Sustainable Power: Nuclear Energy Companies

Content

Overview: Leveraging realistic simulation for integrated plant engineering 3

New design of a pressure vessel subjected to blast loads 4
CER

Account of modal damping instead rayleigh one in floor response spectra analysis in civil structures of nuclear power plants (NPP) under aircraft crash 13
JSC Atomenergo proekt

Identification of ductile damage parameters 21
Czech Technical University in Prague, Faculty of Mechanical Engineering
SKODA – JS
Comtes FHT

Abaqus/Standard simulation of coupled multi-physics processes at an underground nuclear waste disposal site 32
Abaqus Technology Brief

Failure of a prestressed concrete containment vessel 35
Abaqus Technology Brief

The combined method of dynamic analysis of the “soil-structure” systems at aircraft crash onto civil structure of NPP 39
JSC Atomenergo proekt

Stress and fatigue analyses of a PWR reactor core barrel components 53
TÜV SÜD ET

Realistic simulation assists nuclear power plant certification 61
TÜV SÜD ET
Leveraging Realistic Simulation for Integrated Plant Engineering

**ACHIEVING SAFETY, COMPLIANCE, AND PERFORMANCE WITH “NO DEFECT” PLANT ENGINEERING**

“No defect” plant engineering means that your plant design and the delivered facility meet all project requirements. These large projects involve highly complex multidisciplinary engineering, procurement, and construction. They must conform to stringent regulatory, environmental, and compliance standards on both local and global levels, while meeting initial project expectations and final operational performance.

Integrated Plant Engineering 3DEXPERIENCE® solutions enables organizations to create better project control, minimize risks, and reduce development costs through optimal execution of project development processes.

**VIRTUAL TESTING OF PLANT SAFETY AND PERFORMANCE**

In addition to meeting initial requirements, the plant must operate reliably and safely. Integrated Plant Engineering advanced simulation capabilities enable companies to accurately model complex real-world behaviors, such as strength and deformation in large structures, pipes, and equipment; and the impact of thermal loads, vibrations, fractures, and failures.

Volume 2 of this eBook focuses on the nuclear power generation segment. Inside you’ll find case studies, papers, and tech briefs on how leading EPC companies, owner operator (OO) engineering departments, and specialized academic institutions are leveraging realistic simulation applications to achieve expected levels of operating performance for complex facilities, while meeting high safety and environmental standards.

Realistic Simulation, as part of the Integrated Plant Engineering industry solution, helps to reduce development time and cost, improve quality and power innovation.
New design of a pressure vessel subjected to blast loads

L. Cousin, P. Evrard (CEA, DAM, DIF, F91297 Arpajon, France)

Abstract: A new design of containment vessel has been proposed to conduct confined detonation experiments. In order to guarantee the confinement of the detonation products, the risk expressed as a probability of failure has to be quantified. This quantification is done using probabilistic analyses which require data from experiments and simulations in order to be sustained. When a blast stresses a spherical vessel, many different mechanical phenomena appear and have to be studied using adapted models. A blast leads to reflected waves in the structure. A numerical chaining is used to access the vessel dynamic structural response. This chaining consists in linking CATIA V5 with Abaqus/CAE to realize the analysis models and in linking our hydrodynamic code to Abaqus/Explicit in a weak coupling: high speed hydrodynamic simulations provide transient pressures which are used to act on the inner shell of the vessel. Two mechanical phenomena and their associated finite-elements models are of interest. First, we focus on the nominal model used to study the dynamic response of the vessel itself, underlining the most important components (internal furniture, material laws, type of mesh...) to be taken into account. The spherical vessel has five ports. Each of them has a cover bolted to the vessel. Then, we review a model the aim of which is to understand the bolts behavior when the vessel is subjected to high dynamic stresses. This model consists in a first implicit analysis followed by an explicit analysis which continually takes into account the bolts pre-tension loads. Experiments were performed in order to validate this weak coupling. We end up showing the good agreement between experimental and numerical results.

Keywords: Bolt Loading, Connectors, Constitutive Model, Coupled Analysis, Dynamics, Experimental Verification, Explosive, Impact, Safety, Vibration.

INTRODUCTION

The CEA has developed a new design of containment vessel for carrying out confined detonation experiments. This paper briefly presents the confinement vessel used for experiments and its associated instruments which allows understanding its behavior when submitted to blast loads. This behavior of the vessel has to be simulated because of the need existing in safety demonstration: to guarantee the confinement of the detonation products, the risk expressed as a probability of failure has to be quantified. This quantification is done using probabilistic analyses which require data from experiments and simulations in order to be sustained. The second point of the paper focuses on the numerical chaining used to master the blast effects on the vessel: this chaining goes from the Design Office, working with CATIA V5, to the Simulations Office, in which Abaqus models are prepared. Abaqus/Explicit is also linked in a weak coupling with our hydrodynamic code which provides information to load the inner shell of the vessel. The third point then describes the nominal Abaqus/Explicit model used to study the response of the vessel when submitted to a transient impulse loading. Results from the chaining are compared to experimental data. The fourth and last part consists in a description of another Abaqus model used to understand the behavior of the bolts which link five covers to the vessel body. The model consists in a first Abaqus/Standard analysis followed by an Abaqus/Explicit analysis using the *IMPORT functionality. A different bolts modeling is proposed because of the limitations of the *Pre-tension Section functionality only available in Abaqus/Standard, allowing for the bolts pre-tension loads to be taken into account in the dynamic simulation.

1. CONTAINMENT VESSEL CONFIGURATION

The containment vessel is presented in Figure 1. It consists of a steel spherical body with five ports each equipped with a cover:

- a steel cover at the top, used for general access,
- four aluminum side covers, used for diagnostics.

Each cover is connected to the vessel body by two concentric circles of bolts and three independent seals. The vessel is connected to a steel frame by its four side ports.
The vessel is monitored during each experiment in the same way:

- several strain gauges are attached to the outer skin vessel, at strategic locations and at the center of the five covers,
- for each cover, a gauge is attached to a bolt belonging to the inner circle of bolts and another gauge is attached to a bolt belonging to the outer circle of bolts.

2. NUMERICAL CHAINING

We use a numerical chaining (Figure 2) which links:

- the Design Office to the Simulations Office which uses Abaqus (Figure 2, path (1))
- and also the Simulations Office which uses our hydrodynamic code to our Simulations Office which uses Abaqus (Figure 2, path (2)).

2.1 From Design Office to Simulations Office

The first link enables to go from the geometric product assembly done with CATIA V5 in the Design Office to the finite elements model done in the Simulations Office using Abaqus/CAE (Figure 2, path (1)). A first VB script tool is used to put in the user’s folder all native CATIA V5 files (CATParts and CATProducts) concerning the model as well as materials data associated with the parts. The materials data consist in a ‘.lib’ file, an input file and a file giving the links between materials and their associated parts in the CATIA model. Materials are defined in a complete manner in the CATIA library: mechanical characteristics defining thermal behavior.
as well as dynamic behavior are known for most materials. Once the user’s folder contains all these data, Abaqus/CAE is opened and the user accesses a plug-in allowed for the CATIA V5 and the materials data to be loaded in. Once the file to import is chosen:

- CATIA V5 geometries are imported in Abaqus/CAE,
- Imported parts are renamed following definitions names existing in the CATIA model,
- Instances are created automatically in Abaqus/CAE according to the imported parts,
- Materials data existing in the folder are imported and materials are assigned to the suitable parts,
- A table ‘material/behavior’ is available for the user: in this table one can choose the most appropriated behavior for the analysis he wants to lead (characteristics of a thermo-mechanic behavior, characteristics of a dynamic behavior type Johnson-Cook plasticity law...)

