11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

DROP TESTS AND DYNAMIC FINITE ELEMENT ANALYSES OF STEEL SHEET CONTAINERS FOR FINAL DISPOSAL OF RADIOACTIVE WASTE

CHRISTIAN PROTZ[1], UWE ZENCKER[2] AND ROBERT LIEBICH[3]

[1] BAM Federal Institute for Materials Research and Testing
Unter den Eichen 87, 12205 Berlin, Germany
e-mail: christian.protz@bam.de, www.bam.de

[2] BAM Federal Institute for Materials Research and Testing
Unter den Eichen 87, 12205 Berlin, Germany
e-mail: uwe.zencker@bam.de, www.bam.de

[3] Technical University Berlin
Chair of Engineering Design and Product Reliability
Straße des 17. Juni 135, 10623 Berlin, Germany
e-mail: robert.liebich@tu-berlin.de, www.kup.tu-berlin.de

Key Words: *Explicit Dynamic FEM, Impact, Drop Test, Steel Sheet Container.*

Abstract. Within a safety assessment, containers for radioactive waste have to withstand drop tests at defined conditions. Alternatively to prototype drop tests, numerical methods can be applied, if they are suitable and sufficiently verified. This paper describes the development of a finite element (FE) model of a thin-walled steel sheet container used to investigate dynamic load scenarios due to impact events. Experimental and numerical analyses were performed for different drop orientations. The results are compared to prove the suitability of the FE model.

1 INTRODUCTION

The use of steel sheet containers is planned for final disposal of non-heat generating radioactive waste in the German Konrad repository. Until now the mechanical safety of the container designs has been proved by drop tests with prototype containers according to the Waste Acceptance Requirements [1]. Alternatively, the safety assessment by calculation is allowed, if the calculations are suitable and sufficiently verified [2] e.g. by comparison with experimental results. For example, in [3] the mechanical behaviour of cast iron containers was analysed by FE calculations. However, reliable numerical simulations of drop tests with thin-walled steel sheet containers for safety assessment purposes currently do not exist. Therefore, a research project was started at BAM Federal Institute for Materials Research and Testing in order to develop a realistic finite element model and an explicit calculation procedure for steel sheet containers under dynamic loads. This paper presents the development of a FE model for the steel sheet container Konrad Type V. In a first step numerical and experimental investigations of flat bottom-side drop tests as well as the drop with the long bottom edge onto an essentially unyielding target with an unloaded steel sheet container are discussed.

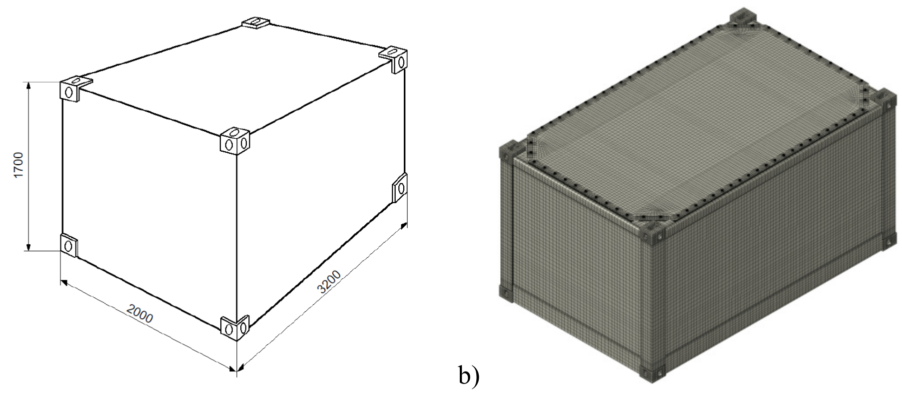
2 DEVELOPMENT OF THE FINITE ELEMENT MODEL

For numerical simulation and investigation of dynamic load scenarios, a reliable FE model is indispensable. At first the load scenario must be transferred into a mathematical model with specification of geometrical and physical properties. Then, this model is transferred into a numerical model by meshing with finite elements [4]. The mesh for the FE model presented in this paper, consisting of the steel sheet container and the impact target, was generated using the Pre- and Postprocessing software LS-PrePost 4.0 and tools from ANSYS 14.5.

2.1 Modelling of the steel sheet container

Within the research project, the steel sheet container Konrad Type V was selected as test object for the experimental and numerical investigations. With its outer dimensions of 3.2 m x 2.0 m x 1.7 m it represents the largest container type according to the Waste Acceptance Requirements for the German Konrad repository [1], cf. Figure 1(a). The container consists of a welded framework structure with eight ISO corner fittings for handling purposes. Its side walls have a sheet thickness of 3 mm, the lid and the bottom plate are 5 mm thick. The lid is connected to the container body with 72 bolts. The unloaded steel sheet container has a weight of approximately 1850 kg.

