then the assembly of the verified sub-models to the whole model.

Component tests or cask tests are recommended for new components or cask designs. This allows a check whether all experimentally found effects are contained in the simulation.

ACKNOWLEDGMENTS

The authors wish to acknowledge the financial support of research projects ConDrop by BAM and ENREA by the German Federal Ministry of Education and Research under contract no. 02S8588.

REFERENCES

- [1] BAM: Guidelines for numerical safety assessments within the scope of design testing of transport and storage casks for radioactive materials (in German), BAM-GGR 008, Rev. 0, Berlin, 2003
- [2] International Atomic Energy Agency (IAEA): Regulations for the safe transport of radioactive material, 2012 Edition, Specific Safety Requirements No. SSR-6, Vienna, 2012
- [3] Nuclear Waste Management Commission (ESK): Guidelines for the storage of radioactive waste with negligible heat generation (revised version of June 10, 2013) (in German), Recommendation, Salzgitter, 2013
- [4] Nuclear Waste Management Commission (ESK): Guidelines for dry cask storage of spent fuel and heat-generating waste (revised version of June 10, 2013) (in German), Recommendation, Salzgitter, 2013
- [5] Brennecke, P. (Ed.): Requirements on radioactive waste for disposal (Waste Acceptance Requirements as of October 2010) - Konrad Repository (in German), Report SE-IB-29/08-REV-1, Federal Office for Radiation Protection, Salzgitter, 2011
- [6] Steyer, S. (Ed.): Radioactive waste product control, Radiological aspects – Konrad Repository – as of October 2010 (in German), Report SE-IB-30/08-REV-1, Federal Office for Radiation Protection, Salzgitter, 2011
- [7] Qiao, L., Zencker, U., Musolff, A., Kuschke, C., Komann, S., Völzke, H.: Numerical investigation of a complex drop test scenario, Radioactive Materials Transport and Storage Conference (RAMTRANSPORT 2012), London, May 23-24, 2012
- [8] Zencker, U., Weber, M., Wille, F.: Finite element mesh design of a cylindrical cask under puncture drop test conditions, Packaging, Transport, Storage and Security of Radioactive Material 22 (2011), No. 2, pp. 112-116
- [9] Kasperek, E., Völzke, H., Scheidemann, R., Zencker, U., Wolff, D.: Experimental and numerical studies of shock absorbing materials for containers for radioactive waste, atw 57 (2012), No. 6, pp. 397-401
- [10] Qiao, L., Zencker, U., Kasperek, E. M., Völzke, H., Scheidemann, R.: Development of a finite element model for damping concrete under severe impact loads, Proc. 17th Intern. Symp. on the Packaging and Transportation of Radioactive Materials (PATRAM 2013), Paper #127, San Francisco, August 18-23, 2013
- [11] Weber, M., Wille, F., Ballheimer, V., Musolff, A.: From experiment to appropriate finite element model – safety assessment for ductile cast iron casks demonstrated by means of

- IAEA puncture drop test, Packaging, Transport, Storage and Security of Radioactive Material 22 (2011), No. 3, pp. 148-153
- [12] Zencker, U., Weber, M., Qiao, L., Droste, B., Völzke, H.: Mechanical safety analyses of cast iron containers for the Konrad repository, Proc. 16th Intern. Symp. on the Packaging and Transportation of Radioactive Materials (PATRAM 2010), Paper #220, London, October 3-8, 2010
- [13] Zencker, U., Qiao, L., Kasperek, E. M., Völzke, H.: Modeling of cylindrical casks under horizontal drop test conditions, Proc. Intern. High-Level Radioactive Waste Management Conference 2013 (IHLRWM 2013), Paper #6862, Albuquerque, April 28 - May 2, 2013
- [14] Protz, C., Völzke, H., Ellouz, M., Kasperek, E. M., Zencker, U., Grünwald, H., Gründer, K.-P.: ConDrop – Development of numerical simulation approaches for drop test analyses of steel sheet containers, Proc. Institute of Nuclear Materials Management (INMM) 53rd Annual Meeting, Paper #7939, Orlando, July 15-19, 2012