Looking to the future
PATRAM 2010

FINITE ELEMENT MESH DESIGN OF A CYLINDRICAL CASK UNDER PUNCTURE DROP TEST CONDITIONS

Uwe Zencker **Mike Weber** **Frank Wille**

BAM Federal Institute for Materials Research and Testing
Berlin, Germany

ABSTRACT

Transport casks for radioactive materials have to withstand the 9 m drop test, 1 m puncture drop test and dynamic crush test with regard to the mechanical requirements according to the IAEA regulations. The safety assessment of the package can be carried out on the basis of experimental investigations with prototypes or models of appropriate scale, calculations, by reference to previous satisfactory safety demonstrations of a sufficiently similar nature or a combination of these methods. Computational methods are increasingly used for the assessment of mechanical test scenarios. However, it must be guaranteed that the calculation methods provide reliable results. Important quality assurance measures at BAM are given concerning the preparation, run and evaluation of a numerical analysis with reference to the appropriate guidelines.

Hence, a successful application of the finite element method requires a suitable mesh. An analysis of the 1 m puncture drop test using successively refined finite element meshes was performed to find an acceptable mesh size and to study the mesh convergence using explicit dynamic finite element codes. The finite element model of the cask structure and the puncture bar is described. At the beginning a coarse mesh was created. Then this mesh was refined in two steps. In each step the size of the elements was bisected. The deformation of the mesh and the stresses were evaluated dependent on the mesh size. Finally, the results were extrapolated to an infinite fine mesh or the continuous body, respectively. The uncertainty of the numerical solution due to the discretization of the continuous problem is given. A safety factor is discussed to account for the uncertainty. The calculation results are compared with experimental data from a puncture drop test with a half-scale model of a cylindrical cask.

This paper supports the convergence studies of the Task Group on Computational Modeling for Explicit Dynamics reporting to the ASME BPV Code Working Group on Design Methodology.

INTRODUCTION

The Federal Institute for Materials Research and Testing (BAM) assesses safety analysis reports within the approval process of transport and storage casks for radioactive materials. For transport packages BAM is part of the competent authority for the mechanical and thermal safety assessment in Germany. Safety analysis reports increasingly base completely or partly on numerical calculations. In most cases the finite element method (FEM) is used. BAM issued guidelines for the quality assurance at the preparation, checking and evaluation of numerical safety analysis reports [1]. Their main purpose is to provide a basis for state-of-the-art computing in cask analysis and to support complete safety analysis reports which clearly describe the calculation strategy, the underlying conditions and assumptions, and the calculation results.

The safety assessment of transport casks for radioactive materials is carried out by BAM on the basis of the IAEA regulations [2]. They are obligatory in Germany by national and international regulations for the transport of dangerous goods. The test conditions and accident scenarios within the approval procedure for the safety assessment of transport packages are defined there. In Germany further national test requirements for the interim storage and final disposal of spent fuel and radioactive waste result from site-specific accident analyses. From these numerous complex load scenarios we selected the easily understandable 1 m puncture drop test of a cask onto a steel puncture bar according to the IAEA regulations. Here it is only an example to have a well-defined load scenario for a mesh refinement study.

There are numerous regulations for safety relevant calculations, e.g. the ASME BPV Code [3] or in Germany the KTA rules [4]. Recommendations also exist to general aspects of the quality assurance of numerical calculations [5]. However, the running of reliable numerical calculations still requires much know-how of the computational engineer. In particular this applies to dynamic calculations. For example, there is no generally valid rule for the generation of a finite element (FE) mesh which is suitable for a numerically stable and sufficiently accurate dynamic finite element calculation.

An approximate solution does not necessarily have to be correct. Generally, a deviation of the approximate solution from the exact solution of the examined physical problem appears due to the spatial discretization of the components and simplifications or inaccuracies at the initial or boundary conditions, contact conditions and incomplete description of the material behavior of the numerous parts of the finite element model. Therefore, the found solution has to be validated. The present finite element model is validated by means of a recently tested half-scale cask model made of ductile cast iron which was extensively equipped with strain gauges and accelerometers.

