Proceedings of the 17th International Symposium on the Packaging and Transportation of Radioactive Materials PATRAM 2013 August 18-23, 2013, San Francisco, CA, USA

NUMERICAL ANALYSIS OF CASK ACCIDENT SCENARIOS IN STORAGE FACILITIES

Mike Weber **Uwe Zencker**

Dietmar Wolff **Holger Völzke**

Uwe Herbrich **Sven Nagelschmidt**

BAM Federal Institute for Materials Research and Testing
Unter den Eichen 87
12205 Berlin, Germany

ABSTRACT

Mechanical drop test scenarios for Type B (U) packages according to the IAEA regulations have to be carried out onto the so-called “unyielding target” (usually with cask impact limiters) and onto the puncture bar respectively. They are predefined and do not require any further investigation of scenarios that really could happen on transportation routes. Cask accident scenarios in the framework of approval procedures for interim storage sites are derived from a detailed analysis of the handling procedures necessary from arrival of cask at the site to its storing position. In that case casks are usually handled without impact limiters. Dependent on possible drop heights, drop positions and underground, conservative cask accident scenarios are derived for further safety proofs.

According to the mechanical assessment concept of the considered approval procedure numerical calculations have to be provided by the applicant to demonstrate mechanical cask safety. Stresses and strains in the cask body as well as in the lid system have to be identified and assessed. Using the example of a 3-m-vertical-drop of a transport and storage cask for spent fuel elements onto the floor construction made of damping concrete covered by screed, BAM developed a finite element model. The finite element code ABAQUS/Explicit™ [6] was used. Additional experimental investigations are not provided and therefore parametric studies are necessary to identify the sensitivity of the finite element model to significant parameters and to verify the finite element models according to the requirements of the BAM GGR-008 [3] (Guidelines for Numerical Safety Assessments…).

The paper describes the investigation of modeling the material behaviour and attachment of bottom side cask components. Questions concerning the modeling of crack length limiting reinforcement in the screed layer are discussed. The influence of the mesh density of the screed layer and its strength is considered as well. Finally, the developed finite element model could be used for an independent

safety assessment. It can help to understand the complex mechanisms of the interaction between the cask components and floor construction.

INTRODUCTION

The dry storage of spent fuel from power reactors is currently managed in Germany in dual purpose metal casks designed for both transport and storage. These casks are placed in storage buildings of storage facilities. Interim storage is inevitable until the national decision on final disposal is made and a repository is put into operation. Therefore storage licenses are granted by the competent authority, the Federal Office for Radiation Protection (BfS), based on § 6 of the German Atomic Energy Act (AtG) [1]. Such licenses are currently issued for 40 years beginning with the first cask emplacement. For the safety evaluation of application documents, the BfS contracts technical expert organizations according to § 20 AtG. Amongst others, the Federal Institute for Materials Research and Testing (BAM) is contracted in order to evaluate all cask related safety issues concerning safe enclosure, decay heat removal, subcriticality and shielding. In particular, BAM checks the complete cask design including material specifications, manufacturing procedures, and quality assurance measures for fabrication and operation according to the state of the art in science and technology.

The concept for dry interim storage of spent fuel and heat-generating waste is defined by the German Nuclear Waste Management Commission (ESK) in specific guidelines [2]. Thus, an accident analysis is required e.g. to identify decisive accident scenarios derived from handling procedures of the casks inside the interim storage facilities. This accident analysis has to be presented by the applicant and to be evaluated by a technical expert organization (e.g. TÜV NORD EnSys Hannover), which is contracted by BfS as well. Based on those results, BAM evaluates resulting mechanical and thermal loads for casks and components with regard to the safety goals.

In addition to an accurate assessment of the provided documents and (numerical) calculations, the development of an independent finite element model can be necessary to investigate the sensitivity of a finite element model to modified parameters. This is of particular importance if no results from drop tests are provided for validation [3]. Using the example of one possible accident scenario the development of an independent finite element model of a generic storage cask is described. The influence of the investigated modifications on the calculated loads and their distribution is discussed.