The finite element model of the container consists of all relevant structural parts (side walls, columns, lid, bolts, etc.). Due to the unsymmetrical construction of the steel sheet container and in order to be able to analyse complex scenarios without symmetry conditions, the specimen is modelled as full model, cf. Figure 1(b). The lid, the bottom side and the side walls, representing thin-walled structural parts, as well as the columns connecting the corner fittings are discretised using shell elements. Volume elements have the disadvantage that due to locking effects several elements in thickness direction are necessary in order to capture bending properly. For the thin-walled container with its aforementioned outer dimensions this would lead to a large number of small elements and additionally to a reduction of the critical explicit time step size for a stable numerical solution, both resulting in an excessive increase of numerical effort. Fully integrated four-node shell elements are used for the FE model. Figure 1(b) shows the FE model with an element size of 40 mm for the shell elements. For investigations considering the mesh sensitivity the element size is bisected in two steps, resulting in a 20 mm and a 10 mm mesh.



a) b)

Figure 1: Steel sheet container Konrad Type V: Schematic drawing acc. to [1] (a) and developed FE model (b)

The ISO corner fittings as well as the stiffening profiles welded to the inner surfaces of the side walls are modelled with fully integrated eight-node solid elements.

2.2 Modelling of the welded joints

All welded connections of the steel sheet container are simplified in the FE model to tied contacts in which the nodes are inextricably tied to the element segments of adjacent structural parts. This has the advantage that all structural parts can be meshed separately, congruent meshes are not necessary. Furthermore, the detailed modelling of the welding seams with throat thicknesses of a few millimetres has the disadvantage that a fine mesh with very small volume elements would be required and hence the related computational costs would increase. A tied contact with an offset option is used which transmits forces and moments between the slave nodes and master segments and also allows the modelling of the sheet geometries with shell elements at the mid-surfaces. This modelling technique for the welded joints is capable to describe the global mechanical behaviour of the steel sheet container during the numerical simulation of the impact events investigated within this paper.

2.3 Modelling of the bolted joints

Modelling of realistic bolted joint behaviour depends on the modelling technique and the contact between the bolts and the clamped parts. In LS-DYNA [5] bolted connections can be represented in different ways by beam elements, discrete springs or solid elements. In order to enable a realistic behaviour concerning both preload representation in the screws and contact between the clamped parts and bolt shanks, the modelling technique with solid elements has been chosen. The M16 bolts and the square profiles with the screw holes are modelled with fully integrated eight-node solid elements, whereas the clamped parts (lid and metal plates) are modelled with four-node shell elements, cf. Figure 2. Numerical contacts between the bolt heads, clamped parts and surfaces of the square profiles are defined with an automatic surface-to-surface contact with a friction coefficient. The bolts are prestressed during a dynamic relaxation phase prior to the explicit transient simulation of the impact event. In order to prevent the lid from penetrating into the bolt shanks in drop scenarios with large deformations and relative displacements between lid and container, a local contact between the nodes of the through-holes in the lid and the outer surface of the bolt shanks is defined using an automatic nodes-to-surface contact.

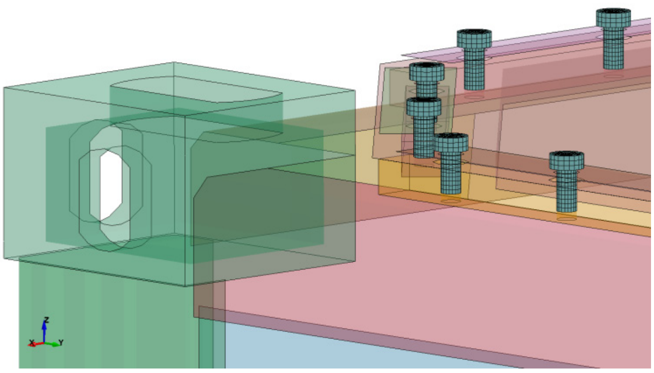


Figure 2: Detailed lid area with modelled bolted joints

2.4 Modelling of the material behaviour

The constitutive behaviour of the structural parts of the steel sheet container is based on an elastic-plastic material model with isotropic hardening except for the bolts which are assumed to behave elastically. The different material parameters for each structural part were taken from literature or material tests and are defined within the elastic-plastic material model by the elastic parameters and piecewise linear hardening true plastic strain vs. true stress hardening curves. Damage or fracture criteria are not considered in the material models used for the FE model described in this paper.

2.5 Modelling of the impact target

The drop tests discussed in this paper were carried out at the BAM drop test facility Test Site Technical Safety in Horstwalde nearby Berlin, Germany, where drop tests with objects up to a weight of 200 metric tons can be performed. The impact target of the test facility consists of a reinforced concrete block with the dimensions 14 m x 14 m x 5 m and an embedded steel plate as impact pad. The steel plate, 10 m long, 4.5 m wide and 0.22 m thick, is fixed to the concrete block. The concrete block and the steel plate have a total mass of 2 613 000 kg [6], which is more than 10 times the mass of possible specimens. Thus, according to the IAEA Regulations [7] the impact target can be considered as an essentially unyielding foundation.

The steel plate is modelled using fully integrated eight-node solid elements with an elastic material model and parameters for steel. The surrounding concrete block is modelled as an elastic half-space using non-reflecting boundaries to prevent artificial stress wave reflections generated at the model boundaries from re-entering the model and contaminating the results. The boundary surfaces between steel plate and concrete block are simplified by using a continuous mesh for both parts.

Figure 3 shows the developed FE model of the steel sheet container Konrad Type V as well as the impact target. An automatic single-surface contact is used for the contact between container and target which also considers contact between all structural parts of the steel sheet container including self-contact of each part. Therefore, any kind of possible drop scenario can be simulated under consideration of large deformations and rotations.