QUALITY ASSURANCE MEASURES

Computational methods are increasingly used for the assessment of mechanical test scenarios. However, it must be guaranteed that the calculation methods provide reliable results. For that, the developed numerical model must be suitable for the description of the examined load scenario and the used calculation code must work correctly. Hence, quality assurance measures are the basis of any reliable numerical analysis and also of a mesh refinement study.

The suitability of the finite element code may be shown among others by reference to publicly available studies. At least two appropriate examples should be provided to ensure the reliability of the calculation code with regard to the specific problem [1]. These examples may either be prepared by the applicant or be taken from literature. However, it must be guaranteed that they are validated. The calculation algorithms of the FE code are primarily verified by means of benchmark tests, e.g. on the basis of analytically solvable physical problems [6]. Comparative calculations with different codes using the same finite element model also belong to benchmarking according to the approval of the code for a special application (cf. for example SANDIA report “Sample Problem Manual for Benchmarking of Cask Analysis Codes” [7]). This will mainly supply knowledge concerning the suitability of the corresponding FE code for handling the investigated technical problem. However, this is not yet a validation of the used FE model.

Due to changes of the source code (resulting from improvements or from the introduction of new features) and to the necessary adaptation to new hardware or operating system software, FE codes must be subject to permanent quality control [8]. Usually this is assured by the suppliers of commercial FE software. If the supplier of FE software does not work according to an approved quality assurance system, or if the applicant uses software available as source code, quality assurance measures must be implemented and documented based e.g. on [5]. This applies both for the FE code itself as well as for the used pre and post processors, i.e. software for the FE model generation and processing of calculation results.

PUNCTURE DROP TEST

The BAM procedure for the safety assessment of transport and storage casks for radioactive materials includes both numerical and experimental analyses. Recently an extensive drop test series was carried out with a half-scale test cask which has been used mainly for the verification of numerical calculation methods [9]. The tests took place at the drop test facility of the BAM Test site Technical Safety (TTS) near Berlin. The experimental results from the horizontal 1 m drop of the test cask with the center of the cask wall onto a puncture bar can be used below for a verification of the derived FE mesh.

Figure 1. Cask before puncture drop test

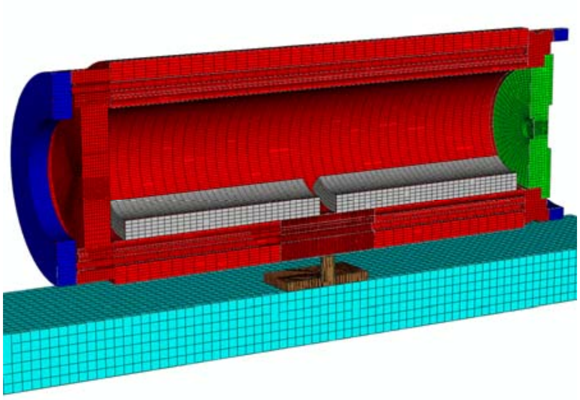


Figure 2. Complete finite element mesh

Figure 1 shows the cask, cooled down to a temperature of -40 °C, before the drop test. It is a cylindrical cask made of ductile cast iron. The cask wall contains two series of boreholes for moderator material (not filled during the test) located near the shaft. The test scenario meets the requirements of the IAEA regulations [2]. This test represents a highly dynamic local load for the cask. The measurement data needed for the verification of the finite element model were recorded with an extensive instrumentation consisting of strain gauges and accelerometers.

