20th International Conference
ENGINEERING MECHANICS 2014
Svratka, Czech Republic, 12 – 15 May 2014

SIMULATION OF DROP TESTS OF THE CASK FOR RADIOACTIVE WASTE

V. Ivančo[*], M. Orečný[**], R. Huňady[***]

Abstract: *The paper deals with finite element simulation of steel cask drop test. The analyzed cask is designed for transport of solid radioactive waste. The paper describes simulation when the cask is falling down from a height of 9 meters on the side part. The main attention of the paper is devoted to procedures and methods of evaluation of the results. The performed simulation revealed that the design is insufficient for qualification of the cask according to relevant rules.*

Keywords: Cask, Radioactive waste, Drop test, Simulations, Finite Element Method.

1. Introduction

In order to eliminate risk of radioactive pollution, special conditions defined by international rules as International Atomic Energy Agency regulations and national decrees (e.g. NRA 2006) have to be satisfied. The rules define conditions for storage and transport of shipments containing radioactive materials. Important parts of the rules are requirements of shipments resistance in accidental conditions. They define some hypothetical accidents as a drop of the shipment from a given height, specific action of fire etc. Resistance is usually evaluated by testing, computations or by employing both methods. Tests performed for full-scale prototypes or for their models of appropriate scale (Droste, 2007, YooJeong-Hyoun et al., 2011, Trebuňa et al., 2012) are demanding and expensive as special testing facilities are necessary. Moreover, specimens used in tests are damaged and more specimens have to be manufactured for repeated tests. This is a reason for numerical simulation of tests by Finite Element Analysis (FEA). If tests are performed, simulations represent very useful tools for improving knowledge and for better understanding of the test results. Simulations usually serve for the assessment of cask resistance during the design process to reduce time and cost required to develop a final product (e.g. Rueckert et al., 1993, Jakšič and Nilsson, 2007). As it follows from comparison of experimental and simulation data, FEA based simulations can give good agreement with tests and realistic estimation of product resistance (Qiao et al., 2011, YooJeong-Hyoun et al., 2011). This paper deals with assessment of resistance of the steel cask exposed to a drop from height of 9 m. The simulations were performed using drop test simulation module of CAD system SolidWorks 2012. Described and discussed are methods of results evaluation for the assessment of the cask applicability. Description is limited to the initial engineering design and of the cask.

2. Computational Model

The initial engineering design of the cask intended for transport of radioactive waste is schematically drawn in Fig. 1. The cask consists of cylindrical body (1) covered with lid (2). Thicknesses of these parts were determined from requirements on shielding of radioactive radiation. Upper end of the lid is equipped with deformation elements serving also for manipulation. Deformation zone with a plate for fixing the cask to a transport device is at bottom part of the cask body. Both main parts of the cask are joined with

---

[*] Assoc. Prof. Ing.Vladimír Ivančo, CSc.: Faculty of Mechanical Engineering, Technical University of Košice, Letná 9; 042 00, Košice; Slovakia, vladimir.ivanco@tuke.sk

[**] Ing. Martin Orečný: Faculty of Mechanical Engineering, Technical University of Košice, Letná 9; 042 00, Košice; Slovakia, martin.orecny@tuke.sk

[***] Ing. Róbert Huňady, PhD.: Faculty of Mechanical Engineering, Technical University of Košice, Letná 9; 042 00, Košice; Slovakia, robert.hunady@tuke.sk

bolts (4) to enable loading and unloading transported waste placed in a standard steel barrel (3).Transported waste, mostly pieces of structural parts, can vary from case to case; hence the barrel was modeled as a cylinder of homogenous material with density following from cargo maximum allowable weight and volume of the barrel. Finite element mesh consisting of ten node tetrahedrons is in Fig. 2. As obvious from figure (2), smaller elements had to be used to model bolts and their connections to body and lid.

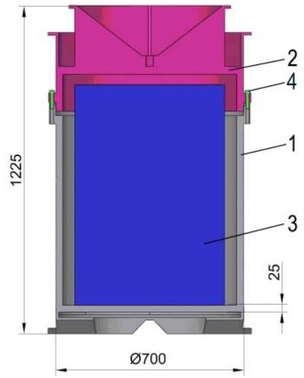


Fig. 1: Scheme of the cask initial design.