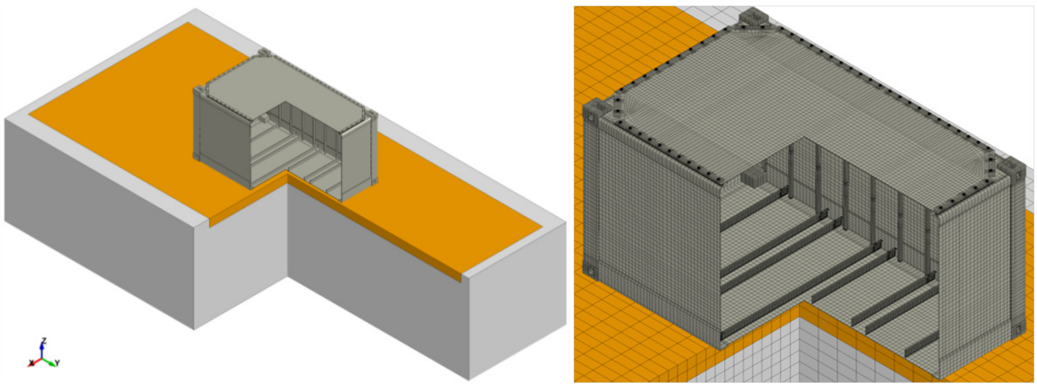


Figure 3: Finite element model of the steel sheet container Konrad Type V and the impact target (one quarter blanked)

3 COMPARISON BETWEEN DROP TESTS AND DYNAMIC FINITE ELEMENT ANALYSES

In view of the dynamic nature of the impact problems investigated, including large plastic deformation and many contacts, the drop tests discussed in this paper were simulated using the explicit nonlinear finite element code LS-DYNA [5]. The numerical results presented here were obtained with the FE model of the steel sheet container meshed with shell elements of 10 mm element size. In order to validate the numerical results and the developed finite element model, several drop tests have been performed with instrumented steel sheet containers at the BAM drop test facility Test Site Technical Safety, cf. Figure 4(a). Hereby, relevant experimental data were gathered by means of accelerometers and biaxial strain gauges, high-speed videos as well as optical three-dimensional (3D) measurement methods. Two flat bottom-side drop tests, one from height of 0.4 m, Figure 4(b), and one from height of 5 m, Figure 4(c), as well as a 5 m drop with the long bottom edge onto the essentially unyielding target, Figure 4(d), were carried out with unloaded steel sheet containers, in order to develop a reliable FE model of the considered container in a first step.

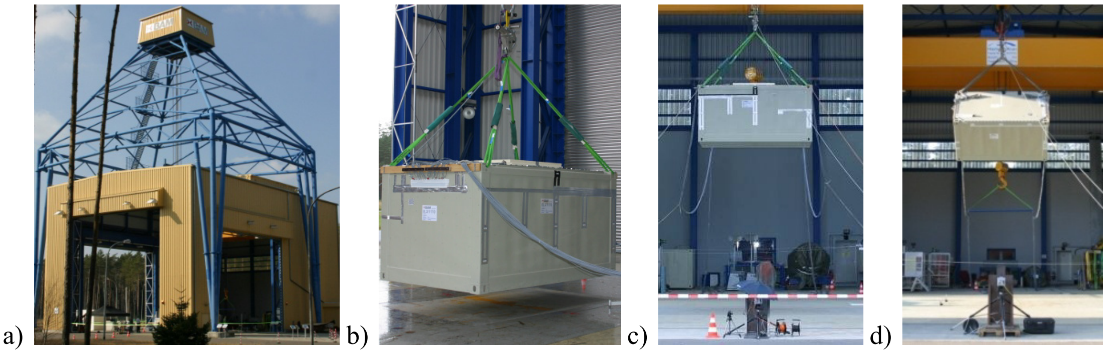


Figure 4: 200 metric tons drop test facility (a), container before the 0.4 m (b) and 5 m (c) flat bottom-side drop test as well as the 5 m drop with the long bottom edge onto the essentially unyielding target (d)

3.1 Flat bottom-side drop tests onto the unyielding target from 0.4 m and 5 m height

Two flat bottom-side drop tests from the heights of 0.4 m and 5 m onto the unyielding target have been carried out consecutively with one unloaded steel sheet container. In both tests the analyses of the high-speed videos and the measured deceleration signals proved that the test container impacted nearly perfectly flat onto the impact target. Nevertheless, small but unavoidable impact angles have been detected and were considered in the numerical simulations in order to obtain comparable results. For both drop heights the comparisons of the measured and calculated local strains and accelerations show a good agreement. Figure 5 depicts exemplarily the comparison of measured and calculated local strains at selected representative measuring points during the 5 m flat bottom-side drop test. The mechanical load due to the impact is initiated into the container at the four bottom corner fittings resulting in the time-strain curve at measuring point 5a03 and proceeds through the welded framework structure into the long upper crossbar, measuring point 5a01. The largest strains can be observed at measuring point 1a05, the middle of the container lid, as a result of its deflection and vibration due to the impact. The experimental results are very good approximated by the numerical simulation in terms of their history as well as the oscillation amplitudes.