FINITE ELEMENT MODEL

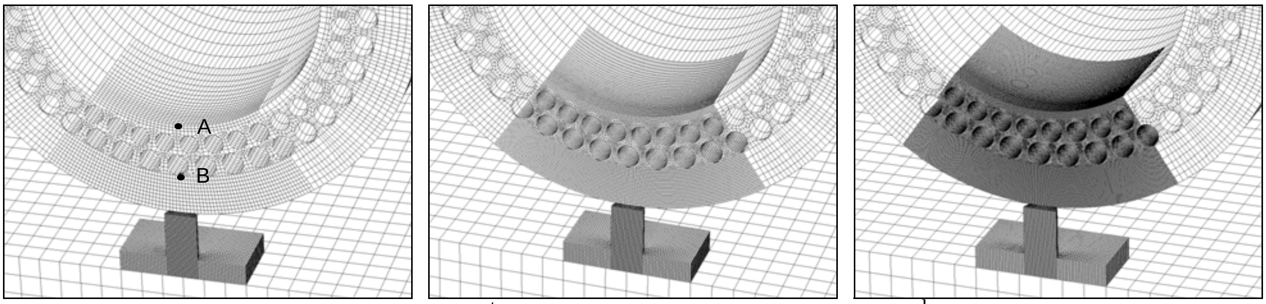
Suitable material models are essential for a realistic simulation of the drop test. The ductile cast iron of the cask body is described with an elastic-viscoplastic constitutive law based on measured dynamic flow curves. The simulation results are substantially determined by the material behavior of the puncture bar and the friction between the surfaces of puncture bar and cask body. The cask is stiffer than the puncture bar. The mild steel of the puncture bar is defined in the IAEA regulations [2] and the corresponding advisory material. Measured dynamic flow curves under pressure were available for the steel used in compliance with the regulations. First the modeling of the puncture bar and of the numerical contact formulation was tested with a very simplified finite element model. In this model the cask could be reduced to a rigid body in good approximation because up to 98 % of the total energy is converted to plastic deformation of the puncture bar. The mass of a hollow cylinder with an outside diameter corresponding to the cask body was adapted to the mass of the test cask by variation of the density. The deviation between calculated and measured remaining height of the puncture bar gives a measure of the quality of the FE model. As expected, the friction coefficient for the contact between cask body and puncture bar determines the calculated typical barrel shape of the bar.

Then a more realistic model of the load scenario was developed using the verified submodel of the puncture bar. The cask hits with the prescribed impact velocity to the puncture bar which is linked with the IAEA target of the test site. The complete model includes the geometry of the cask in detail including the lid system (Fig. 2). The finite element mesh of the cask body was refined near the contact area. The cooling fins in the contact area were missing at the test. The mass of the remaining cooling fins and of neglected parts of the cask was considered by an increased mass density. Dummy masses for the lid-side and bottom-side impact limiters as well as contents guarantee a realistic mass distribution along the horizontal cask axis. The Young's modulus of the simplified contents model was chosen very small to avoid an unintended increase of cask stiffness.

All calculations were done with the dynamic finite element code ABAQUS/Explicit™ [10] but the findings are not limited to this software.

MESH REFINEMENT STUDY

The convergence of the numerical calculation results for the puncture drop test was checked by variation of the finite element mesh near the contact area between puncture bar and cask body. At the beginning a coarse mesh was created (Fig. 3a). Then this mesh was refined in two steps (Fig. 3b and 3c). In each step the size of the elements was bisected. The deformation of the mesh and the stresses were evaluated in dependence on the element size. The lid system and the dummy masses were omitted in the calculations for the convergence study.

