Proceedings of the 3rd International Conference on Nonlinear Dynamics
ND-KhPI2010
September 21-24, 2010, Kharkov, Ukraine

MODELING OF IMPACT DEFORMATION PROCESSES OF THE CERAMIC CONTAINER FOR RADIOACTIVE WASTE STORAGE

Sergey Yu. Sayenko
National Science Centre "Kharkiv Institute of Physics and Technology"
Kharkiv, Ukraine

Oleg K. Morachkovsky[1]
Denis V. Lavinsky
Vladimir N. Sobol
Yuriy M. Andreev
National Technical University "Kharkiv Polytechnic Institute",
Kharkiv, Ukraine

ABSTRACT

Results of researches in the field of designing containers for storage of radioactive materials are presented in the work. The purposes of researches include a development of an effective method of modeling static and dynamic deformation processes at shock impact on the ceramic container with radioactive materials at transportation. The next tasks have been solved: on the basis of the mathematical description of physic-mechanical processes of deformation of the complex design container has been chosen method of finite-element's for effective modeling of the stress-strain State of static and dynamic deformation processes in the containers made of ceramic elements; laws of deformation, estimations of durability and rigidity for designed containers have been received after calculations by means of the computer software; recommendations on perfection of a design of the container for maintenance of requirements to safety are given. Analysis Finite Element Method (FEM) has been conducted in the ANSYS system and results is presented.

INTRODUCTION

One of the most probable events radiating danger at storage of a radiating active waste is connected with threats of in a regular mode of storage and/or destructions at falling at transportation, or unforeseen shock-impact on containers for the purpose of their destruction. The container is a vessel for transportation and-or storages of radioactive materials. It serves several functions: provides chemical, mechanical, thermal and radiological protection, disseminates a heat of disintegration during processing, transportations and storage. Considering quantities of radioactive materials stored in Ukraine it is possible to draw a conclusion that the damage and scales of radiating pollution in case of degradation of capacities of storage and dispersion of radioactive substances can be considerable. Therefore, in Ukraine, as well as in many other countries now there is a question of decrease in radiating danger of existing crucial objects, such as storehouses of a radioactive waste. One way is working out of new protective materials and designs on their basis for creation of containers for storage of the radioactive materials, different the raised durability, in comparison with traditional concrete. Refusal of use of packing's container from concrete it is one of variants of achievement.

Ceramic material it is crystalline solid, usually has contains silicon dioxide (SiO_2) and other inorganic oxides, this material produce at high temperatures (800°C or above) and, usually, at elevated pressures. Among such materials most interesting materials it is the ceramic composites of system B_4C-ZrB_2 received by methods of hot pressing and reactionary hot pressing of powder components on the basis of boron, carbon, carbide of boron and oxides zirconium. Told above is cause expediency of continuation the spent researches at [2] and workings out of methods of modeling of behavior of ceramic containers on the basis of ZrO2 and B4C, and also their compositions under the influence of various dynamic loadings for rationalization design and structure of these new materials and containers.

[1] Corresponding author. Email morachko@kpi.kharkov.ua

1. PROBLEM STATEMENT

The package used to transport radioactive materials must be safe under normal and hypothetical accident conditions. These requirements for ceramic container design are verified through test or finite element (FE) analysis. Since the cost for FE analysis is less than one for test, the verification by FE analysis is mainly used. But FE analysis may show different results for the same problem due to several assumptions of models to simplify real states and modelling technique. This may have different results as FE-codes. In this paper, finite element analysis is carried out for the 5 meters free drop and the puncture condition of the hypothetical accident conditions using COSMOS and ANSYS. Energy and effective stress on each component are presented and compared between two FE codes where, the effective stress is designated the maximum von Mises stress on inner and outer walls of container. In this work, a detailed analysis of the drop problem the maximum allowable stress is performed considering several attitudes at impact in order to find which attitude results in maximum damage to the container, and which part of the container deformed severely. Once the failure mode of the hypothetical drop is defined, a full-scale drop on the container will be performed. The numerical results are compared and some analytical dimensional and physic-mechanical parameters are readjusted to obtain a better correlation between forthcoming natural test and analysis.

This paper presents the details of a simulated Programs of numerical researches conducted in support of an forthcoming natural experimental test program performed in order to prepare ceramic containers. The information regarding the waste package used in this calculation is based on the proposed designs presented by the drawings and sketches.