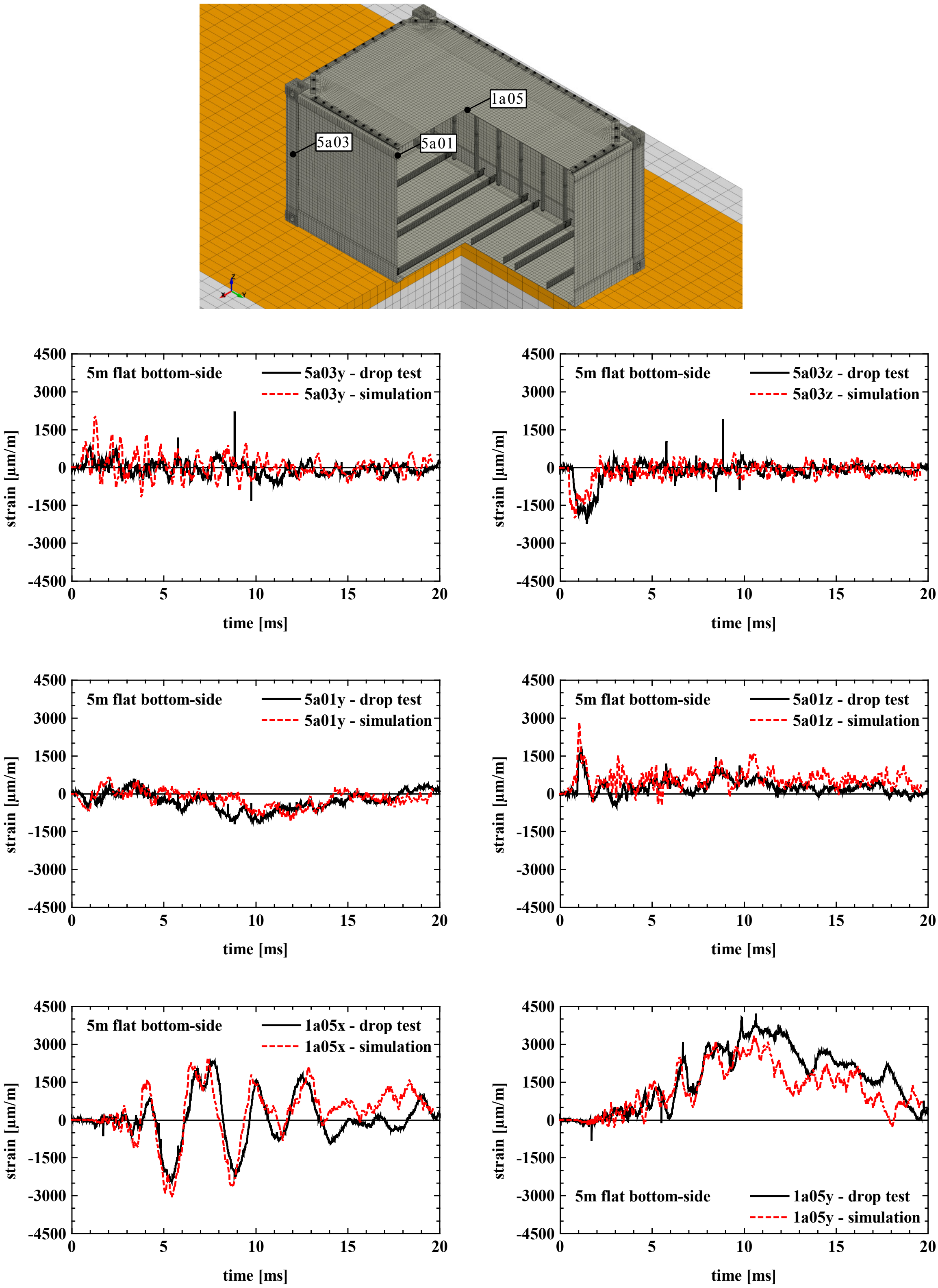
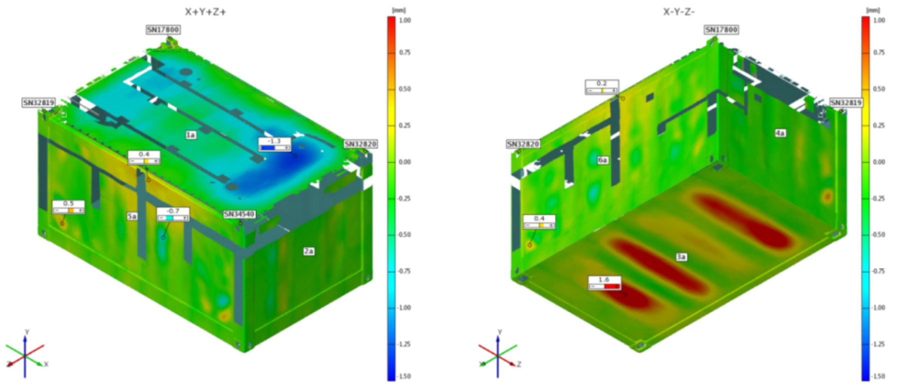


Figure 5: 5 m flat bottom-side drop test: Comparison of calculated and measured local strains at selected measuring points