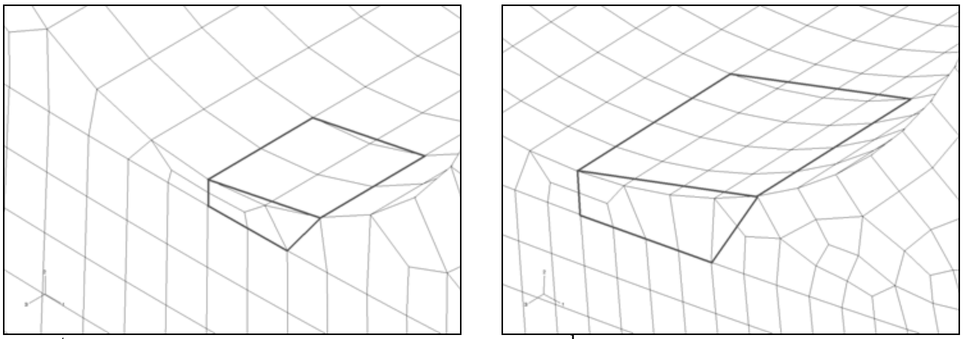


a) Coarse mesh b) 1st refinement c) 2nd refinement

Figure 3. Finite element mesh with refined submodel

Two domains are inspected: the intersection point of the vertical puncture bar axis with the inner surface of the cylindrical cask shaft (position A) and the intersection point of the vertical puncture bar axis with the outmost element of the outer series of the moderator boreholes (position B). The surface of the shaft is of special interest because the strains can be measured there with strain gauges very well. On the other hand, high stresses are expected near the moderator boreholes due to the geometrical conditions. The outer series of the boreholes, and especially a surface point close to the puncture bar, was selected with regard to a separate fracture mechanical analysis not of interest here. In both positions the element closest to the discussed intersection point was investigated to find a local value on the body surface. The generation of average values was not necessary because of only very small deviations between the results of the four elements symmetrically arranged around this point.

The elements are almost well-shaped cubes in the region of the inner shaft, what is best with regard to the numerical accuracy of the spatial discretization. All edges of the elements were bisected at every mesh refinement step. First-order (linear) interpolation elements with hourglass control were used. Therefore stresses and strains are constant within the finite element. As a result of the mesh refinement, the spatial course of stresses and strains in the structure is described more exactly by the larger number of smaller elements. The quantities were evaluated directly at the integration point of the element.



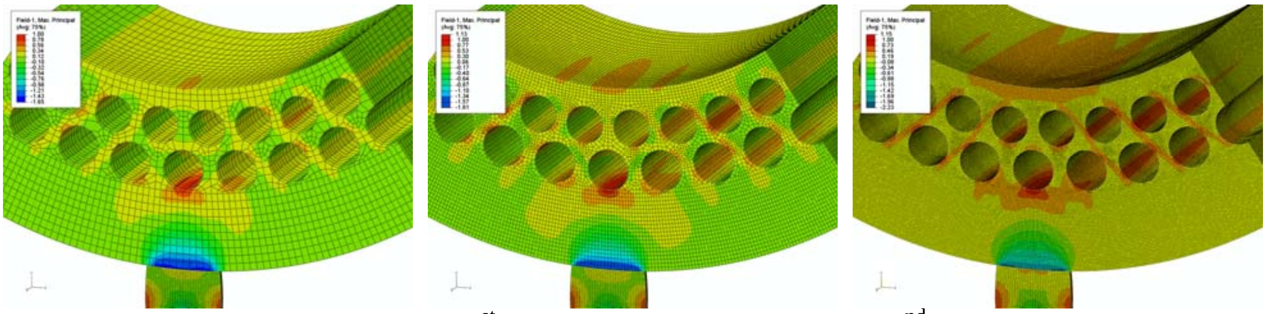
a) 1st refinement b) 2nd refinement

Figure 4. Details of finite element mesh

However, the mesh consists not only of ideal cubic elements in most parts of a real cask. An example is the local mesh nearby the moderator boreholes. Bisecting the edges of the elements, the complicated geometrical shapes also must be described well. The mesh refinement strategy is illustrated in Fig. 4. The starting element of the corresponding coarse mesh is marked. While at the first refinement step the element edges were bisected in all three directions, the element edges were bisected only in two directions at the second refinement step to reduce the total number of finite elements of the model.

