COMPUTATIONAL MECHANICS 2019
35th conference with international participation
Srní
November 4 - 6, 2019

Contribution to computational analysis of tightness prediction of complex technical systems

M. Sága[a], M. Handrik[a], M. Vaško[a], P. Kopas[a], A. Sapietová[a], L. Jakubovičová[a]

[a] *Department of Applied Mechanics, Faculty of Mechanical Engineering, University of Žilina, Univerzitná 1, 010 26 Žilina, Slovakia*

1. Introduction

We are faced with the requirement to analyze the state of a technical device in the event of its fall from some height in technical practice. Special attention must be given to the various storage and transport containers intended for the transport of dangerous substances. The problem of loss of tightness of technical equipment during its fall is a complex computational problem. Therefore, it often has to include the following partial tasks:

- Solution the dynamic problem of continuum mechanics using implicit or explicit time integration.
- Solution with a relatively small time step, requiring the solution of a large number of time steps.
- Solution of geometrically nonlinear problem.
- Solution of material nonlinear problem.
- Solve contact problem.

2. Model of solved problem

The simulation of the fall of the transport container is solved in the FE program ADINA. The geometry of the transport container is formed by a large number of bodies (the model consists of almost 200 geometric bodies). A large number of geometric bodies have a significant effect on the number of defined contact pairs in the model. By considering all possibilities of contacting bodies we receive more than 200 contact pairs. The transport container consists of two cylindrical vessels, which are inserted into each other. In this case, there are large areas in contact with a large number of elements. A large number of contact pairs and large contact areas have a high demand for contact detection in the calculation process.

The complexity of the shape geometry requires the use of cubic elements in size from 0.25 mm to 2 mm in meshing. The geometric model is shown in Fig. 1. The finite element mesh has more than 20 million elements and the number of equations exceeds 14 million. Considering the size of the elements and the convergence criterion for explicit time integration, it is necessary to use a very small time step (of the order of 1×10^{-9} seconds). When solving the impact of the transport container we consider a solution time of 4×10^{-3} seconds. Body contact needs to be detected more than 4 million times using explicit time integration. By using implicit time integration, time step 1×10^{-5} and 400 time increments can be used for the solution to the desired time. The solution of 15 to 20 iterations of the non-linear solver is required at each time step.