This gateway is useful to guarantee the quality of our Abaqus models: users are sure to work with the appropriated geometries and materials properties.

2.2 Weak coupling between a home-made hydrodynamic code and Abaqus/Explicit

The second link (Figure 2, path (2)) consists in a weak computational coupling between our hydrodynamic code and Abaqus/Explicit. In a mechanical way, the vessel is submitted to a blast load. This blast load is computed using a specific hydrodynamic code developed by the CEA. The code simulates explosive detonation, internal gas expansion and shock waves propagation in 2-D Eulerian analysis models. A characteristic load function of an explosive detonation inside the vessel is presented in Figure 3 (a): a transient impulse load can be observed, due to the shock wave, followed by a long term quasi-static load caused by the evolution and the heating of gaseous detonation products. Because of furniture inside the vessel, the pressure field versus time is not homogeneous on the inner shell. The pressure field must be computed at different points onto the inner shell in order to take into account the heterogeneous dynamic load and must be computed for a sufficient duration to cover the several reverberations inside the vessel. The way these pressure data are used in Abaqus is explained in paragraph 3.1.

![Figure 3](https://example.com/image3.png)

**Figure 3**: (a) Typical pressure-time evolution; (b) Example of an 2-D Eulerian hydrodynamic model

3. ABAQUS/EXPLICIT ANALYSIS MODEL IN ORDER TO STUDY THE VESSEL RESPONSE

3.1 Constitutive Abaqus/Explicit analysis model

**Working assumptions**

The modeling of the vessel presented in Figure 1 is accomplished using a half-structure shown in Figure 4. Furniture exists rounding the explosive charge which is not represented here but taken into account in the modeling.
Boundary conditions

This half-vessel is fixed in the Y-direction by two AXIAL connectors type which have an elasticity modulus of one. Symmetry plane conditions are introduced for the nodes belonging to the cross-section plane (Figure 4).

Mesh

A 3D continuum elements mesh is applied. The analysis model contains 1,5 million 7 millimeter linear hexahedral elements C3D8R (Figure 5).

Loads

Furniture is driven using speeds computed from the 2D-Eulerian hydrodynamic analysis. The speed data are used through a predefined field defined in the Abaqus/CAE Load Module. The resulting keyword is:

*Initial Conditions, type=VELOCITY

Source: SIMULIA Community Conference, 2015
In the same time, all the pressure-time fields are used to load the inner shell of the vessel. We define one amplitude curve per pressure-time field. These fields are applied by slices which are defined creating:

- a surface: the inner shell,
- analytical fields to define the altitude of each slice in the Y-direction. This is done in the Abaqus/CAE Load Module through a table presented in Figure 6 (a), accessible from the Tools/Analytical Field menu,
- a load, type ‘pressure’ which allows to link the appropriated analytical field, surface and amplitude curve. The good definition of the slices can be monitored in the Visualization Module (Figure 6 (b)), loading the model of interest.

This functionality allows us to load the vessel without introducing any partitions of the structure which are a problem to keep a regular mesh.

![Figure 6](image)

**Figure 6**: (a) Definition of an analytical field describing a slice named ‘RF_carlo_pression_S10’; (b) Verification of the definition of the slice in the Visualization Module

**Materials**

The high strength steel vessel behavior, the steel top cover behavior and the aluminum side covers behavior are supposed to be elastic perfectly plastic. The shields behavior is supposed to follow a Johnson-Cook law.

The whole steel furniture behaviors are considered to be elastic perfectly plastic, coupled with a shear failure criterion to take into account their failure when impacting the inner shell of the vessel.

**Interactions**

A general frictionless contact is used in the model between the furniture and the vessel inner shell. In order to take into account impacts between the furniture, failed or not, and the vessel, we create different surfaces:

- an exterior surface on the inner shell of the vessel, named ‘exterior_vessel_inner_shell’
- an exterior surface on the whole furniture, named ‘exterior_furniture’
- interior surfaces in the whole furniture in order to take into account contacts existing between the future failed furniture and the inner shell vessel. These surfaces are named ‘interior_furniture’

Exterior surfaces are created on geometric entities, in a classical way in Abaqus/CAE, using the default selected tool. In order to create interior surfaces, the user has to define surfaces type ‘Mesh’ and change the default selected tool with ‘Select From interior Entities’ (Figure 7).

![Figure 7](image)

**Figure 7**: Modification of the default selected tool to define interior surfaces
In the *Contact keyword, the following relations between surfaces must appear:

*Contact, op=NEW

*Contact Inclusions

Interior_furniture ,
Exterior_furniture,

Interior_furniture , Exterior_furniture

Interior_furniture, Exterior_vessel_inner_shell

Exterior_furniture ,Exterior_vessel_inner_shell

**Outputs**

Some of the elements belong to nodes sets according to the gauges attached to the outer shell of the vessel during experiments. The strain tensor is the output of interest. We assign a specific spherical material orientation (R, T, Z) on the vessel body so that the strain tensor is computed in the same orientation as the gauges orientation. This is done to be able to compare experimental and numerical data.

### 3.2 Analysis model validation

Several experiments were carried out using different explosive charges. The vessel is always monitored with gauges which locations are the same as the ones presented with the red dots in Figure 8. Experimental and numerical maximum Von Mises stresses values are compared for each gauge. One of the comparisons made is presented in Figure 8. Good agreement can be observed excepted at the south pole and the north pole (top cover) of the vessel: the finite elements model underestimates the experimental data at the poles. These phenomena can be observed for each experiment comparison. Some more work is led at the moment in order to improve this fact.

![Figure 8: Maximum Von Mises stresses - Experimental measures and numerical results comparison](image)

**4. AN ANALYSIS MODEL FOR THE BOLTS BEHAVIOR STUDY**

#### 4.1 Constitutive Abaqus analysis model

**Working assumptions**

The Abaqus model used in order to understand the behavior of the bolts when the vessel is submitted to a blast load is presented in Figure 9. It consists in the same half-model as before, with the same symmetry plane and boundary conditions. No furniture is taken into account. Bolts used to assembly the covers to the vessel body are modeled. The top cover is assembled by two concentric circles of M36 bolts. The side covers are assembled by two concentric circles of M24 bolts.
Interactions
The two shields are linked to the vessel using the *TIE* functionality. Contacts defined between the covers and the vessel body take into account a friction of 0.1.

Mesh
All the parts are meshed with C3D8R linear hexahedral elements. The analysis model contains 1.5 million 7 millimeter linear hexahedral elements C3D8R.

Bolts modeling

General concept
The M36 bolts (top cover) and the M24 bolts (side covers) are modeled using beam elements (in red on Figure 10) associated with connectors in order to firstly impose the pre-tension load in the bolt and secondly monitor the existing axial force in the bolt when the vessel is submitted to a dynamic solicitation due to the blast.