NUMERICAL RESULTS

The mesh of the cask was refined step-by-step only near the puncture bar to reduce the computational effort. A refined submodel was pasted into the coarse mesh by means of so-called tied contact conditions (adhesive contact). From the fringe plots all three FE meshes seem to be suitable (Fig. 5). However, the fringe plots alone do not suffice for the assessment of the quality of the FE mesh.



a) Coarse mesh b) 1st refinement c) 2nd refinement

Figure 5. Normalized first principal stress at time of maximum pressure

Near the shaft the behavior of the cask material is elastic. Localized plastic deformation occurs only in the vicinity of boreholes. Figure 6 shows the first principal stress at position A at the shaft for the impact duration. The stress is normalized on the maximum of the stress from the calculation with the coarse mesh. At these investigations the cask model consisted only of the cask body to reduce the computational effort. The impact duration is about 18 ms with this simplified cask model. The stresses in the cask body increase if the mesh is refined. The first refinement step leads to a noticeable stress increase while the further stress increase at the second refinement step is smaller. It can be seen that the solution is numerically stable and approaches a limit curve.

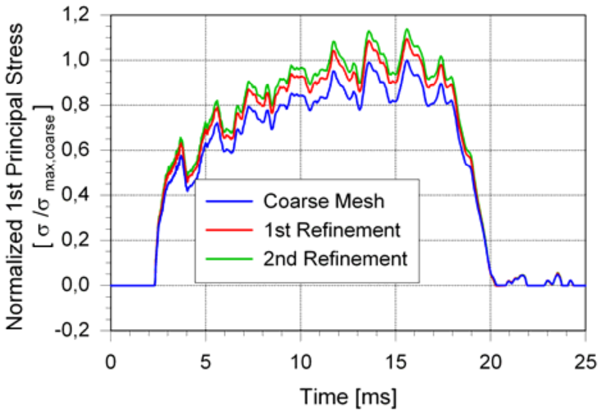


Figure 6. History of normalized first principal stress at shaft

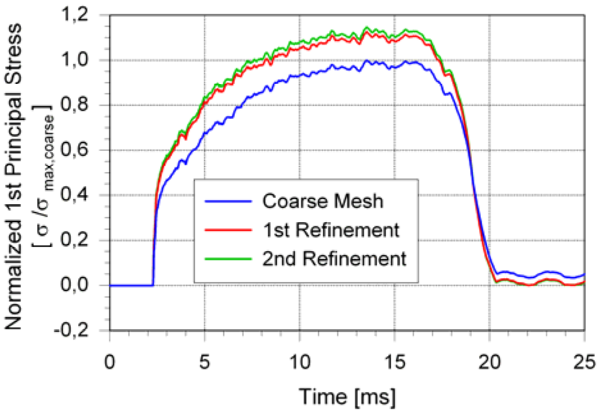


Figure 7. History of normalized first principal stress at borehole

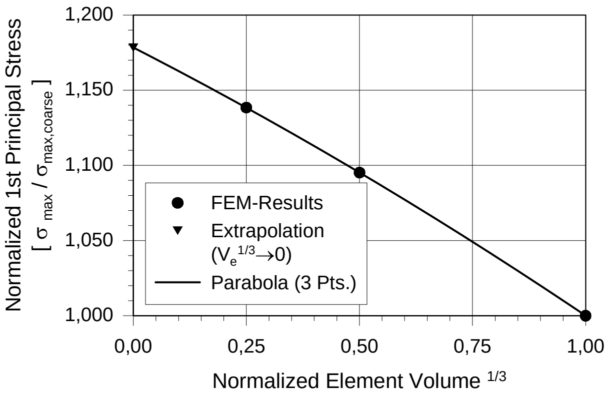


Figure 8. Convergence of normalized first principal stress at shaft

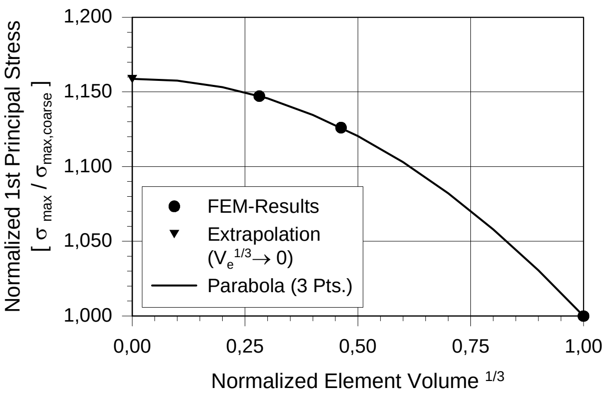


Figure 9. Convergence of normalized first principal stress at borehole

The mean element size ($V_e^{1/3}$) with the element volume V_e is used for the characterization of the FE mesh to be able to take into account non-cubic element shapes in real meshes. The mean element size corresponds exactly to the real element size only for a cube-shaped element. Figure 8 shows an almost linear curve for the normalized first principal stress at the shaft in dependence on the mean element size. Merely three supporting points are available. A parabola is one possibility among others for an interpolation curve to illustrate the trend. The extrapolation to a vanishing element size corresponds to the infinite fine mesh or the continuous body respectively. The limit stress is about 18 % higher in comparison with the coarse mesh if a parabola is used for a data fit.

The effect of the mesh refinement is even more distinctive at the moderator borehole. Figure 7 shows the normalized first principal stress at the outer moderator borehole series at the position closest to the puncture bar (position B). Already the first refinement step leads to a distinct improvement in the numerical solution which can be hardly improved by the used mesh scheme at the second refinement step.