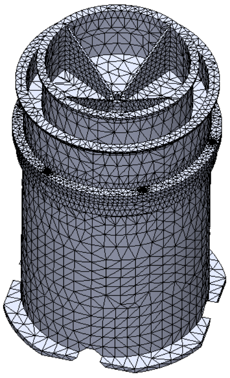


Fig. 2: Finite element mesh.

Simulations were performed by explicit method. The mesh refinement then brought some complications to computations as velocity c of stress waves in a 3D continua modeled by solid finite elements is

$$c = \sqrt{\frac{E(1-\mu)}{\rho(1+\mu)(1-2\mu)}}, \tag{1}$$

where E, μ and ρ are modulus of elasticity, Poisson's number and material density respectively. In order to achieve stability of the method, time increments Δt had to be less than $\Delta t = h/c$, where h is the size of the smallest element in the mesh.

The cask and lid were designed to be made of welded hot-rolled steel. Designed material of bolts was heat treated steel, with high tensile strength and low ductility. For these parts a bilinear elasto-plastic material model with kinematic hardening was used. The corresponding material properties are listed in Tab. 1, where R_e, R_m and A are the yield strength, ultimate strength and ductility respectively. The impact pad was considered as rigid; hence no material properties were needed to define.

Tab. 1: Material properties.

	E (MPa)	**μ (–)**	**R_e (MPa)**	**R_m (MPa)**	**A (–)**	**ρ (kg/m^3)**
Cask	200 000	0.30	345	625	0.3	7 850
Barrel	14 400	–	–	20	–	1 600
Bolts	200 000	0.30	1 100	1450	0.08	7 800

3. Results Evaluation

Evaluations were based on stress plots in individual time instances, plots of stress envelopes (i.e. their maximal values from all time instances), plots of strains and time courses of stress and strain in selected points of the model specified as sensors.

The cask impacted onto its left side and the contact between the impact pad and the bottom and upper part of the cask occurred at same time. Progression of stress wave is observable from stress fields in Fig. 3 and Fig. 4. As color scale was set from zero to yield strength, it is visible that the von Mises stress reached and in some areas exceeded the yield strength of the material. This implies that plastic strains of the cask will occur after the impact. As an accidental situation is investigated, permanent deformation of cask jacked can be tolerated (or they cannot be avoided) providing that they have no effect on its integrity. To check structural integrity of the cask jacket the maximum values of nodal von Mises stresses are depicted in Fig. 5. Setup of color map is such that the red color represents the values of stress equal to the ultimate strength 625 MPa. It is obvious that the values of stresses are less than 418 MPa i.e. slightly above the yield strength but markedly under the ultimate strength. Deformation of the cask is notable in Fig. 5. On

Fig. 3 the stress wave is moving from the impact zone to the upper side of the container. The maximum value of the von Mises stress in time 50 µs is 347.4 MPa in the contact zones.

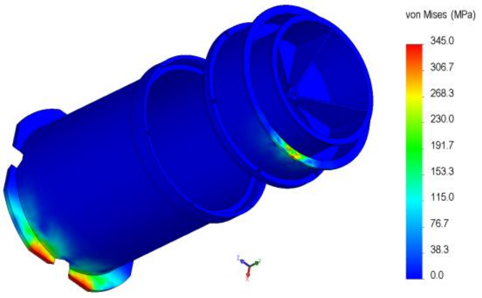


Fig. 3: Von Mises stresses at time 50 µs.

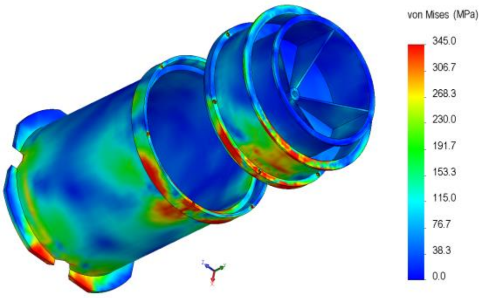


Fig. 4: Von Mises stresses at time 750 µs.