The connector defines a kinetic relation between two points. One of these nodes is the end of the head bolt side beam. The second node is named differently from the first one but has the same location: the connector element length is null. Figure 10 shows a zoom of the modeling: the connector, built between node 1 and node 2, is connected in series with the beam (in red). The second node of the connector is the kinematic coupling node reference. This coupling is representative of the head bolt leaning on the

Source: SIMULIA Community Conference, 2015
covers: the slave nodes underlying surface is defined according to the diameter under the head bolt. A kinematic coupling is also defined at the other end of the beam in order to take the bolt thread grip height into account: the slave nodes underlying surface is defined according to the real bolt thread grip height.

**Beam elements characteristics**

Linear beam elements, B31, are used to model the bolts. The behavior of these steel bolts is supposed to be elastic. The diameter, \( d \), and the length, \( L \), of the beam are computed in order to obtain the suppleness in tension and flexion of the M24 bolts and the M36 bolts respectively. These assembly parameters are computed using the COBRA software which follows the VDI2230 [1].

**Connector elements characteristics**

‘Cartesian + Cartan’ connectors type are used. They are associated with local cylindrical orientation (\( R, T, Z \)) such as defined in Figure 9. All translation and rotation degrees of freedom are activated (\( U1, U2, U3 + UR1, UR2, UR3 \)). The designation ‘1’ is equivalent to the local \( R \)-direction, the designation ‘2’ is equivalent to the local \( T \)-direction, and the designation ‘3’ is equivalent to the local \( Z \)-direction. The connectors are used to impose the pre-tension load in the bolt and secondly to know the axial force history when the vessel is submitted to a dynamic solicitation due to the blast so that their stiffness is infinite.

**Sequence of analysis**

The simulation is conducted in two times:

- A first implicit simulation is accomplished using Abaqus/Standard in which the pre-tension load is imposed. This time is a two steps analysis:
  - the first step is a step for contact initiation. The degrees of freedom of the connectors linked with beams on the top cover are all locked. The degrees of freedom of connectors linked with the beams of the side covers are all locked excepted the translation degree of freedom in the beams axis direction. A very small displacement is given to the side covers in this beam axis direction,
  - in the second step, the imposed displacements are off and the connectors linked with the beams of the top cover are relaxed in the beams axis direction. The pre-tension load is then passed through the whole bolts defining a load type ‘Connector Force’ in the third local direction (\( Z \)).

- The final state of this first Abaqus/Standard analysis becomes the initial state of a dynamic explicit simulation accomplished using Abaqus/Explicit using the *Import functionality. The vessel is submitted to the blast load following the same methodology as described in the paragraph 3.1, the bolts pre-tension load being taken into account. This pre-tension load state is maintained locking all the degrees of freedom of the connectors. Finally, we access the axial force history in the connectors which is equivalent to the axial force seen by the bolts.

**4.2 Analysis model validation**

Uniaxial gauges are attached to two M36 bolts and to two M24 bolts. In both cases, one of these bolts is located on the inner circle of bolts and the other on the outer circle of bolts. These gauges allow us to know the initial force level obtained as a result of the tightening torque and to follow the time-force evolution during the experiment.

The measures accuracy is +/-13%. The finite elements model is evaluated on its capacity to estimate the greatest force passing through the bolts. Figure 11 shows the experimental and numerical maximum levels of force for a given experiment in:

- The M36 bolts of the top cover located in the inner circle (\( TC \) INT),
- The M36 bolts of the top cover located in the outer circle (\( TC \) EXT),
- The M24 bolts of the side cover located in the inner circle (\( SC \) INT),
- The M24 bolts of the side cover located in the outer circle (\( SC \) EXT).

Very good agreement is found. This notification is valid for others experiments.
5. SUMMARY

This paper describes the tools of a numerical chaining used to design a confinement vessel submitted to blast load. This chaining links first CATIA V5 to ABAQUS/CAE and secondly the CEA hydrodynamic code to ABAQUS/Explicit in a weak coupling. The ABAQUS/Explicit model dealing with the dynamic response of the vessel is presented and a comparison between experimental and numerical results is shown. The paper describes also an ABAQUS/Standard-ABAQUS/Explicit model the aim of which is to be able to estimate the higher axis force existing in the bolts of the covers when the vessel is solicited by a blast load. Comparisons with experimental results validate the two proposed models.

6. REFERENCES

1. VDI 2230 “Systematische Berechnung hochbeanspruchter Schraubenverbindungen Zylindrische Einschraubenverbindungen” Verein deutscher Ingenieure
Account of Modal Damping Instead Rayleigh One in Floor Response Spectra Analysis in Civil Structures of Nuclear Power Plants (NPP) Under Aircraft Crash

Ph.D. V. Korotkov, A. Ivanov, A. Naumkin (JSC Atomenergoproekt, Moscow)

Abstract. Presently the problem of safety providing of NPP under aircraft crash (AC) has special importance, because rather often floor response spectra (FRS) obtained in the AC analysis are dominant with respect to spectra, obtained in other special dynamic loadings, such as seismic load and explosive wave. It leads to necessity of obtaining more realistic solution of the problem under consideration with taking into account the following factors. Firstly it is necessary to consider inelastic behavior of concrete in the area of aircraft crash using model of concrete from Abaqus software. Secondly it is also necessary to use more accurate damping in materials. The case is that in direct integration method Rayleigh formula is used to describe material damping, which gives too conservative results. In this paper we suggest approach, which gives an opportunity to change global Rayleigh damping matrix on global modal damping matrix, that previously was determined by special way. Also in this paper we represent results of the analysis of test model and of the real example, where authors represent significant diminishing of spectral accelerations in case of changing Rayleigh global damping matrix on modal global damping matrix.

Keywords: civil structures of NPP, Floor response spectra, Direct integration method, modal method, Rayleigh damping, modal damping.

1. INTRODUCTION

To obtain realistic solution of the problem of obtaining floor response spectra of NPP structures under aircraft crash we have to take into account the following factors: contact interaction of striker and barrier, inelastic behavior of concrete in impact area, influence of inelastic soil. Consideration of these factors is possible only in the direct integration method in Abaqus. However with an advantage (possibility of solving nonlinear problems, including contact interaction) this method has a significant disadvantage – accounting material damping in soil-structure system. The case is that Rayleigh damping, used in direct integration method, gives too conservative results, comparing to normative values of material damping, used in modal methods of analysis.

Rayleigh formula to obtain the global damping matrix is:

\[ \tilde{N}_i = \alpha M + \beta K \]  

Matrices \( M \) and \( K \) are global matrices of mass and stiffness respectively, parameters \( \alpha \) and \( \beta \) obtained as follows:

\[ \alpha = 2\xi\omega_1\omega_2/(\omega_1 + \omega_2) \]  
\[ \beta = 2\xi(\omega_1 + \omega_2) \]

In formulas 2 \( \omega_1 \) and \( \omega_2 \) are values of natural frequencies, where material damping is equal to \( \xi \) – normative value. On other frequencies damping values can be obtained as follows:

\[ \xi = 0.5(\alpha / \omega + \beta \omega) \]

From equation 3 it can be seen, that for loads, exciting oscillations on frequencies between \( \omega_1 \) and \( \omega_2 \), we will get significantly higher values of floor response spectra (FRS), because corresponding calculated values of damping are lower than their normative values. On the contrary for loads, exciting oscillations higher than \( \omega_1 \) or lower than \( \omega_2 \), spectra will be nonconservatively lower, because damping values are rather higher than normative.
Here is the demonstration of the technology of application the Rayleigh damping in case of aircraft type Phantom RF-4E crash with load amplitude, represented in figure 1, on concrete box-type building, with material damping 7% and frequency of the first mode 3.46Hz. Notice, that detailed description of the building is represented further in section 3.