The normalized first principal stress at the borehole over the mean element size (Fig. 9) seems to converge to a mesh-independent value. Nevertheless, the stress is 16 % higher at extrapolation to the infinite fine mesh related to the coarse mesh and with reference to a parabola fit. The increase of the von Mises stress is of the same order while all three principal stresses play a role.

Of course, mesh refinement is limited by computer resources. However, the investigations demonstrate that coarse FE meshes might produce uncertainties in calculated stresses. So a safety factor of at least 1.2 should be used at the assessment of stresses calculated with the coarse mesh because of the limited spatial discretization. Therefore, domains with critical stresses have to be investigated additionally by means of submodels.

EXPERIMENTAL VERIFICATION

The complete model (Fig. 2) was used for the comparison with the experimental data. The mesh density corresponds to the FE mesh of the first refinement step from the convergence study with a slightly simplified model as described above. The verification of the FE model was carried out with regard to a plausible kinematic behavior and comprehensible energy quantities as well as by comparison of calculated with measured strains. Special attention was paid to the regions of high stress in the cask body. Although far away from the contact area near the puncture bar and with a considerably lower stress level, the measuring points nearby the lid system and the cask bottom also have to be included in a complete assessment of the total model.

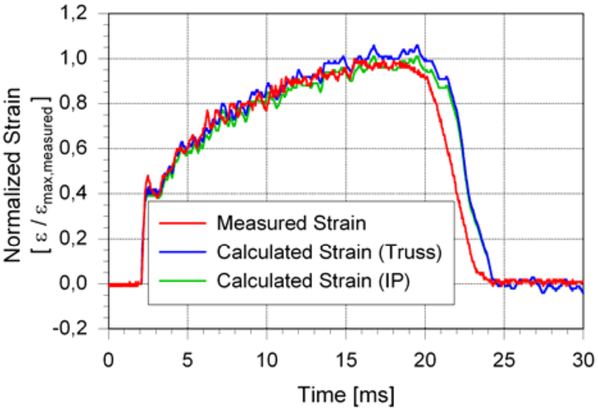


Figure 10. Comparison of measured and calculated strains (complete model)

At first the stresses and strains at the integration points (IP) of the elements were evaluated. In addition, one-dimensional truss elements were linked with surface nodes at the position of selected strain gauges. They allow a direct measure of local strains on the cask surface. Figure 10 shows the comparison of measured and calculated strains at position A on the surface of the shaft. The maximum values and the qualitative behavior of the curves agree well. The impact duration is slightly longer than in Fig. 6 and 7 since the complete cask model was used here for the verification. The differences between the measured and calculated energies are acceptable small. The curves for the calculated strain at the integration point of the assigned element and in the corresponding truss element are insignificantly different. The assessment of other positions on the shaft surface leads to similar results.