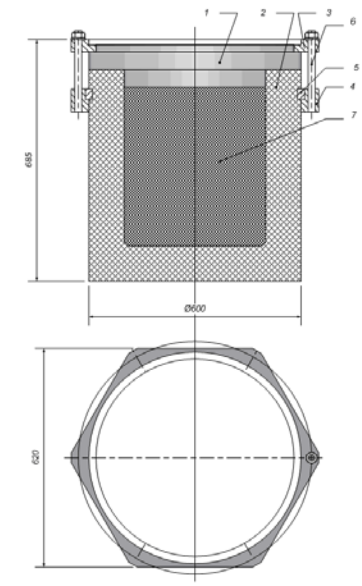

Fig. 1 The sketches of proposed designs waste package

The dimensions used in this calculation refer to the dimensions associated with design and the maximum dimensions. The bounding and maximum weights used.

Table 1 – Physic-mechanical properties of materials for the ceramic container details

Mechanical properties	**Steel 30Cr13**	**Ceramics ZrO_2**
Strength at compression $(\sigma_B)_c$, MPa		1400
Strength at a bend $(\sigma_B)_b$, MPa		750
The module of elasticity E, GPa	210	200
Poisson's ratio ν	0.3	0.28-0.36
Impact strength $a_н$, kg/sm^2		
Density ρ, g/$sм^3$	7.8	5.8-6
Factor of friction		0.2-0.3

№	Name	m, kg
1	Cover	149.544
2	Case	626.923
3	Flange	17.0683
4	Flange	19.13
5	Ring cutting	6.45
6	Hairpin	0.337
7	Radioactive material	141.37
	$\sum m$	962.5

Method of solution and numerical results are presented. A detailed conclusion and assumption are presented in publication [1,2].

2. METHOD OF SOLUTION

For identification of the maximum allowable stress for a pressured container made from the ceramic composites of system B_4C-ZrB_2, it is necessary to determine the stress components derived from pressure. For hermetic sealing preliminary compression of a cover by bolts is used a deformation of compression of a bolt is varied. The stress component derived from pressure contains a stress concentration factor which is dependent on pressure and hermetic sealing preliminary compression.

In this work, identification of the maximum allowable stress and stress concentration factor values is obtained by the Finite Element Method (FEM) analysis conducted in the ANSYS program is presented. A ceramic container is analyzed for a 5 m free drop to a rigid ground. The complete finite element model is built and analysis is then carried out using Program Package ANSYS. Results are again completely processed using SCA KIDIM in terms of time history plots of momentum and contact force; deformation of the container during impact; and stress/strain distribution in the container at different times.

The model assumes that all the materials are homogeneous and isotropic and that the mechanical properties are associated with an isothermal environment. The impact problem is an initial condition formulation where the velocity of the body at a time of impact is fixed at 13.2883 m/s which value is from an assumption of 5 m free-drop. The orientation of the velocity vector onto horizontal target surface defines the attitude of the body at the time of impact. The numerical analysis has been programmed to start 1msec before the time of impact, which time is required for the stabilization of initial condition.

Two distinct attitudes at impact have been considered in this study. The cases considered are: 1) Vertical impact on base assembly, 2) Oblique impact on bottom corner at angle of 30° from the horizontal.

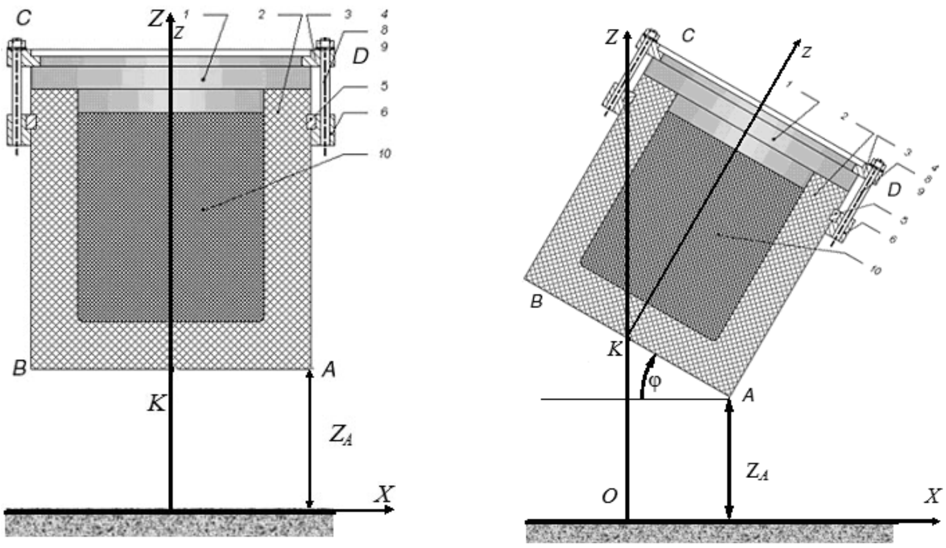

Fig. 2 The cases at impact: 1) Vertical impact on the base,
2) Oblique impact on bottom corner at angle of 30° from the horizontal