EXEMPLARILY SELECTED ACCIDENT SCENARIO AND SAFETY GOALS

After arriving of the transport vehicle in the unloading area of an exemplary interim storage, the cask has to be attached to the crane, erected and moved to its final storing position after final preparation work (e.g. to complete secondary lid installation). After leaving the vehicle area and before lowering, there is a maximum distance of 3 m between the bottom of the vertically suspended cask and the floor (Fig. 1). If one element of the load chain fails in that position, the cask would drop in vertical position onto the storage floor. A damping concrete in the floor construction, covered by screed, has to minimize the loads in the cask. It has to be shown, that the cask body and lid sealing systems stay intact. The leakage rate must not exceed a specified value after the accident. Minimum and maximum cask temperatures have to be considered. It is important to note that these temperatures are not defined in any regulation as fixed values but derived from the thermal analyses of the cask and the site specific interim storage conditions. Since it is known from previous evaluations of various drop scenarios that the critical loads in the lid and sealing system occur in other accident scenarios, the following investigations are restricted to the loads in the bottom side impact area.

THE FINITE ELEMENT REFERENCE MODEL – R 1

The finite element model generated by BAM (Fig.°2, 8), using the pre-processor ABAQUS/CAE™, represents a generic storage cask with a mass of about 126 Mg. Cask body, the parts of a multi-barrier lid system, bottom closure plate and the appropriate connecting elements are the main parts of the finite element half model. 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 behavior of the cask body, made from ductile cast iron, was described with strain rate dependent true stress versus logarithmic strain curves.

The parts of the multi-barrier lid system, primary and secondary lids including bolts and nuts were modeled very detailed with linear elastic/ideal-plastic material behavior. However, as mentioned above, the loads in these parts and their deformation are neither critical for themselves nor does their behavior influence the loads at the considered bottom side impact area. For this reason and to safe computational time the pre-tension of lid screws was neglected.

It was assumed that the bottom closure plate and its attachment are much more relevant for the loads in the impact area. In a first step the closure plate was modeled in a pure elastic way. The pre-tension in the closure plate screws was generated by definition of a thermal expansion coefficient and a local reduction of temperature in the screw shanks. At first, the neutron shielding plate, made from polyethylene, positioned between cask bottom and closure plate in reality, was not modeled.

A dummy mass, positioned at the inner bottom side, represents the content (e.g. spent fuel elements). At the considered accident scenario this simplification does not influence the kinematic behavior of the cask during the impact. The Young's modulus of the simplified content was chosen very small to avoid an unintended increase of the cask body stiffness.

At the beginning of the finite element analysis the whole cask was positioned 20 mm above the target, loaded due to gravity and initialized with a velocity of 7.67 m/s resulting from the drop height of 3 m.