CONCLUSIONS

The puncture drop test is well suited to the generation and verification of a dynamic finite element model because of the local load and a comparatively simple dynamic contact problem. The essential prerequisites for a realistic simulation of the puncture drop test are a good knowledge of the material behavior of cask body and puncture bar, a sufficiently fine finite element mesh and suitable contact (i.e. friction) parameters. Available test data allow a validation of the model.

The convergence behavior of the finite element mesh was examined with a step-by-step refinement of the mesh. The first principal stress was analyzed inside the cask structure far away from the puncture bar as well as close to the contact area between cask body and puncture bar. It could be shown for the considered mesh that the stress converges to a mesh-independent value for the continuous body which is less than 20 % above the stress calculated in the coarse mesh. However, this does not mean that a safety factor of 1.2 in other numerical simulations would always suffice. The given statements are mesh-dependent and hence valid only for the presented meshes. Finally, the cask model could be validated by experimental data from a puncture drop test.

Even simple load scenarios possibly require very fine meshes with many elements. Complex load scenarios often cannot be calculated in total with such optimal meshes. In these cases a submodel technique might be used as presented. However, there are no obvious rules about the number of elements over a given wall thickness of a cask or the element size. This always depends on the given situation. Here a manually made finite element mesh was examined to clarify the human factor. The practical experience of the engineer is of decisive importance at manual meshing.

REFERENCES

[1] Bundesanstalt für Materialforschung und -prüfung (BAM): 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, Berlin, Germany, 2003. http://www.bam.de/pdf/service/amtl_mitteilungen/gefahrgutrecht/regeln/ggr-008deu.pdf

[2] International Atomic Energy Agency (IAEA): Regulations for the Safe Transport of Radioactive Material, Safety Standards Series No. TS-R-1, Vienna, Austria, 2009.

[3] American Society of Mechanical Engineers (ASME): Boiler and Pressure Vessel Code, New York, NY, 2007.

[4] Nuclear Safety Standards Commission (KTA): Components of the Reactor Coolant Pressure Boundary of Light Water Reactors, Part 2: Design and Analysis, Safety Standard KTA 3201.2 (06/96) (incl. rectification from BAnz 129, 13.07.00), Salzgitter, Germany, 2000. http://www.kta-gs.de/e/standards/3200/3201_2e.pdf

[5] NAFEMS Quality Standard Supplement (QSS) 001, Engineering Simulation – Quality Management Systems – Requirements, ed. J.M. Smith, NAFEMS Ltd., Glasgow, UK, 2007.

[6] Davies, G. A. O.: Background to Benchmarks, NAFEMS Ltd., Glasgow, UK, 1993.

[7] Sandia National Laboratories: Sample Problem Manual for Benchmarking of Cask Analysis Codes, Report SAND88-0190.TTC-0780.UC-71, Albuquerque, NM, 1988.

[8] International Atomic Energy Agency (IAEA): Quality Assurance for Software Important to Safety, Technical Reports Series No. 397, Vienna, Austria, 2000.

[9] Quercetti, T., Müller, K., Schubert, S.: Comparison of Experimental Results from Drop Testing of a Spent Fuel Package Design using a Full-scale Prototype Model and a Reduced-scale Model. Packaging, Transport, Storage & Security of Radioactive Material, 19 (4) 2008, pp. 197-202.

[10] ABAQUS Version 6.8, Dassault Systèmes Simulia Corp., Providence, RI, 2008.