3. NUMERICAL RESULTS

For identification of the maximum allowable stress for a pressured container made from the ceramic composites of system B_4C-ZrB_2, it is necessary to determine the stress components derived from pressure. For hermetic sealing preliminary compression of a cover by bolts is used a deformation of compression of a bolt is varied. The stress component derived from pressure contains a stress concentration factor which is dependent on pressure and hermetic sealing preliminary compression.

In this work, identification of the maximum allowable stress and stress concentration factor values is obtained by the Finite Element Method (FEM) analysis conducted in the ANSYS program is presented.

FE representations of the container with radioactive materials with dimensional structure elements, weight and prepress differences are created and solved for analysis of drop events using ANSYS program. The numerical stress results are reviewed to determine the maximum response locations and magnitudes. The results of this calculation are evaluated for wall-averaged stress in tensities. The stress responses for the different FE representations are compared to each other to determine the sensitivity of the calculations to variations in the input parameters.

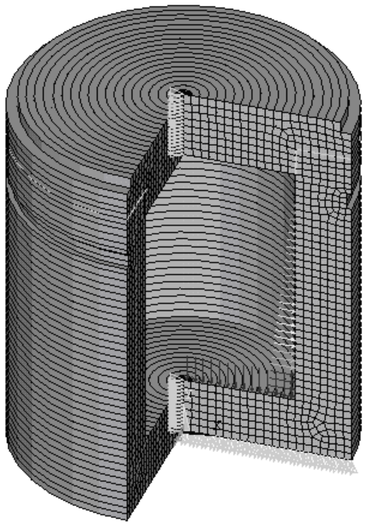

Fig. 3 Finite element model of the container

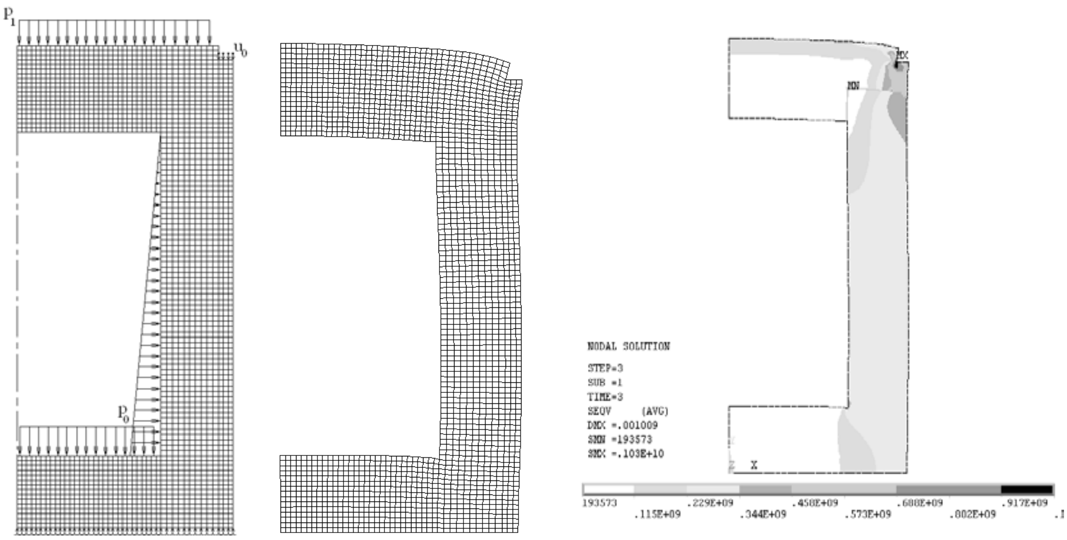

Fig. 4 Finite Element Model, Deflection Contour and Von Mises Stress at walls of container

Number of calculations for an estimation of value influence of a coupler on the strained condition and hermetic sealing preliminary compression of a cover by bolts is used. A deformation of compression of a bolt – U is varied. On Fig.5 (left figure) dependences of the maximal intensity of stress and the maximal first main stress from in a range from 0.01 mm up to 0.1 mm with step of 0.01 mm are resulted. On Fig.5 (right figure) dependences of the maximal intensity of stress and the maximal first main stress from deflection in a range from 0.1 mm up to 1 mm with step of 0.1 mm are resulted. Both in the first and in the second cases linear dependence is observed. And at size U=1 mm the maximal value of the first main stress makes 732 MPa (stretching) that is close to limiting stress. It is found that the maximum Von Mises stress in the bottom cover is about 1000 MPa, which is significantly lower than the ultimate stress intensity of 1400 MPa. The maximum contact pressure on lines AB and AC are equal 590 MPa.