To obtain boundary frequency $\omega_2 = 2\pi f_2$, we recalculate load from time domain to frequency domain by means of floor response spectra method. Resultant load in frequency domain is represented in figure 2.
From figure 2 it can be seen, that on frequency about 85 Hz spectrum becomes asymptotically equal to the maximum value of load amplitude (110MN), and higher frequencies don’t represent any value in obtaining Rayleigh damping. Considering foresaid, we will get $\alpha=2.547$ and $\beta=0.000253$. Plot I - damping, relative to frequency (equation 3) is represented in figure 3. Modal damping coefficients (III) are also represented on this plot.

As can be seen from figure 3, damping on operating frequencies decreases to 2.7%, which is significantly lower, than corresponding modal damping coefficients. Therefore, to decrease conservatism, an attempt to replace Rayleigh damping with modal damping is made in this paper.

Considering Rayleigh damping, we should notice, that if we take, for example, the upper frequency limit 33Hz (boundary frequency for seismic analysis), instead of 85Hz, we can get some decrease of conservatism (plot II on figure 3). But this solution isn’t acceptable for aircraft crash analysis, because there is a significant high-frequency loading, that has influence on the whole system response in frequency range higher than 33Hz. And in case of using damping from plot II this influence is nonconservatively lowered because of too high damping on this frequency range.

2. THEORETICAL OVERVIEW

To solve the problem of replacing Rayleigh damping matrix on modal damping matrix in direct integration method let us firstly consider some aspects of modal method. This method is based on orthogonal property of global matrices in system of equations of motion. It means that projecting these matrices on eigenmodes basis make them diagonal. Considering matrices of mass and stiffness, there is strong mathematic proof of orthogonal property, but considering damping matrix – it is just a hypothesis, which can be wrong, for example, considering “soil” dashpot, which takes into account radiation damping. But, when considering only material damping, this hypothesis is acceptable. Orthogonal property for material damping matrix $C_1$ looks as follows:

$$\Phi^T C_1 \Phi = 2\Omega Z$$, where

$$\Phi (N \times N)$$ – fundamental matrix, composed of eigenmodes,

$$\Omega=\text{diag}[\omega]$$ – diagonal matrix, elements are eigenfrequencies,

$$Z=\text{diag}[\xi]$$ – diagonal matrix, elements are modal damping values.

Upper script $T$ means that matrix is transposed.

Source: SIMULIA Community Conference, 2014
To obtain detailed matrix $C_1$ from equation 4 we can use following normalization condition of eigenmodes:

$$\Phi^T M \Phi = E$$

(5)

Multiplying both parts of this equation on $(\Phi^T)^{-1}$, we obtain:

$$(\Phi^T)^{-1} \Phi^T M \Phi = (\Phi^T)^{-1} \Phi$$

(6)

Transposing matrix 6, we obtain:

$$\Phi^T = (M \Phi)^T$$

(7)

Then multiplying both parts of equation 4 on $(B^T)^{-1}$ on the left and on $B^{-1}$ - on the right and considering matrices 6 and 7 we obtain equation for global modal damping matrix:

$$C_1 = 2M \Phi \Omega Z (M \Phi)^T$$

(8)

Notice, that equation 8 was mentioned earlier, for example in [1, page 112]. But authors demonstrated independent way of obtaining it to show that there is a strong theoretical justification and no restrictions in further usage.

Using the following matrix property:

$$(AB)^T = B^T A^T$$

we can get:

$$C_1^T = 2(M \Phi \Omega Z (M \Phi)^T)^T = 2M \Phi (M \Phi \Omega Z)^T (M \Phi)^T = 2M \Phi \Omega Z (M \Phi) = C_1$$

So matrix $C_1$, defined by equation 8 is symmetric. It is important in further calculations.

To get matrix $C_1$, all components of the right part of equation 8 were obtained in Abaqus. Then using the additional software, wish damping matrix $C_1$ was generated. *Matrix input option is not supported in Abaqus with *Dynamic step, but there is a possibility to generate a substructure and to embed it in the model as superelement, suchwise we can get elements with recalculated damping matrix $C_1$, adopted to Abaqus software.

As noticed above, replacing matrix 1 on matrix 8 in direct integration method leads to significant decrease of conservatism, and this is an obvious advantage. But matrix 8 is fully populated so more computational recourses are needed to realize this method.

### 3. FE MODEL DESCRIPTIONS AND RESULTS.

Two following tests were made to verify the method and to analyse advantages of the development.

**Test 1.**

Forced oscillations of 3 masses on 3 springs, connected consequentially are considered. Loading is a harmonic function $A = A_0 \sin 2\pi f(t)$, where $A_0 = 1$.

Test model is represented in figure 4.

![Test 1 model](https://source.com/simulia/community_conference_2014)

**Source:** SIMULIA Community Conference, 2014
Masses are the same and equal to 1kg.

Spring stiffnesses:

- $K_1 = 1300 \text{ N/m}$
- $K_2 = 3250 \text{ N/m}$
- $K_3 = 6750 \text{ N/m}$

Damping in system is considered 5% and accounted as Rayleigh damping in masses, where $\alpha = 1.641$, $\beta = 0$, and $\xi = 0.5\alpha/2\pi f_i = 1.2\%$.

Eigenfrequencies are:

- $f_1 = 3 \text{ Hz (18.856 rad/s)}$
- $f_2 = 11.3 \text{ Hz (70.978 rad/s)}$
- $f_3 = 20 \text{ Hz (126.12 rad/s)}$

As a result of different methods (direct integration method with Rayleigh damping matrix and direct integration method with recalculated damping matrix) of application of harmonic load $A = A_0 \sin 2\pi f_2(t)$ to the base of the model, response spectra were obtained. They are represented in figure 5.

![Comparison of Response Spectra](image)

**Figure 5.** Response spectra in test 1.

From figure 5 a significant decrease of spectral acceleration with replaced damping matrix can be noticed on resonance frequency 11.3Hz. This is because of the modal damping value, which is frequency independent and is equal to 5% on all frequency range, and Rayleigh damping on 11.3Hz is about 1.2%. Besides, response spectrum, obtained by modal method, when damping on all 3 modes was 5%, was almost identical to spectrum, obtained by direct integration method with replaced damping matrix. Time increment was constant 0.003s.

Source: SIMULIA Community Conference, 2014
Test 2. In this test considered a crash of aircraft type Phantom RF-4E on industrial box-type building with two bridge cranes. Model of the building and its section are represented in figure 6.

Building is modelled with shell and beam elements, dimensions of the foundation are 48.3m x 54.9m.