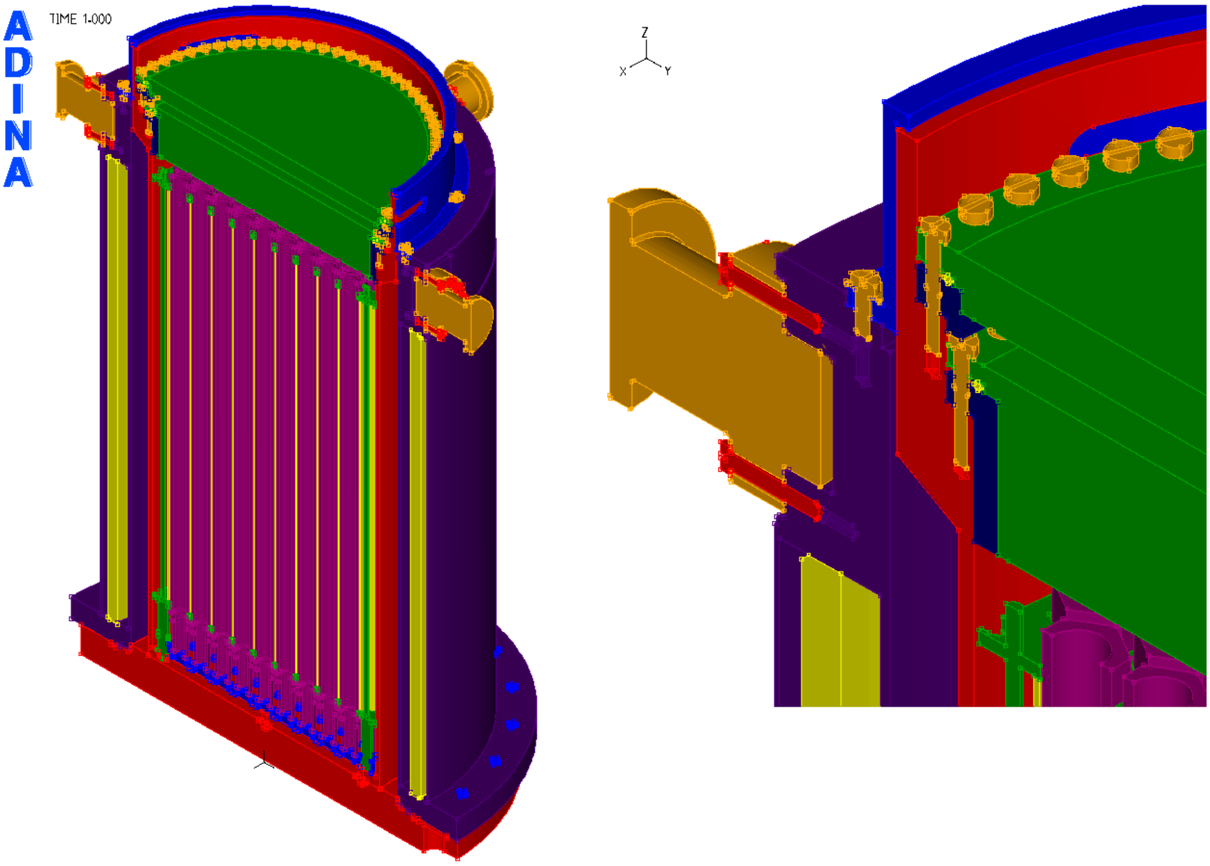

Fig. 1. The geometric model of transport container

Practical calculations have shown that using implicit time integration is significantly more efficient than using explicit time integration. Implicit time integration is more than five times faster than explicit time integration. Placing the body near the bottom and defining the initial velocity determined by the free fall relationship of the body allows a solution for a short period of time even when falling from a large height.

The following material models are used to define the material properties of the individual parts of the model:

- bilinear material model - for less important parts of the structure,
- multilinear material model - for parts of the structure that are directly related to the tightness of the transport container.

The elements switch-off criterion when reaching the material strength limit is used in a multilinear material model. Fall test simulations were performed for 20 positions of impact of the shipping container on the base.

3. Analysis of results

We focus on evaluating the following properties of the structure when analyzing the results:

- Breaking the casing tightness of the transport container, the formation of a crack - monitoring the switching off of elements in the casing of the transport container.
- Breaking the fastening screws of the transport container, breaking the screws - monitoring the switching off of the elements in the screws.
- Plastic deformation of screws of the transport container, permanent loosening of the gasket - loss of contact forces on the gasket or their significant decrease. The gasket release status is shown in Fig. 2.