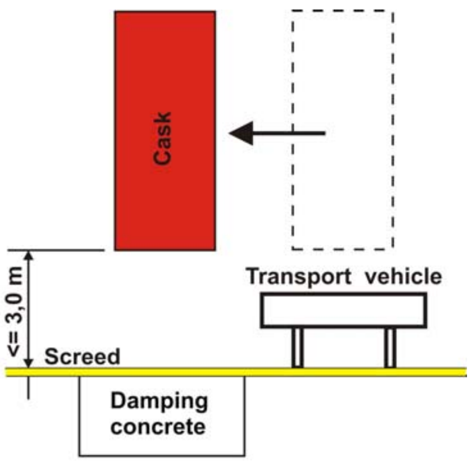

Figure 1. Principle sketch of the considered accident scenario

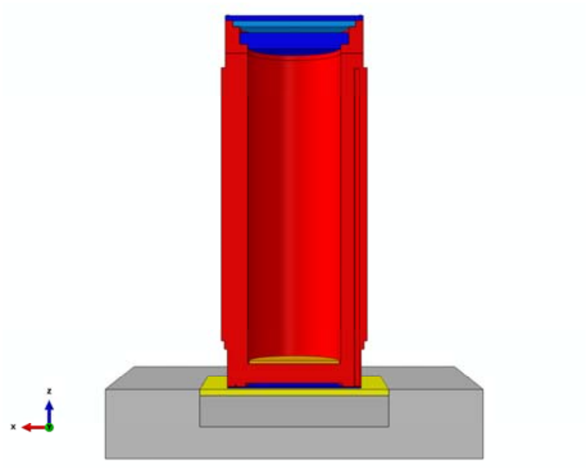

Figure 2. Finite element model of the considered accident scenario

The target in the finite element model represents the unloading area in an exemplary interim storage. It consists of three parts: concrete foundation slab with a block out for the damping concrete and a screed layer. It was assumed, that the concrete foundation slab is very stiff compared to the damping concrete. For that reason it was modeled as a rigid body.

While the modeling of the cask material behavior has already been investigated and verified very well in the recent past, the experience with the modeling of damping concrete and screed have been very poor until now. The finite element material model for the damping concrete, used in the

presented numerical simulation, is the result of comprehensive efforts in the framework of the research project ENREA. In collaboration with partners from industry, different materials, suitable for shock absorbing, have been investigated by specimen and large scale tests [7]. The development and calibration of the used finite element material model for damping concrete is described in detail by [4]. Therefore, the provided parameters for the ABAQUS™ CRUSHABLE FOAM material model were applied but not discussed in this paper. Due to reasons of conservatism the available damage criteria were not applied.
The screed was modeled by using the ABAQUS™ CONCRETE DAMAGE PLASTICITY material model. In addition to the Young's modulus and Poisson's ratio, the plastic compression and tension behavior was described by stress versus inelastic strain curves. The used parameters are based on the German standard for reinforced concrete [5]. They represent a concrete type that is two strength classes above the concrete that is required by the constructional drawings. With that, the assumed increase of concrete strength by aging was considered conservatively.
The crack length limiting reinforcement in the screed layer was modeled as realistic as possible. Crosswise arranged bars, modeled by embedded beam elements, represent the reinforcement mesh. The element size of beams was adapted to the size of screed solid elements. The material behavior of the reinforcement was modeled bilinear and represents a typically used reinforcement steel.
The described finite element model provided the basis of the following modifications and parametric studies. At first, the global kinematic behavior of this reference model (R 1) was checked. The energy balance was plausible. The rate of artificial energy, caused by "hourglassing", remained very small (≈1.3 % referred to the entire model). Further results are exemplarily presented in Fig. 10-15.

MODEL MODIFICATIONS AND PARAMETRIC STUDIES

General

As mentioned above, the investigations presented in this paper are restricted to the cask bottom side impact area. It was assumed that the following modifications and parametric studies influence the loads and their distribution in this region directly:

M 1/2:	Modification of mesh density of screed layer
M 3	Modeling the reinforcement by shell elements
M 4:	Increase of screed strength
M 5:	Reduction of closure plate screws pre-tension
M 6:	Material model change for closure plate from pure elastic to elastic/ideal-plastic
M 7:	Consideration of neutron shielding plate

In order to quantify and to describe the effects of the modifications, three areas at the cask bottom were defined (Fig. 3). Area 1 (left) is the ring shaped outer bottom side under the cask wall. Loads are caused by pressure, normal to the surface, followed by relative tangential displacements between cask and closure plate. The elements at the inner edge were excluded from area 1 and assigned to area 2 (middle). Their loads are additionally caused by the edge pressure of the bending closure plate. The stress state in area 2 is dominated by high shear stresses. A group of elements in the center of bottom cut-out, area 3 (right), was chosen to represent the loads caused by the contact between the closure plate and the oscillating bottom. In each case only the surface layer of elements was analyzed. The maximum values of first principal and equivalent von-Mises stresses of the three areas were compared for all variations (Fig. 10-15).