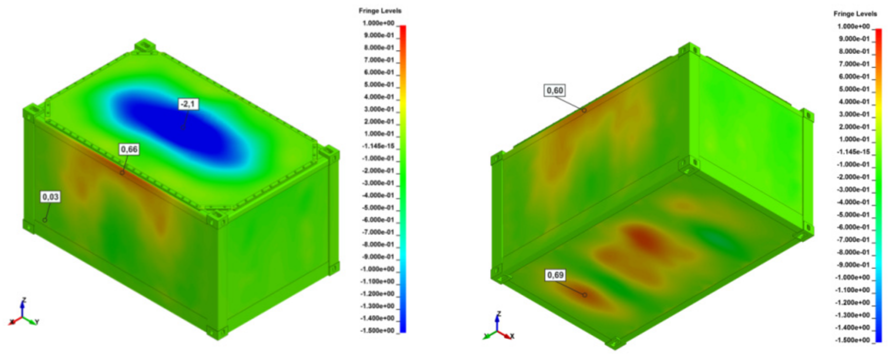
In addition to the usual strain and acceleration measurements, the whole container shape has been measured before and after each drop test using the optical 3D measurement fringes projection method. By calculating the difference between pre- and post-drop measurements, the inelastic container deformation resulting from the impact can be displayed qualitatively and quantitatively. Figure 6 shows exemplarily the 3D container deformation measured after the 0.4 m flat bottom-side drop test. Despite the rather low drop height, plastic deformation is observed at all parts of the test container, in particular at the lid and the bottom sheet.

After simulating the impact event of the corresponding drop test in a first simulation step, the simulation has been restarted in a second step under consideration of a global damping factor (system damping constant [5] of 50.0) and executed to the point of time where the free vibrations due to the impact came to rest. Similarly to the optical 3D measurement method, the difference between the geometrical shape of the container at this point of time and the pre-impact shape was calculated and displayed in LS-PrePost as a user-defined value. Figure 7 depicts the numerically calculated 3D deformation of the FE model of the steel sheet container resulting from the 0.4 m drop test simulation. The comparison of Figure 6 and Figure 7 shows qualitatively a good agreement between the measured and simulated deformation of the container after the 0.4 m flat bottom-side drop.



Experimental results

Figure 6: Measured 3D deformation of the test container after the 0.4 m flat bottom-side drop test



Numerical results

Figure 7: Calculated 3D deformation of the FE model after the 0.4 m flat bottom-side drop simulation

3.2 5 m drop test with the long bottom edge onto the unyielding impact target

For the drop test from a height of 5 m with the long bottom edge onto the unyielding impact target, the test container was positioned in a way that its centre of gravity was located perpendicular above the impacting corner fittings, so that unnecessary rotations due to the impact were avoided. Thus, the impacting energy and the damage of the container were maximized. The analyses of high-speed videos and corresponding deceleration signals show that the container impacted nearly perfectly parallel onto the impact target. However, in order to obtain comparable numerical results, the FE simulation of the drop test was carried out considering the very small impact angle derived from the experiment. The FE model of the container was placed close above the impact target, having imposed an initial velocity of v_0 = 9.904 m/s corresponding to the drop height of h = 5 m.

In the numerical calculations, high strains are predicted at the stiffening profiles welded to the inner side of the side walls, cf. Figure 8. Due to the impact, the side wall deflects in vertical direction resulting in compressive stress and strain as well as plastic deformation at the inner surface of the stiffening profiles. The comparison of the measured and the calculated strain at the corresponding measuring point 5i02 in Figure 8 shows a very good agreement between simulation and experiment. Hence, the measured strain signal proves the suitability of the chosen modelling technique for the welded connections and the material parameters.

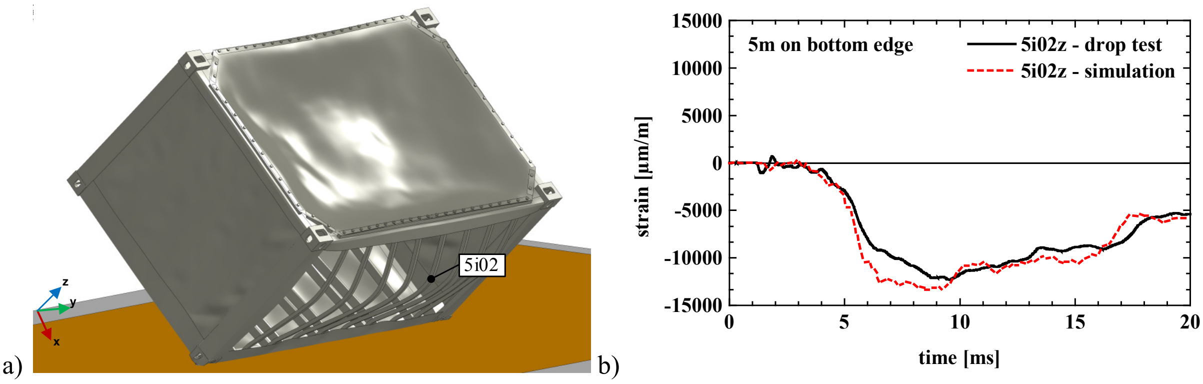


Figure 8: 5 m drop with the long bottom edge onto the unyielding target: 5-times magnified displacement of the container (side wall blanked) at t = 10 ms (a) and comparison of measured and calculated strains at measuring point 5i02 on the inner surface of a stiffening profile (b)