Material used: concrete
- Young’s modulus \(3.31 \times 10^{10}\) N/m\(^2\)
- Poisson’s ratio 0.2
- Damping ratio 7%

Equivalent stiffness and damping of the soil were calculated, considering ASCE 4-98 [3, sect 3.3.4.2.2]. Homogeneous elastic half-space soil was taken with the following properties: Velocity of shear wave \(V_s = 600\) m/s, density of the soil \(\rho = 2.2\) t/m\(^3\), Poisson’s ratio \(\nu = 0.4\). Distribution of equivalent stiffness on foundation slab was made with a “saddle” form, equivalent damping was distributed evenly. The solution was made with variable time increment, but not more than 0.0002s. Loading of aircraft crash, represented in figure 1, was applied in point A, and comparative analysis of obtained response spectra (figure 7) in point B (figure 6). From figure 7 a significant difference between spectra can be seen, especially in horizontal directions. Maximal difference is about 30%. But it should be noticed, that on high frequencies (more than 35Hz) sometimes replacing damping matrix leads to enlargement of spectral accelerations, but these frequencies are not interesting in facilities analyses.

Source: SIMULIA Community Conference, 2014
Figure 7. Comparison of response spectra in point B of box-type building.

Source: SIMULIA Community Conference, 2014
4. CONCLUSION
1. Developed a new method, allowing to use global modal damping matrix instead of Rayleigh damping matrix in direct integration method, while solving dynamic problems.
2. Method realisation showed significant decrease of spectral accelerations (up to 30%), which gives positive effect.
3. Method can be used not only for aircraft crash analysis, but also in explosive wave and seismic analyses.
4. Considering high positive effect, authors suggest to include this method in Abaqus software with some improvements.

5. REFERENCES
3. ASCE 4-98. ASCE STANDARD. Seismic ANALYSIS of safety Related Nuclear Structures and Commentary. Approval 1992

6. ACKNOWLEDGEMENT
The Authors would like to thank Ph.D Bouzinov Pavel, Dassault Systèmes Simulia Corp. (Providence) for the help with adopting global damping matrix in Abaqus environment.
Identification of Ductile Damage Parameters

Jan Ruzicka, Miroslav Spaniel, Milos Moravec (Czech Technical University in Prague, Faculty of Mechanical Engineering)
Antonin Prantl (SKODA - JS a.s.)
Jan Dzugan, Pavel Konopík (Comtes FHT a.s.)

Abstract: This paper describes the calibration process for uncoupled material model of ductile damage by Johnson-Cook and Rice-Tracey that is implemented in FE package Abaqus. As the work was supported by the grant “Identification of ductile damage parameters for nuclear facilities”, calibration was done for typical steel used in nuclear power plant industry. The project includes both design and realization of experiments, and application of experimental outputs in calibration process of material constants of mentioned ductile damage models. Calibration process of material model uses fifteen types of experimental specimens corresponding with literature. The result of the calibration process was verified through the comparison of FE simulation of each specimen with experimental response. As the material model describing ductile damage within the wide range of stress states was searched in this project, the model was tested on a several another specimens which exhibit higher stress concentration.

Keywords: Phenomenological material modeling, Ductile damage, Fracture Locus

1. INTRODUCTION

Ductile damage is the process of metallic material damage in conditions of monotonic loading. Evolution of the damage follows plastic straining and ends by fracture of component. Problems of ductile damage play significant role in industry, for example in optimization of technological processes, evaluation of safety in automotive and aeronautic industry, analysis of steel civil structures etc. Complex material models of ductile fracture require calibration based on extensive experimental tests and their computational is costly as well. Thanks to progress of computational methods, namely FEA, such models can be nowadays used in engineering.

Phenomenological material models describing ductile damage in continuum mechanics mostly introduce extension of plasticity models. Two types of material models can be distinguished. Uncoupled models are separate plastic response and ductile damage and failure. Coupled models modify plastic response in dependence on damage evolution. Even though coupled models have huge potential, their complexity and calibration costs results into small extension in practice. Easier calibration process is an essential advantage of uncoupled material models, for which the calibration of plastic response and calibration of ductile damage can be separated. The calibration is distinctly easier if the uncoupled material model is used.

From the point of view of interpretation of real process inside material the ductile damage material models can be classified as either phenomenological models either micro-structural models. Micro-structural models attempts to describe damage by natural way. Damage occurs on the basis of two different mechanisms. Initiation, growing and connecting of micro-cavities dominates in domains with tri-axial stress. Load carrying cross section is reduced during damage process and finally leads to failure. Based on representative volume with cavity some micromechanical continuum models were derived (Rice and Tracey, 1969, etc). However models based on this damage concept exhibit unrealistic response in domains at that pressure and shear stress are dominant. In this case material is damaged by localized shear strain in plane of maximal shear stress that holds an angle 45° with first principal plane. The phenomenological models do not have physical meaning and try best capture the behaving of real material on base of empirically designed relations. They are used for description of both of the two previously mentioned mechanisms. These models should be based on parameters having significant influence on ductile damage. The parameters are usually designed based on experience, intuition or are deduced from micromechanical continuum models. By the latter way the substantial influence of triaxiality stress was demonstrated. Formulation of failure criterion describing efficiently the process of damage is another crucial point of phenomenological models. Phenomenological models provide us with acceptable description of response in case of materials they was designed and calibrated for. On the other hand the experimental tests are costly and their portability to other materials is limited.

2. DUCTILE DAMAGE MODEL

Material model discussed in this paper is based on both classical incremental model of plastic response with isotropic hardening and phenomenological concept of damage in continuum mechanic. This model supposes isotropy, and for description of stress state uses Von Mises stress $q$, stress triaxiality $\eta$, and Lode parameter $\xi$. These quantities are defined using second and third invariant of deviatoric stress

$$J_2 = \frac{1}{2} \left( S_1^2 + S_2^2 + S_3^2 \right), \quad J_3 = S_1 S_2 S_3.$$

Source: SIMULIA Community Conference, 2013
Principal deviatoric stresses \( S_1, S_2 \) and \( S_3 \) are principal values of stress deviator

\[
S = \dot{\sigma} + pI, \quad p = -\frac{1}{3} \text{tr}(\dot{\sigma}),
\]

where \( p \) is hydrostatic stress. Von Mises stress is defined as

\[
q = \sqrt{\frac{3}{2} J_2},
\]

Stress triaxiality \( \eta \) and Lode parameter \( \xi \) can be expressed as

\[
\eta = -\frac{p}{q}, \quad \xi = \frac{27 J_1}{2 q},
\]

The model of plastic response works with simple surface of plasticity that is based on Von Mises stress

\[
q = \sigma_\gamma(\bar{\varepsilon}_{pl}),
\]

associated flow rule and has the only one history dependent state parameter – accumulated intensity of plastic strain, resp. accumulated plastic strain

\[
\bar{\varepsilon}_{pl} = \int_0^t \dot{\bar{\varepsilon}}_{pl} dt, \quad \bar{\varepsilon}_{pl} = \sqrt{\frac{2}{3} \bar{a}_{pl} : \bar{a}_{pl}}.
\]

Dependence of \( \sigma_\gamma(\bar{\varepsilon}_{pl}) \) is calibrated experimentally.