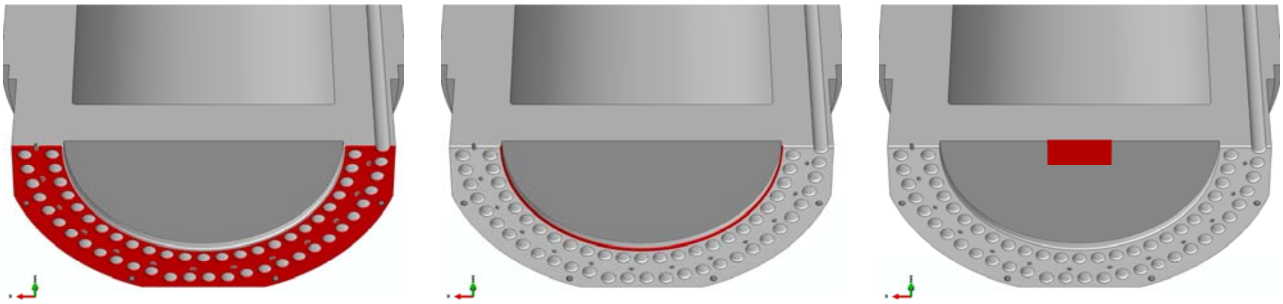

Figure 3. Areas of analysis: area 1 (left), area 2 (middle), area 3 (right)

Modification of Mesh Density of Screed Layer - M 1/2

Based on the reference model R 1 (4 elements/thickness of screed layer), the edge lengths of the nearly cubical-shaped solid elements in the screed layer were doubled (M 1, 2 elements/thickness of screed layer) and bisected (M 2, 8 elements/thickness of screed layer). The lengths of the embedded beam elements, representing the reinforcement bars, were adapted in the same way. The curves of maximum principal stress and effective von-Mises stress, presented for the three considered areas (Fig. 10-15) show no significant discrepancies compared to R 1. The small differences in the maximum values are caused by numerical effects and would have no influence on the assessment results.

Modeling the Reinforcement by Shell Elements - M 3

In reality, reinforcement bars can only absorb tension forces in direction of their mounting. A linear cross-section is defined by diameter and distance of the bars for both directions. In the reference model this situation was accurately represented by cross-wise arranged bars, modeled by beam elements (Fig. 4). In this model the beam elements were replaced by shell elements of a representative thickness (Fig. 5). The element length was adapted to the screed mesh again. In contrast to the reality, multi axial stress states occur now. It is not possible to assign a defined cross section to one mounting direction. However, it is to be noted, that the influence of this often applied simplification stays negligible in this case. Both, the penetration of the cask into the ground and the distribution of loads in the cask bottom remain nearly constant.

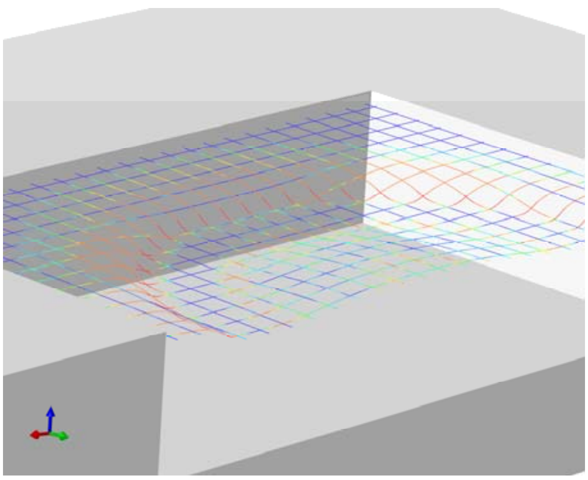

Figure 4. Deformed reinforcement (beam) (blanked screed layer and damping concrete)