For further validation of the finite element model, the calculated local strains are compared to the strains measured during the drop test at representative measuring points. Figure 9 depicts the comparison at exemplarily selected measuring points, one located in the middle of a vertical pillar connecting the upper and lower corner fittings (5a04), one in the middle of the long upper crossbar connecting the upper corner fittings (5a02) as well as the measuring point located in the middle of the outer surface of the lid (1a01). Due to the impact, the mechanical load is applied at the lower corner fittings of the welded framework structure and transmitted over the bolted connections to the container lid. The history as well as oscillation amplitudes have been approximated sufficiently exactly by the numerical simulation at the considered measuring points. Furthermore, the calculated load delay in the FE simulation shows a good agreement with the load delay of the measured strain maxima.

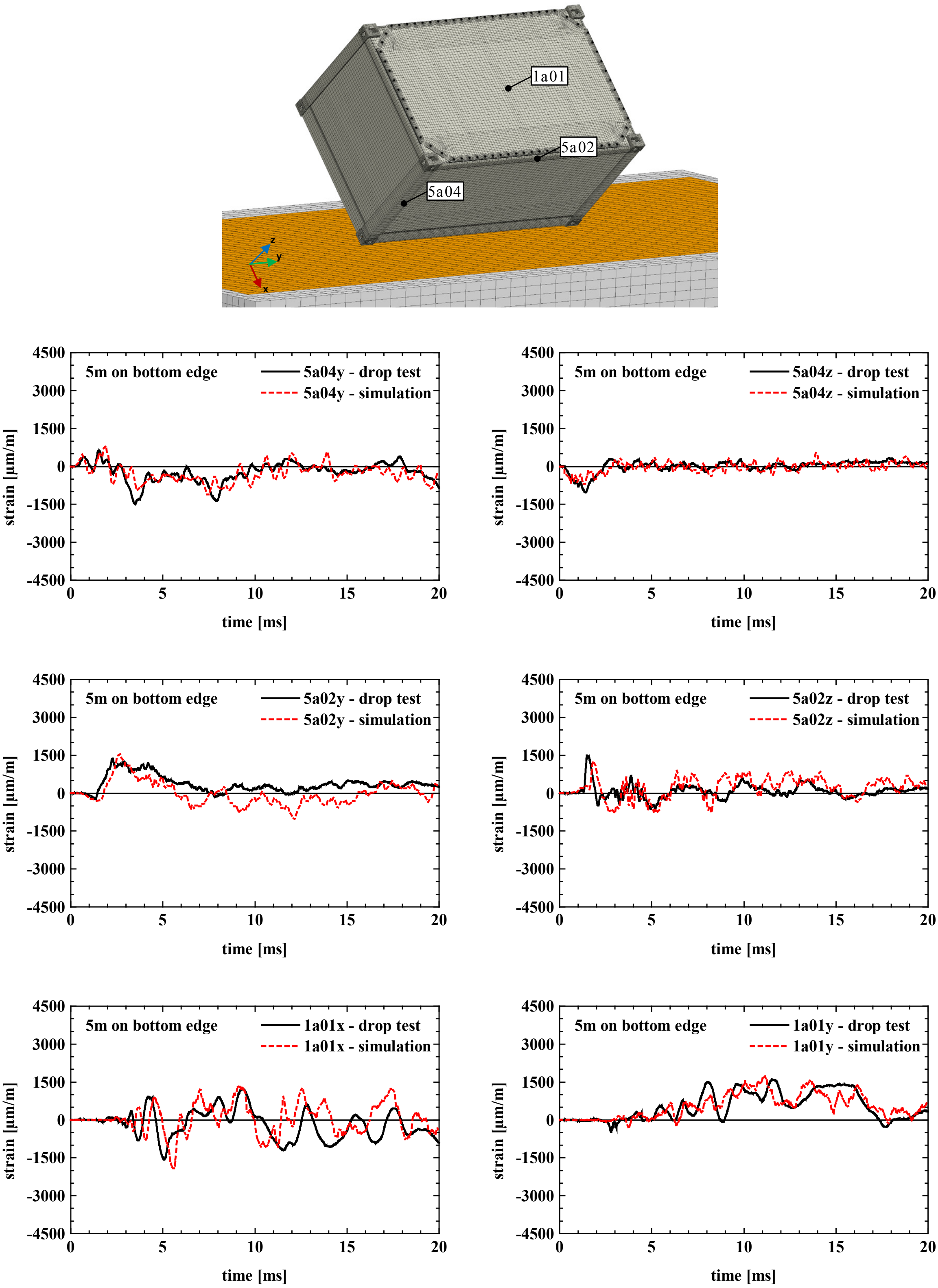


Figure 9: 5 m drop with the long bottom edge onto the unyielding target: Comparison of calculated and measured local strains at selected measuring points

During the drop test, shear failure has been observed at four of the total 72 lid bolts. After hitting onto the impact target, the container decelerates and stops, whereas the lid moves further in vertical direction. After overcoming the clamping and frictional forces of the bolted connections, the lid glides tangentially along the container opening and comes into contact with several bolt shanks, resulting in highly stressed areas and plastic deformation in the parts involved. Figure 10(a) shows exemplarily the plastic deformation after the drop test at a through-hole due to the contact between lid and bolt shank.