Failure criterion is based on phenomenological quantity damage \( \omega \), that is defined as non-decreasing scalar parameter

\[
\omega = \int_0^t \frac{\dot{\bar{\varepsilon}}_{pl}}{\bar{\varepsilon}_{\vartheta}(\eta, \xi)} dt,
\]

that depends on loading history and can be understood as linear accumulation of incremental damage in process of monotonic loading. Fracture locus \( \bar{\varepsilon}_{\vartheta} \) is function of stress triaxiality and Lode parameter and it has to be calibrated experimentally. Ductile fracture of material occurs as soon as critical damage value \( \omega_{\text{crit}} \) is reached. The fracture locus has physical meaning of accumulated plastic strain at the instant of ductile damage initiation at the end of hypothetic monotonic loading with both triaxiality and Lode parameter constant. In such loading process the damage at failure reaches value \( \omega_{\text{crit}} \), so damage defined in (7) can be said to be normalized.

In this paper Johnson-Cook material model (Abaqus 6.12, 2012) was employed for description of ductile damage

\[
\bar{\varepsilon}_{\vartheta}(\eta, \dot{\bar{\varepsilon}}_{pl}, \bar{T}) = \left[ d_1 + d_2 e^{-d_0} \right] \left[ 1 + d_4 \ln \left( \frac{\dot{\bar{\varepsilon}}_{pl}}{\dot{\varepsilon}_0} \right) \right] \left( 1 + d_3 \bar{T} \right),
\]

where \( d_1, d_2, d_3, d_4 \) are failure parameters, \( \dot{\varepsilon}_0 \) is the reference strain rate, and \( \bar{T} \) is the dimensionless temperature. In this paper quasi-static loading at room temperature is supposed. Therefore only the first term (parameters \( d_1, d_2, d_3 \)) of Johnson-Cook model is calibrated.

Exponential dependence of fracture locus on stress triaxiality also appears in material model Rice-Tracey (Rice, 1969) that is based on description of growing micro-cavities in basic material matrix. The Rice-Tracey model is defined

\[
\bar{\varepsilon}_{\vartheta}(\eta) = C_{RT} e^{-\frac{3}{2} \eta},
\]

where \( C_{RT} \) is failure parameter that has to be calibrated.

Bao and Wierzbicki (Bao, 2005) determined experimentally fracture locus in wide range of stress triaxiality and showed that the dependence of fracture locus on stress triaxiality need not be monotonic decreasing function. Xue and Wierzbicki (Wierzbicki, 2005) expanded the dependence of fracture strain on triaxiality by third invariant of stress tensor. This invariant was included in form of Lode parameter. Xue and Wierzbicki used elliptical function for description of dependence of fracture locus on Lode parameter. They defined symmetric function (fracture locus) that shows the same dependence of axisymmetric tension \( (\xi = 1) \) and axisymmetric

Source: SIMULIA Community Conference, 2013
pressure ($\xi = -1$). Further generalization is introduced in Bai-Wierzbicki model (Bai, 2007) that expects fracture locus generally asymmetric (Figure 1).

The artificial degradation function described by parameter of degradation is implemented in Abaqus software and is created due to prevent stepped loss of stiffness in the whole element at the instant of failure criterion is reached. Difference between damage and degradation process is in dependence on fracture strain. The degradation is not included as material parameter and therefore it is not included in calibration process. Nevertheless the results of numerical simulation can be affected by choice of degradation. Since failure is indicated in the element of FE mesh the degradation manifesting itself as decrease of elastic modulus is started

$$E' = (1 - D)E$$

(10)

After the critical value $D = 1$ is reached in element, it is removed form FE mesh. The damage process could be controlled by more ways in Abaqus. In this paper the description based on Hillerborg's fracture energy was used.

$$G_f = \int q \, du_{pl} = \int Lq \, d\varepsilon_{pl}$$

(11)

Degradation process is defined as

$$D = \int \frac{Lq}{G_f} d\varepsilon_{pl},$$

(12)

where $L$ is characteristic size of element. As the Equation 12 shows, dependence of damage course on characteristic size of element $L$ is disadvantage of this approach. In area of expected damage the mesh with the same element size should be used. Material parameter $G_f$ must be recalculated for different mesh density. From Equation 11 the fracture energy $G_f$ can be expressed for any characteristic element length $L$ differing from $L_0$ using values of $G_{f0}$

$$\frac{G_f}{L} = \frac{G_{f0}}{L_0} = \int q \, d\varepsilon_{pl}, \quad G_f = \frac{G_{f0}}{L_0} L$$

(13)

Source: SIMULIA Community Conference, 2013
Dependency of degradation development on mesh density can be removed by setting $G_f \rightarrow 0$. The phase of degradation is minimized and the full degradation of element occurs immediately after the critical damage value $\omega_{\text{crit}}$ is reached. All samples simulated in this paper have the same size of element edge (0.2 mm) in expected damage initiation. Small punch is exception. The element size of this type of specimen is 0.05 mm.

3. THE METHODS OF CALIBRATION

Several specimen types with different values of both stress triaxiality and Lode parameter at expected locations of ductile fracture should be used for successful calibration of material model. Because in most cases the course of quantities $(\eta, \xi, \bar{E}_{pl})$ during loading process can not be described using analytic formulas, the specimens have to be analyzed via FE. The calculated course of these quantities serves as input into calibration process. Experimental determination of fracture strain $\bar{E}_f$ is essential. Mostly the critical extension $\Delta L_f$ at the instant of failure is determined from experimental data. Fracture strain $\bar{E}_f$ in expected location of failure is then calculated using FE simulation. The critical extension could be determined for example via direct surface observation and first individual cracks detection. This approach is limited onto specimens at which the fracture starts form surface. Usually, critical extension is identified on base of sudden decrease of loading force in force-displacement record. It is possible to use the method of digital image correlation for direct evaluation of fracture strain in case of failure on the surface.

Two approaches are commonly cited to be used in calibration process of uncoupled ductile damage model. The first one is based on averaged values of stress triaxiality and Lode parameter (Bai, 2007) etc. Averaged values of stress triaxiality $\eta_{av}$, resp. Lode parameter $\xi_{av}$ are based on plastic strain weighted average

$$
\eta_{av} = \frac{1}{\bar{E}_f} \int_0^{\bar{E}_f} \eta(\bar{E}_{pl}) \, d\bar{E}_{pl}, \quad \text{resp.} \quad \xi_{av} = \frac{1}{\bar{E}_f} \int_0^{\bar{E}_f} \xi(\bar{E}_{pl}) \, d\bar{E}_{pl}.
$$

The point $[\eta_{av}, \xi_{av}, \bar{E}_f]$ each individual sample can be determined by this approach. Fracture locus $\bar{E}_f(\eta, \xi)$ passes through this point. The main disadvantage of this approach is wide range of stress triaxiality and Lode parameter $\xi$ for some specimen types resulting into non-negligible error caused by averaging of these quantities. The point of minimum of target functional $F_{av}$

$$
F_{av} = \sqrt{\frac{1}{N} \sum_{i=1}^{N} \left| \bar{E}_f - \bar{E}_f \left( \eta_{av}, \xi_{av} \right) \right|^m}
$$

is searched employing suitable optimization tools. This functional, at which $N$ means total number of calibrated specimens, $m$ expresses the rate of weighting of individual deviations, expresses total error of $\bar{E}_f(\eta, \xi)$ in points $[\eta_{av}, \xi_{av}, \bar{E}_f]$ corresponding with i-th specimen. More balanced deviation can be expected with growing value of $m$ ($m = 2$ corresponds with least squares method). Bai and Wierzbicki use modified target function $F_{av}^*$ in their work (Bai, 2007). Individual deviations are weighted by fracture strain.

$$
F_{av}^* = \frac{1}{N} \sum_{i=1}^{N} \left| \bar{E}_f - \bar{E}_f \left( \eta_{av}, \xi_{av} \right) \right|
$$

Optional application of least squares enabling employment of linear regression is quality of this approach. Some material models can be modified so that the linear regression can be used partially.