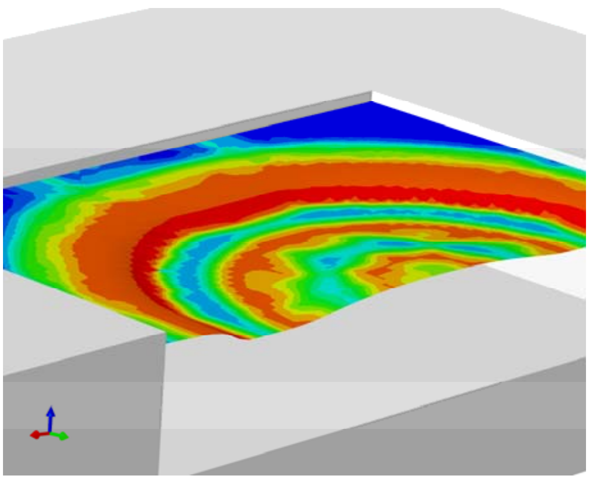

Figure 5. Deformed reinforcement (shell) (blanked screed layer and damping concrete)

Increase of Screed Strength - M 4

Because of the safety requirements in nuclear facilities it was resigned to retrieve concrete samples from the ground construction in the interim storage building. A further increase of one concrete strength class according [5] should consider the uncertainties in the real concrete strength itself and the questions concerning the finite element modeling. Young's modulus, pressure and tension strengths and the assigned strains were adapted to the higher quality. The influence on the relevant loads also remains small. The most remarkable effect of this modification was realized at area 3. The stress curves in Fig. 14/15 and the analysis of the assigned but not presented stress components show for nearly all considered models that the cask bottom oscillates and is bended downwards after the impact . If the center of the closure plate gets in contact (Fig. 8), the bottom is bended upwards. The higher screed strength causes a small increase of the stress, normal to the bottom, in area 3.

Reduction of closure plate screws pre-tension - M 5

Causing from the required locking torque for the closure plate screws, a range of pretension forces is possible. In the reference model R 1 the middle of lower and upper boundary values was chosen to preload the screws. In a modified model the pre-tension was completely neglected. The loads in the considered areas remain nearly unchanged. The maximum principal stress in area 1 for instance decreased by 2.4 %. As a main reason for this fact, the difference between the sum of pre-tension forces as well as the contact forces caused from the impact was identified. The amount of impact force, pressing the closure plate against the cask bottom (area 1/2) is much higher then the resulting pre-tension force. So, the model is nearly insensitive to this modification. It can be assumed, that a variation of the friction coefficient keeps negligible too.

Material model change for closure Plate from purely elastic to elastic/ ideal-plastic - M 6

When the cask penetrates the ground, the upward directed pressure force by the compacted damping concrete causes a strong bending of the closure plate in the area of the bottom cut out (Fig. 8). The material behavior of the closure plate influences the shape of bending and so the contact with the cask body. While it was modeled purely elastic in the reference model, its behavior was changed to elastic/ideal-plastic now. Fig. 6 and 7 show the distribution of maximum principal stress at the moment when the center of closure plate gets in contact with the bottom for R 1 and M 6. The strong bending of the pure elastic closure plate in area 1 and the localized contact at the inner edge of area 1 cause high stresses. The smoother contact of the elastic/ideal-plastic closure plate causes a remarkable reduction of the stresses in area 1 (Fig. 10/11). Plasticization is limited to area 2 now (Fig. 11/13). The plasticized closure plate absorbs the pressure from the damping concrete and prevents the upward bending of the bottom (Fig. 14/15).

However, it should be noted that both the pure elastic and the elastic/ideal-plastic modeling represent extreme cases of the closure plate behavior. The generated results have to be interpreted carefully if they are used in an assessment procedure.