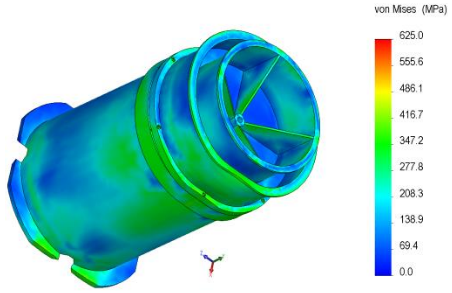


Fig. 5: Envelope of von Mises stresses.

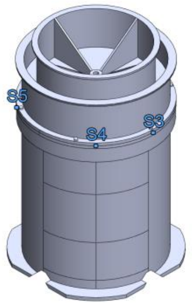


Fig. 6: Sensors on flange of the cask.

In order to acquire more information about the cask behavior during impact, other sensors were inserted on flanges of the cask, see Fig. 6. Time course of the von Mises stress in sensor S3 (see Fig. 7) shows that after sudden increase at beginning of impact, the stress does not change almost at all. This is caused by continual deformation of contact zones. The maximal value of the von Mises stress is over the yield stress of the material, therefore the cask will deform plastically. In contrary, values of stress in sensor S4 changes more frequently, see Fig. 8. The maximal value of stress is below the yield strength of the material.

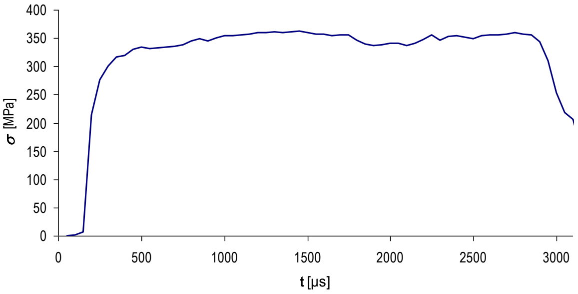


Fig. 7: Time course of stress in sensor S3.

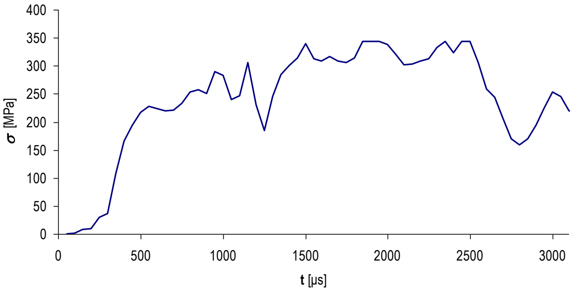
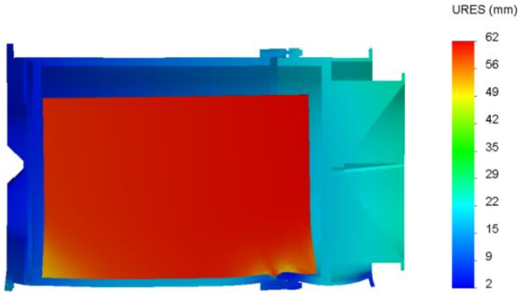


Fig. 8: Time course of stress in sensor S4.

For better understanding of the cask behavior after impact it is necessary to evaluate displacements. Full contact of both bodies occurred after time 3000 µs. This is visible in Fig. 9 displaying resultant displacements at time 5000 µs. As deformed shape is drawn in true scale, comparatively large gap between cask body and lid flanges is obvious.

To investigate origination of the gap, time courses of displacements in the y-axis direction of two adjacent nodes of both flanges are drawn in Fig. 10. It is clear that gap originated after 2500 µs from impact. As stresses in flanges are small, the only explanation is that plastic deformation originated in bolts. This is documented by envelopes of tensional stresses in bolts in Fig. 12.

Stress range in Fig. 11 was set from 0 to 1200 MPa. It is visible that tensile stresses in all six bolts exceeded the yield strength of material which is 1200 MPa. Bolts deformed plastically which caused origination of a gap between flanges. The maximal node value of the tensile stress is 1569 MPa. This value is above the tensile strength of the material, meaning that the bolts would rupture. It should be noted that simulation program does not enable modeling damage and rupture of a material, hence stress values could be larger than the ultimate strength.