An impact analysis of a ceramic storage container using Program Package ANSYS for pre- and post-processing has been illustrated. PP/ANSYS enables the response of the container system in terms of various parameters at different stages of impact to be studied in details. The information obtained is very useful for container design.

Next example is kinetostatic problem solving by using ANSYS program software. Ceramic container has been dropped vertically from height about 0.5 m as you can see on the Fig. 2. The acceleration values are calculated at the moment of container shock and equal to 21700 m/sec^2. Such data is used as inertia loads in ANSYS software. Compression deformation of the bolt (U) was taken equal to 1mm. Solution results have been done for such case taking to account equality of geometric, mechanical and physical parameters for container to previous example. The maximal values of stress intensity by von Mises and contact pressures are less to previous solution example and equal to 856 MPa and 488 MPa respectively. Hermetic properties of the ceramic container kept safety.

It is found that damage tends to localize in the vicinity of contact with stresses and strains decreasing rapidly toward remote areas. A stainless steel ring cutting can give the impact on the container effectively. Even though the impact limiter experiences severe damage, damage on the container itself is not observed as it remains essentially elastic.

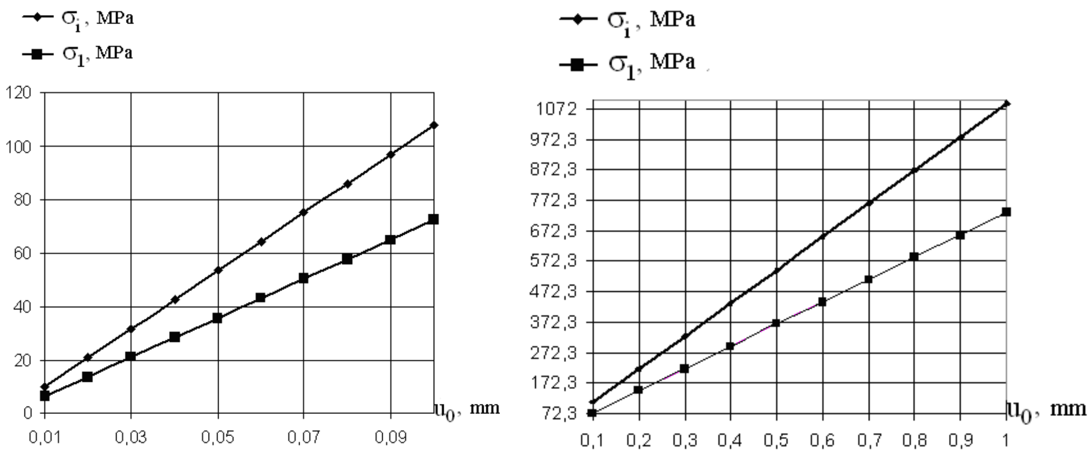

Fig. 5 Dependences of the maximal intensity and the first main stresses from U

CONCLUSIONS

In this work, identification of the maximum allowable stress and stress concentration factor values is obtained by the Finite Element Method (FEM) analysis conducted in the ANSYS program is presented.

Results of the impact analysis described in this paper show that complex structural deformation patterns associated with the impact problem could be predicted. The time and cost for the impact testing of Type C container with complex geometries are reduced considerably. In summary, the impact simulation provides the preparer of the approval of Type C container with a convenient tool that can be used in support to licensing efforts.

REFERENCES

[1] Sayenko S.Yu. Technological approaches for good safety at the reference with radioactive waste, *Power, Economy, Technologies, and Ecology*, Vol. 21, N.2, pp. 27-35, 2007 (in Russian).

[2] Nekljudov I.M., Morachkovsky O.K., Andreev Yu.M., Sayenko S.Yu., Biryukov O.V. Dynamic's Unsociable persons modeling on test for durability of containers-packing's for highly active waste of nuclear stations, *East-European Magazine of High Technologies*, Vol. 30, N.6/4, pp. 4–7, 2007 (in Russian).