The second approach defines target as deviation of damage $\omega_i$ integrated up to fracture strain $\omega_{\text{crit}} = 1$ for i-th specimen averaged over all specimens.

$$
F_{\omega} = \sqrt{\frac{1}{N} \sum_{i=1}^{N} \left| 1 - \omega_i \right|^m}, \quad \omega_i = \frac{\int_0^{\bar{E}_f} \frac{d\bar{E}_{pl}}{\bar{E}_f(\eta, \xi)}}{ar{E}_f}
$$

This approach eliminates averaging in Equation 13 however calibration costs are higher in comparison with the first approach and

Source: SIMULIA Community Conference, 2013
moreover existence of global minimum uncertainty is higher in this case. For this reasons fine tuning of parameters that was found using averaging quantities is preferred application. This approach was used for example in (Vaziri, 2010).

4. EXPERIMENTS

Material calibration of ductile damage material model was carried out for steel typically used in nuclear industry. The portfolio of fifteen calibration specimen types for quasi-static loading was designed to calibrate fracture locus. Single specimens in space of stress triaxiality and Lode parameter are shown in Figure 2. It is based on averaged values of \( \eta_{av} \) and \( \varepsilon_{av} \) at instant of expected fracture.

The substantial subset of calibration portfolio is based on special specimen with double curvature (butterfly) that was described in (Mohr, 2007). Essential advantage is possibility to cover wide range of stress triaxiality and normalized Lode angle with only one specimen geometry. Different stresses are caused by loading in different directions (tension: 0°, tension and share: 30° - 90°, compression and share: 100° - 130°). Another advantage is that damage initiation is always in centre of the specimen. The special testing device was developed to perform calibration tests using uniaxial loading machine. Figure 3 on the left shows example of device configuration. It was shown during the testing verification phase that the experimental data was distorted for the reason of friction in the guiding rods of the device. Therefore, simplified FE model of whole device (Figure 3 in the middle) is commonly used in numerical simulations of butterfly specimen. The friction coefficient was determined using try and error method to get correspondence with the experimental data. The disadvantage of this specimen type is shape complexity. The geometric inaccuracy may cause scattering between experimental data and FE simulations.

The basic configuration of the butterfly for angle position 0° was thanks to symmetry measured without device and undesirable friction effect was removed. FE models of specimens (Figure 3 on the right) use linear volume elements C3D8 with uniform edge size 0.2 mm in damaged area. In basic position 0° two modifications of the butterfly with double thickness (T2) and triple thickness (T3) were measured without device.

Figure 2. The portfolio of calibration specimens.
The second subset of calibration specimens consists of notched round bars with different notches size (R∞, R15, R7 and R4). The specimen damage develops in conditions of constant value of Lode parameter $\xi = 1$. Each of these specimens was designed so that the fracture occurs first in the centre of the specimen. The bilinear axisymmetric elements CAX4 with uniform size 0.2 mm in damage area was used (figure 4).

The last calibration experiment is so called small punch test using miniature flat circular specimen through that the small steel ball is pushed. This experiment is used for verification of material properties of structures in operation from that small piece of material can be cut off without functionality loss. The specimen loading corresponds with biaxial tension. Disadvantage of this test is dependence on friction between ball and specimen. Friction coefficient that affects the place of damage initiation is uncertain, so we have chosen the best value within appropriate range. The bilinear axisymmetric elements CAX4 with global size 0.05 mm in damage area was used for FE simulation of the specimen (figure 5).
5. RESULTS OF CALIBRATION PROCESS OF DAMAGE MATERIAL MODEL

The calibration process was based on portfolio of fifteen different specimen types (Figure 2). The resultant fracture locus is shown in Figure 6. The figure introduces fracture strains of single specimens. Symbol associated with \( i \) specimen is plotted at position \( \eta_{av}, \xi_{av}, \varepsilon_f \).

![Fracture locus](image)

Figure 6. Resultant Johnson-Cook and Rice-Tracey fracture locus.

Fracture strain \( \varepsilon_f \) of this material showed to be independent on Lode parameter \( \xi \). Therefore Johnson-Cook (the first term of Equation 8) and Rice-Tracey (Equation 9) material models are acceptable approximations for this kind of material. Figure 6 shows that Johnson-Cook fracture locus is non-positive for higher stress triaxiality by reason of negative parameter \( d_1 \). Therefore the fracture strain is supposed to be zero for \( \eta > 1.5 \).

The results of FE simulations for both used fracture locuses (Johnson-Cook and Rice-Tracey) are shown in subsequent figures. Response of Johnson-Cook material model is more close to experiments for most of specimens. It is possible to say that both models reach acceptable agreement with experimental data. Configuration with butterfly 120° was tested as well, but this specimen did not exhibit fracture until achievement of device limits which manifests itself as step in experimental loading curve in Figure 9.

Source: SIMULIA Community Conference, 2013
Figure 7. The result of simulations with Johnson Cook and Rice Tracey model.

Figure 8. The result of simulations with Johnson Cook and Rice Tracey model.

Source: SIMULIA Community Conference, 2013
6. THE SPECIMENS WITH HIGH LOCAL STRAIN CONCENTRATION

Calibration of material model that was performed using fifteen different specimens has proved ductile fracture of tested material dependency on stress triaxiality. Calibrated material model was tested on a few another specimen types with higher stress concentration. The series of notched round bars was expanded by new notched round bars R1 and R2 (Figure 4). Further specimen typically used in fracture mechanics – CT - without pre-crack was employed.

The comparison of experimental response of the specimens with FE simulations using Johnson-Cook and Rice-Tracey material model is presented in the Figure 11. The notched round bars with sharp notch showed acceptable correspondence between FE simulation and experimental data. Ductile fracture model exhibits poor results for CT when using Johnson-Cook ductile fracture model, while the result of Rice-Tracey model is acceptable (Figure 12). This is not in contradiction, because Rice-Tracey model was outlined on theoretical basis in the same form as Johnson Cook one with $d_1 = 1.5$. As the $d_1$ is calibrated parameter, Johnson-Cook model is in better correspondence in range of calibration tests, but it can be much worse outside it. The Figure also compares the final CT crack shape between experiment and FE (Rice-Tracey) simulation.
Figure 11. The result of FE simulations of notched round bar R1 and R2 for Johnson Cook and Rice Tracey model.

Figure 12. The result of FE simulations of CT. Johnson Cook and Rice Tracey model. Final CT crack shape.