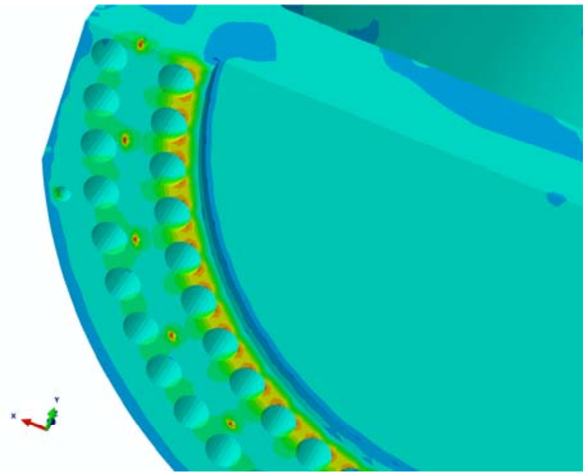

Figure 6. Maximum principal stress (reference model –R 1-)

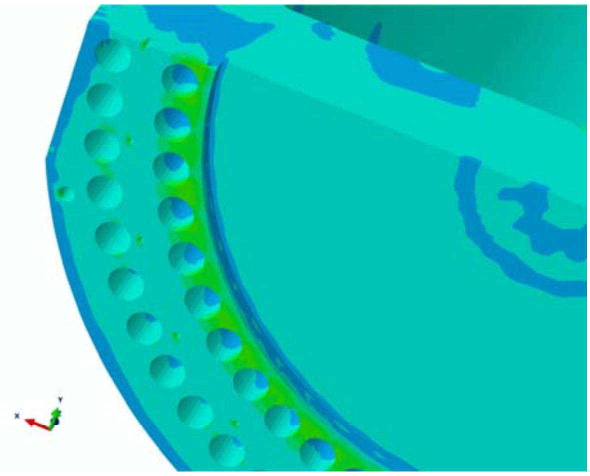

Figure 7. Maximum principal stress (elastic/ideal-plastic closure plate –M 6-)

Consideration of neutron shielding plate - M 7

The effect caused by consideration of the neutron shielding plate, made from polyethylene, is similar to the influence of the elastic/ideal-plastic closure plate. The elastic modeled neutron shielding plate completely prevents the bending of closure plate (Fig. 9). The maximum stresses in area 1 and 2 are significantly reduced (Fig. 10-13). The relatively smooth neutron shielding plate also prevents the upward bending of the bottom. The curve of principal stress (Fig. 14) in area 3 is characterized by oscillations around the zero-point. The pressure stress, normal to the bottom, keeps small. It can be realized, that the neutron shielding plate, in reality and in the finite element model, has a positive impact on the decisive loads in the cask. However, it has to be underlined that the linear elastic material model can not represent the real behavior of the neutron shielding plate. It can be assumed that the correct results are to be found somewhere in between R 1 and M 7.

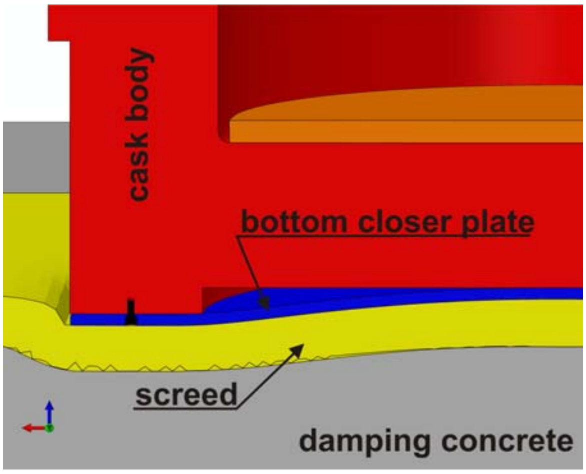

Figure 8. Detail of impact area, deformed (neutron shielding plate not considered)

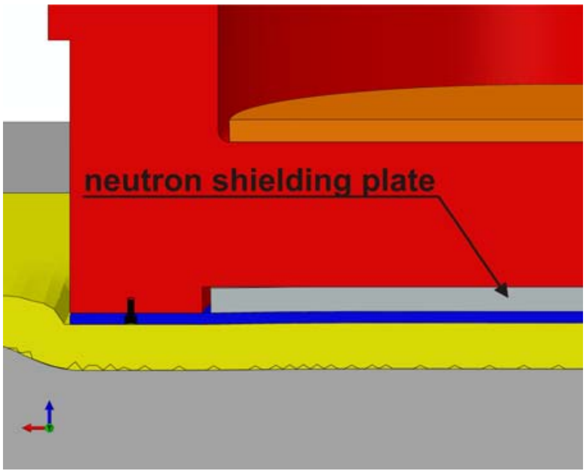

Figure 9. Detail of impact area, deformed (neutron shielding plate considered)