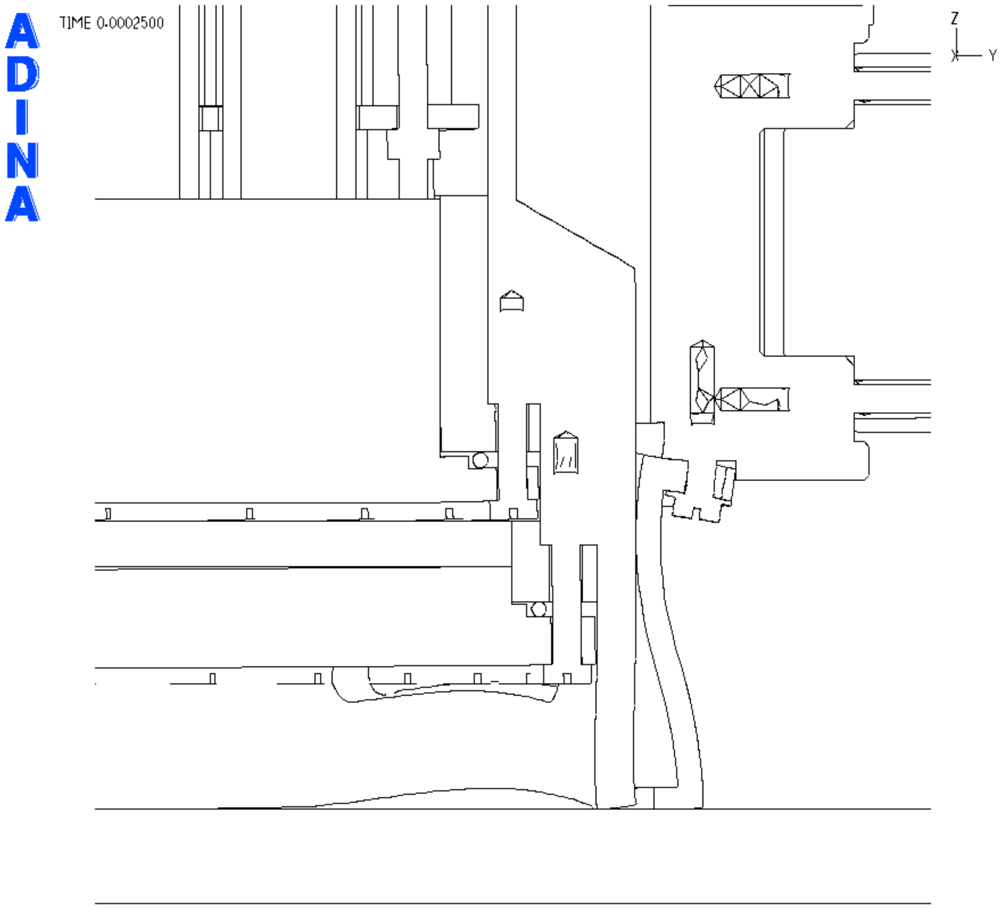

Fig. 2. The gasket release status after analysis

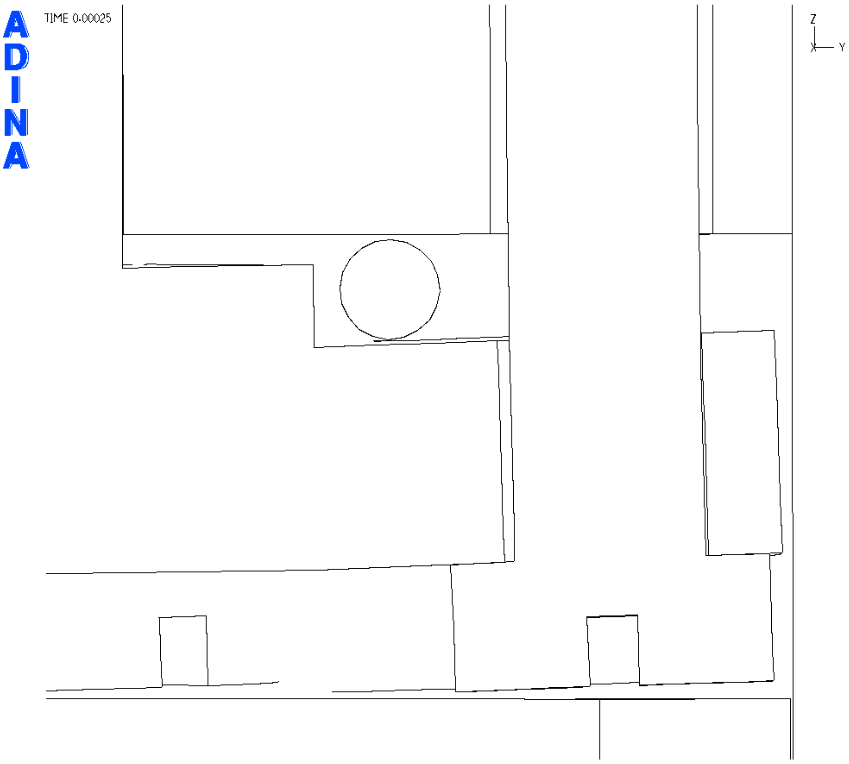

Fig. 3. The gasket release status detail

4. Conclusion

The use of the finite element fall test simulation allows the analysis of the behavior of the structure without the need for its manufacturing and experimental tests at the design stage of the transport container. Simulations allow identification of critical points of the structure and their modification in order to remove them. The use of fall simulations makes it possible to significantly reduce the cost of design and experimental verification.

Acknowledgements

This work has been supported by grant agency KEGA project No. 015ŽU-4/2017 and grant agency VEGA project No. 1/0073/19

References

[1] Bathe, K.J., Finite element procedures, New Persey, Prentice Hall, 1996.
[2] Bathe, K.J., Wilson, E.L., Numerical methods in finite element analysis, New Jersey, Prentice-Hill, 1976.
[3] Haftka, R.T., Gürdal, Z., Elements of structural optimization, Kluwer Academic Publisher, 1992.
[4] Jakubovičová, L., Kopas, P., Contribution to stress and residual strain analyse of the welded specimen, Transactions of the University of Košice, No. 3, Research reports from the universities of Košice, pp. 57-64.
[5] Kwon, Y.W., Bang, H., The finite element method using MATLAB, CRC Press University of Minnesota, 1996.
[6] Vaško, M., Ardeshir, G., Jakubovičová, L., Kopas, P., Effects of the loading rate on contact stresses of a roller bearing: A computational study, Fourth Serbian Congress on Theoretical and Applied Mechanics, Vrnjačka Banja, Serbia, 2013, pp. 547-553.