Therefore, it follows that it is inevitable to use a bolt modelling technique with solid elements in order to model the interaction between bolt shank and container lid appropriately. The numerical simulation with the chosen modelling technique reproduces the contact behaviour in a physically realistic manner. Figure 10(b) shows the distribution of von Mises stress at t = 4.0 ms with the maximum stress calculated at a through-hole due to contact between bolt shaft (blanked) and lid. Bearing stress and compression of the strained shell elements at the through-hole are predicted in the contact zone. The bolts are currently modelled with an elastic material model. For a proper failure prediction the FE model should be extended with an applicable damage and failure modelling technique in the future.

a) b)

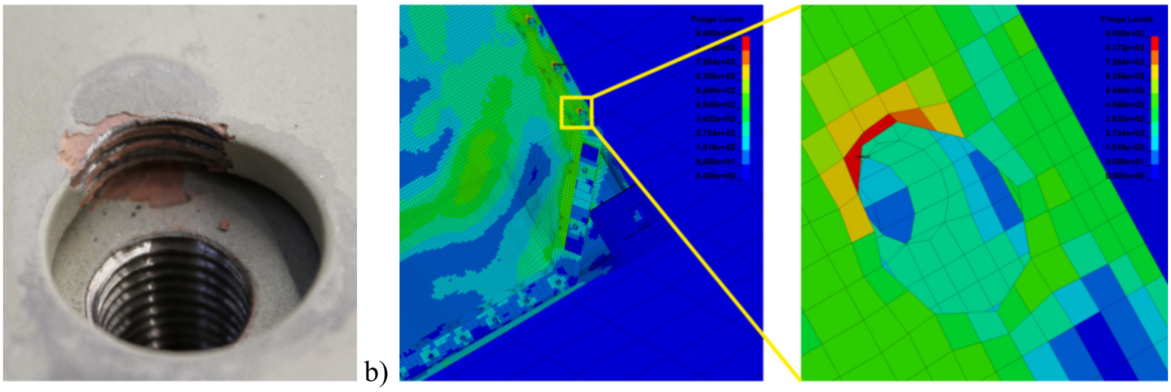


Figure 10: Visible plastic deformation at a through-hole due to contact between lid and bolt shaft (a) and distribution of von Mises stress in the lid during the FE simulation at t = 4.0 ms (screws blanked) (b)

In addition to the strain and acceleration measurements, a stereo photogrammetry approach has been applied in a high-speed camera configuration to investigate the three-dimensional kinematic rigid body decelerations and dynamic deformation of the container and its structural parts, respectively [8]. This provides another option to compare experimental and simulated data in order to validate the numerical simulation. Figure 11(a) depicts the defined distances L1 and L2 between selected container surface points with magnitudes of approximately 1.5 m. The measured time-dependent changes of L1 and L2 are shown in Figure 11(b). Resulting from the impact, the test container is compressed in vertical direction with distance L1 up to a minimum of -0.2 %, whereas the change of the distance L2 shows a stretching effect in horizontal direction with a maximum of about 0.4 %. Comparable results are obtained by the numerical simulation in terms of the spacious compressive and stretching effects. Despite the slightly smaller values in the numerical simulation results, the time-dependent changes of the lengths as well as their remaining inelastic deformation are reproduced in a good agreement with the experimental results.

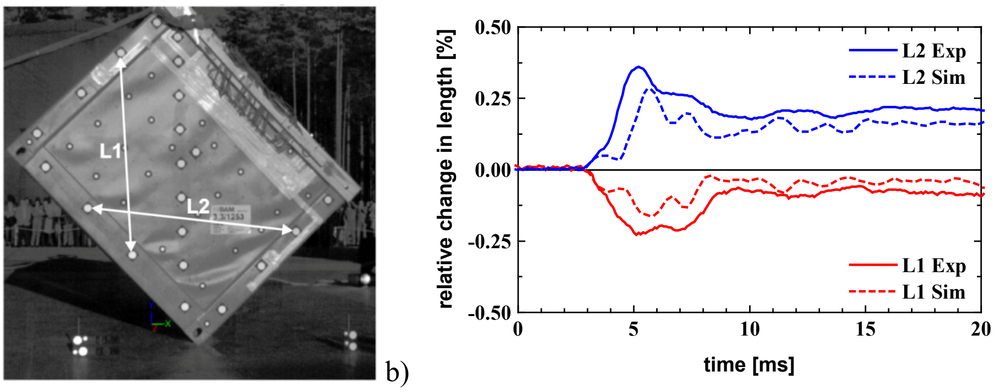
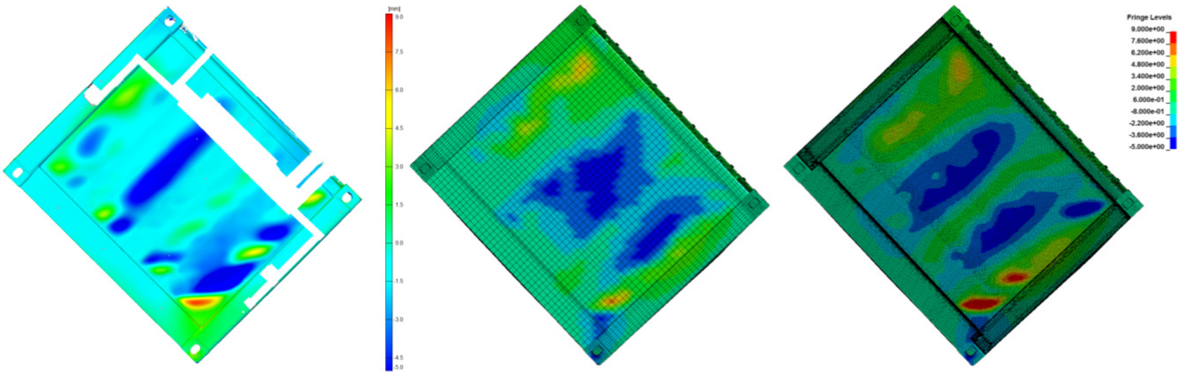
a)  b)