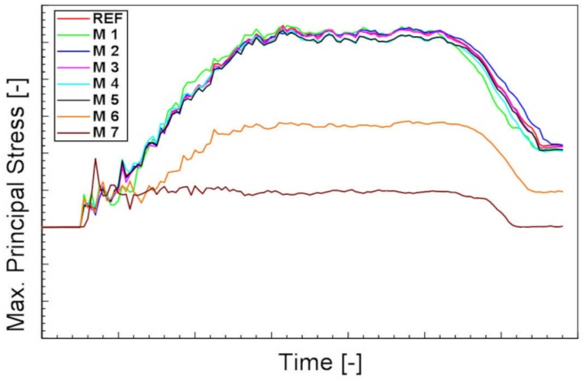

Figure 10. Comparison of maximum principal stress in area 1

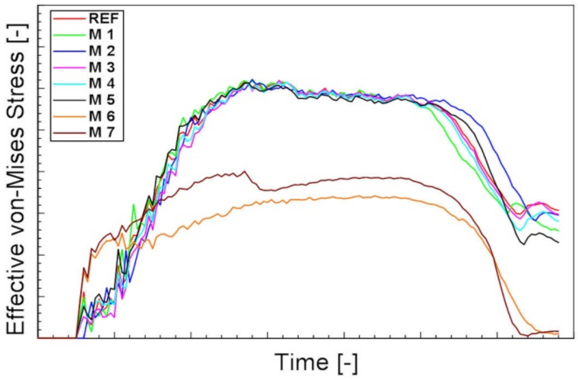

Figure 11. Comparison of effective von-Mises stress in area 1

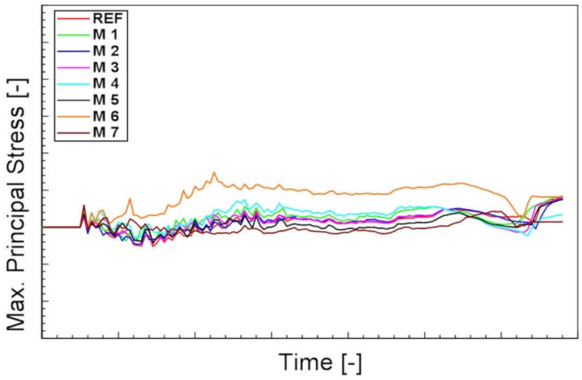

Figure 12. Comparison of maximum principal stress in area 2

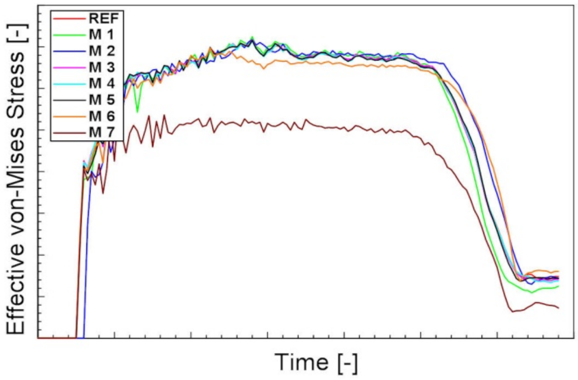

Figure 13. Comparison of effective von-Mises stress in area 2

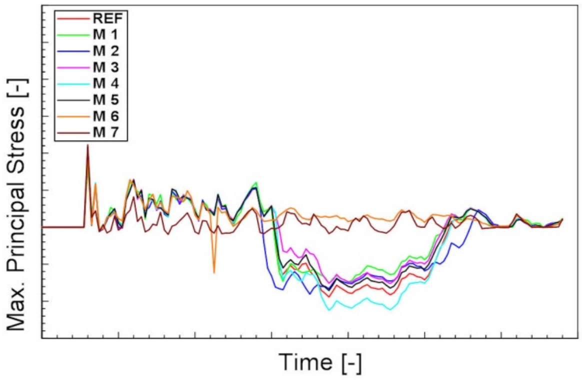

Figure 14. Comparison of maximum principal stress in area 3

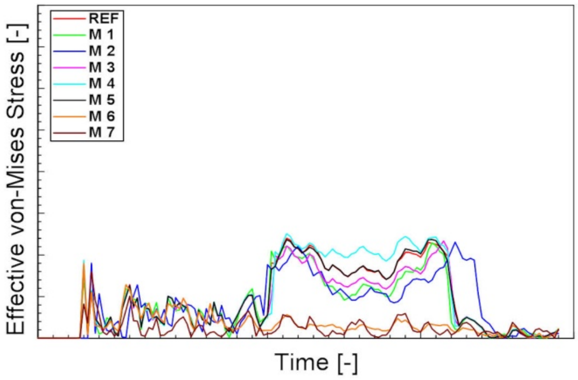

Figure 15. Comparison of effective von-Mises stress in area 3

CONCLUSIONS AND OUTLOOK

Different options exist to evaluate the mechanical behavior of a cask in an assumed accident scenario in a storage facility. In addition to an accurate assessment of the provided documents and (numerical) calculation results, the development of an independent finite element model can be necessary to verify generated results. If no results of drop tests are provided to validate the finite element model it is of particular importance to investigate the sensitivity of the model to various parameters and modifications.

By using the finite element code ABAQUS/Explicit™ an independent dynamical finite element model of a generic storage cask was developed. The influence of selected parameter studies and model modification on the decisive loads to the cask structure during an assumed accident scenario was discussed. The finite element model can help to understand the complex mechanisms of the interaction between the cask components and the floor construction. It provides a tool to carry out further investigations and to answer questions in the context of comparable accident scenarios.