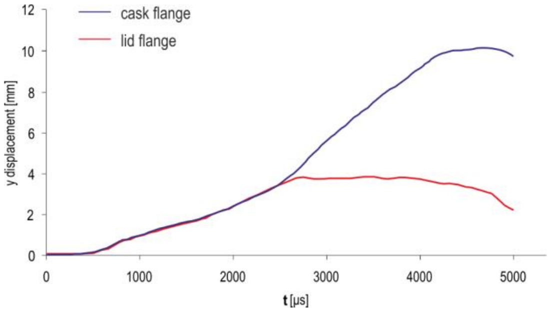
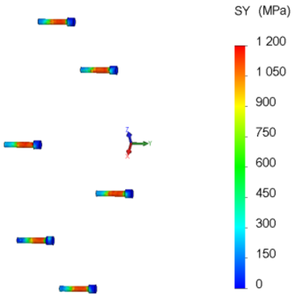


Fig. 9: Displacements at time 5000 μs.

Fig. 10: Displacements of adjacent nodes of both flanges.



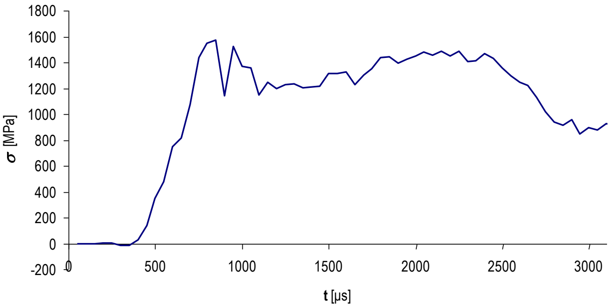


Fig. 11: Maximum tensional stress in bolts.

Fig. 12: Time course of normal stress in maximally loaded bolt.

4. Conclusions

Drop tests simulated with initial engineering design of the cask for transport of radioactive waste were studied. The two of tests demanded by relevant rules are described and methods of results evaluations are presented. From evaluations of results it follows that the design did not meet conditions of its qualifications and therefore changes of the design were necessary. The subsequent design modifications are beyond the scope of this paper.

Acknowledgement

The work has been accomplished under the research project financed by the grant 1/0090/12 of the Scientific Gran Agency of the Ministry of Education of the Slovak Republic and Slovak Academy of Science and the faculty project O-06-103/0008-00.

References

Droste, B. (2007) Testing of type B packages in Germany to environments beyond regulatory test standards, Packaging, Transport, Storage and Security of Radioactive Material, Vol. 18, No. 2, 73-85.

Jakšič, N., Nilsson, K. F. (2007) Numerical simulation of the one meter drop test on a bar for the Castor cask, DG JRC Institute for Energy, Petten (the Netherlands).

NRA (Nuclear Regulatory Authority of the Slovak Republic) (2006) Decree 57/2006 of the Nuclear Regulatory Authority of the Slovak Republic, Collection of laws Nr 57/2006.

Qiao, L., Zencker, U., Musolff, A., Komann, S. (2011) Dynamic Finite Element Analyses of a Spent Fuel Transport and Storage Cask with Impact Limiters by 9 Meter Drop Tests, In: Proc. SIMULIA Customer Conference, Barcelona, Spain, pp. 932-945.

Rueckert, J., Diersch, R., Marcault, P. (1993) Assessment of Cask Loading in Drop Tests with Finite Element Methods, Structural Mechanics in Reactor Technology, In: Proc. 12th International Conference, Vols. J; Stuttgard, pp. 109-114.

YooJeong-Hyoun et al. (2011) Arising Technical Issues in the Development of a Transportation and Storage System of Spent Nuclear Fuel in Korea, Nuclear Engineering and Technology, Vol. 43, Issue 5, 413-420.

Trebuňa, F., Šimčák, F., Huňady, R., Pástor, M., Bobovský, Z. (2012) Modal analysis of transport complex and drop tests of container for transport of spent nuclear fuel, In: Proc. 50th International conference on Experimental Stress Analysis 2012, Tábor, Czech Republic, pp. 493-500.