Figure 11: Dynamic deformation analysis during the 5 m drop onto the long bottom edge: Definition of distances between selected surface points (a) and comparison between experimental and numerical results (b)

As a last point, in order to investigate the influence of the mesh density, the comparison between the measured and simulated inelastic deformation of a small container side wall is considered. The experimental result in Figure 12(a) shows that buckling of the container side wall occurred mainly near the impacting ISO corner fitting.

The drop test has been simulated using a coarse mesh with 40 mm element size and a finer mesh with 10 mm element size created by bisecting the coarse mesh in two steps. The fringe plots of the calculated inelastic deformation results in Figure 12(b) and Figure 12(c) are scaled in accordance to the experimental range used in Figure 12(a).

From the comparison of the experimental and numerical results depicted in Figure 12, it can be seen that in order to display an appropriate deformation pattern by the numerical simulations, a fine mesh with an element size of 10 mm is necessary. The local buckling effect near the bottom corner fitting is reproduced insufficiently by the simulation with the 40 mm mesh, whereas with a 10 mm mesh a good agreement between experimental and calculated result can be reached.



a) Experimental result b) FEM (40 mm mesh) c) FEM (10 mm mesh)

Figure 12: Inelastic deformation of the small container wall after the 5 m drop with the long bottom edge onto the unyielding target: 3D surface shape digitisation of the test container (a) and simulated results with a coarse mesh (b) and a finer mesh (c)

4 CONCLUSIONS

This paper presents the development of a FE model of the steel sheet container Type V for the final disposal of non-heat generating radioactive waste in the German Konrad repository. Due to the fact that reliable numerical simulations of drop tests with thin-walled steel sheet containers currently do not exist, a research project was started at BAM Federal Institute for Materials Research and Testing with the objective to simulate drop tests and analyse dynamic load scenarios. The numerical model is described in view of element definitions, modelling techniques for the welded and bolted joints as well as contact conditions and material models. Different drop test scenarios are discussed and simulated using the explicit nonlinear FE code LS-DYNA. The results of the numerical calculations are validated with experimental results.

Altogether, the presented comparisons of the calculated and measured results prove that the developed FE model, the chosen numerical modelling techniques as well as the contact and material parameters are suitable to describe the impacting behaviour of the steel sheet container during the considered drop scenarios sufficiently. It could be shown that the developed FE model is able to simulate the mechanical behaviour of the box-shaped steel sheet container during the short primary impact of the drop as well as the rebound with free bending vibrations of the container walls. Based on this good agreement of the FE model, it is planned to investigate more complex drop scenarios including possible damage or failure of container components. For container safety assessment, an accurate mechanical simulation allows the prediction of container loads and a comparison of different load cases to determine unfavourable, most damaging drop scenarios within future safety assessment procedures.

REFERENCES

[1] 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, (2010).

[2] BAM Federal Institute for Materials Research and Testing. *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, (2003).

[3] Zencker, U., Weber, M., Qiao, L., Droste, B. and Völzke, H. Mechanical safety analyses of cast iron containers for the KONRAD repository. In *Proc. of the 16th International Symposium on the Packaging and Transportation of Radioactive Materials*, paper #220, (2010).

[4] Zencker, U., Wieser, G., Qiao, L. and Protz, C. Modeling strategies for dynamic finite element cask analyses. In *Proc. of the 17th International Symposium on the Packaging and Transportation of Radioactive Materials*, paper #280, (2013).

[5] Livermore Software Technology Corporation. *LS-DYNA Explicit Finite Element Code*. Version smp d, Release 7.0.0, Revision 53450, (2009).

[6] Müller, K., Quercetti, T., Melnik, N. and Droste, B. Impact target characterisation of BAM drop test facility. *Packaging, Transport, Storage and Security of Radioactive Material* (2008) **19**:217-221.

[7] International Atomic Energy Agency. *Advisory Material for the IAEA Regulations for the Safe Transport of Radioactive Material*. Safety Guide No. TS-G-1.1 (Rev. 1), (2008).

[8] Gründer, K.-P., Kadoke, D., Protz, C. and Zencker, U. Optical 3D methods in steel sheet container drop test analysis. In *Proc. of the 11th International conference on optical technologies for sensing and measurement*, (2013) **O4**:74-78.