For a further improvement of the finite element model the behavior of screed and the neutron shielding plate (polyethylene) has to be modeled more realistically. To verify the finite element material models that have to be developed and improved respectively, comprehensive material tests are necessary. The available damage criteria for the damping concrete [4] should be applied not before the finite element material model for the screed is able to represent a real damage. Finally the finite element model should be validated by a large scale drop test onto a representative target construction.

REFERENCES

[1] AtG – Act on the Peaceful Utilization of Atomic Energy and the Protection against its Hazards (Atomic Energy Act – AtG) of December 23, 1959 (Federal Law Gazette I, p. 814), as Amended and promulgated on July 15, 1985 (Federal Law Gazette I, p. 1565), last Amendment by Article 1 of the Act amending the Atomic Energy Act of April 20, 2013 (Federal Law Gazette I, p. 921).

[2] RECOMMENDATION of the Nuclear Waste Management Commission (ESK) - Guidelines for dry cask storage of spent fuel and heat-generating waste (revised version of 10.06.2013), (in German), http://www.entsorgungskommission.de/englisch/statements---recommendations--letters/index.htm.

[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] Qiao L., Zencker U., Kasparek E.-M., Völzke H., Scheidemann R.: Development of a finite element model for damping concrete under severe impact loads, 17th Int. Symposium on the Packaging and Transportation of Radioactive Materials (PATRAM 2013), San Francisco, CA, USA, August 18-23 2013.

[5] DIN 1045-1:2001-07 Tragwerke aus Beton, Stahlbeton und Spannbeton, Teil 1 : Bemessung und Konstruktion, Beuth Verlag, Juli 2001

[6] ABAQUS Version 6.10, 2008, Dassault Systèmes Simulia Corp., Providence, RI.

[7] Scheidemann, R., Kasparek, E., Müller, K., Droste, B., Völzke, H.: Model-sized and full-scale dynamic penetration tests on damping concrete, 17th Int. Symposium on the Packaging and Transportation of Radioactive Materials (PATRAM 2013), San Francisco, CA, USA, August 18-23 2013.