7. CONCLUSIONS

Calibration of uncoupled phenomenological ductile fracture models (Johnson-Cook and Rice-Tracey) in FE software Abaqus was discussed. Fracture locus is expressed as function of stress triaxiality. Fracture strain was calculated on the base of the specimen extension at material failure. Portfolio of quasi-static calibrating tests was designed. Experiments on special specimen with double curvature (butterfly) which is loaded in different directions provided us with wide range of stress triaxiality and Lode parameter. For this type of the specimen special device allowing different loading angles had to be manufactured. In basic position 0° two modifications of butterfly with double and triple thickness were measured as well. Further tensile tests of notched round bars and small punch test were performed. Calibration of both ductile fracture models was carried out on steel typically used in nuclear industry. More accurate ductile fracture description in range of calibration experiments was reached using Johnson-Cook model. Both material models successfully describe ductile damage of calibration specimens that are commonly presented in the literature. In this paper the material models tests on specimens that showed local strain concentration was introduced. The notched round bars with sharp notch showed acceptable correspondence between FE simulation and experimental data. Ductile fracture of CT is poorly described when using Johnson-Cook model, but the result of Rice-Tracey model is well acceptable. We will focus on strain concentration in our further work.
8. REFERENCES


9. ACKNOWLEDGMENT

This work was done within the work on the project “Ductile damage parameters identification for nuclear power plants - FR-TI2/279” sponsored by Ministry of Industry and Trade of The Czech Republic.

Source: SIMULIA Community Conference, 2013
Abaqus/Standard Simulation of Coupled Multi-physics Processes at an Underground Nuclear Waste Disposal Site

SUMMARY
The integrity of a nuclear waste burial site must be maintained for several millennia. The physical processes taking place at an underground disposal facility include heat generation from residual radioactivity, groundwater seepage, heat transfer, and material swelling. These processes induce stress in the waste containment structure and must be understood in order to develop safe long-term disposal schemes. In this Technology Brief we show how the coupled multi-physics simulation capabilities of Abaqus/Standard can be used to analyze design options for nuclear waste burial.

BACKGROUND
Spent nuclear fuel remains radioactive for centuries and underground burial is one of the most promising disposal methods. Different approaches for waste burial are being researched [1] and one of the main goals of any design is the prevention of leakage. The storage site location thus needs to be geologically stable and the waste needs to be encased in a material that can minimize any possible ground water contamination. Bentonite and similar clays are useful in this capacity, as they have low permeability and expand when their water content increases from an unsaturated state. This property helps maintain a good seal even when these materials become exposed to ground water.

A schematic diagram of a representative design for waste burial is shown in Figure 1. The spent nuclear fuel is placed in a canister, which is then encased in clay. This assembly is placed in an excavation in rock and is then sealed by a plug. A suitable depth in the surrounding rock is chosen based on local geological conditions.

Burial designs can be comparatively assessed by analyzing the likely outcomes of potential failure modes. The fully coupled temperature-displacement-pore fluid flow analysis capability in Abaqus/Standard is used for the analyses. The model is restrained from movement normal to the outer hole represents the access tunnel. The finite element mesh is shown in Figure 2. An axisymmetric model is used for the present analyses, but a full three dimensional model can be used when more detailed analyses are required.

The burial hole diameter is 1.4 m with a depth of 7 m from the base of the tunnel. The canister is cylindrical with a radius of 0.35 m and a length of 3 m. The plug is 1 m in height and the remaining region of the hole is occupied by clay.

The canister, rock, and the plug are modeled as linear elastic. The clay is assumed inelastic with a negligible friction angle and is therefore modeled using Mises plasticity. All four components are thermally conductive. Additionally, the clay and the plug permit pore-fluid flow. A sorption relationship that defines saturation vs. capillary pressure is specified for the clay and the plug. The expansion in the clay that results from water ingress is modeled using the moisture swelling capability. Frictional contact interactions are defined between all components and a tie constraint is used between the clay and the plug.

ANALYSIS PROCEDURE
The fully coupled temperature-displacement-pore fluid flow capability in Abaqus/Standard is used for the analyses. The model is restrained from movement normal to the outer

Key Abaqus Features and Benefits
- Fully coupled temperature, displacement, and pore fluid flow analysis method for modeling the coupled interaction between heat transfer, pore fluid diffusion and stress
- Moisture swelling for modeling expansive materials
- Robust multi-physics contact interactions

Source: Abaqus Technology Brief
The clay and the plug are assumed to be partially saturated at the beginning of the analysis, with a saturation value of 0.3 at the top of the plug. The ground water level is assumed to be 3 m below the bottom of the hole in which the assembly is placed. The clay and the plug are considered to have a void ratio of 0.5 at the beginning of the analysis. The initial temperature is specified to be 30 degrees C in all regions of the model.

The models do not include the interior details of the canister. The increase in canister temperature that arises from radioactivity is specified by temperature boundary conditions; the time variation is shown in Figure 3. Note that the temperature decreases slowly over several millennia after a relatively quick rise to 95° C.

Mixed boundary conditions, wherein the heat flux is a function of temperature, are defined on the top surface of the plug and over all boundary surfaces of the rock. This allows for the generated heat to dissipate away from the burial site.

Two cases are analyzed. Case 1 is the response due to temperature variation with no water ingress; i.e., no change to the ground water level. Case 2 then includes the effects of water ingress at the clay-rock interface. Both analyses are run for a period of 10,000 years.

For Case 1 the pore pressure at the top of the plug is restrained to its initial value of -100,000 Pa for the full analysis time. For Case 2 the water ingress is assumed to start at year 100 and continue to year 1000. During this time the height of the ground water level rises by 10 m. It increases at a constant rate starting from 3 m below the bottom of the hole at 100 years, reaching the top surface of the plug at 1000 years and then remaining constant. For both cases the pore pressure boundary conditions are specified using the DISP user subroutine.

**RESULTS AND DISCUSSION**

The Case 1 temperature distribution 32.5 years after waste burial is shown in Figure 4. The temperature in the canister reaches about 92 degrees C at that time, and a significant rise in temperature is observed to occur in the surrounding clay and rock. The contact pressure on the rock is shown in Figure 5 at time 4 years 10 months.

In the Case 2 model, the clay expands as it absorbs water and the confinement of the surrounding rock can lead to high stresses. These stresses can compromise the integrity of the design and need to be accurately estimated. Figure 6 shows the pore pressure in the clay as water ingress takes place and Figure 7 shows the associated changes in saturation.

Figure 8 shows the Mises stress in the Case 2 model at an intermediate stage of water ingress. High values of Mises stress are seen to occur at the interface between the fully saturated
and partially saturated regions. The clay region that gets newly saturated expands due to moisture absorption and the region that has not yet reached a saturated state occupies a relatively lesser volume. This mismatch leads to the high Mises stresses in that region. The resulting stresses can then remain until the end of the analysis.

**CONCLUSION**

In this Technology Brief we have shown that the contact and fully coupled temperature-displacement-pore fluid flow analysis capability in Abaqus/Standard allows for the simulation of long-term behavior of nuclear waste repositories.

Only a single model configuration, with one-long term analysis case and one failure mode case, have been presented. Further detailed analyses could include complete three-dimensional geometry, plasticity models specific to the clay barrier, inclusion of canister details, fluid permeation in the rock, and creep behavior.

**REFERENCES**


**SIMULIA REFERENCES**

For additional information on the Abaqus capabilities referred to in this document please see the following Abaqus 6.13 Analysis User’s Guide references:

‘Coupled pore fluid diffusion and stress analysis,’ Section 6.8.1
Failure of a Prestressed Concrete Containment Vessel

SUMMARY
Finite element modeling of prestressed concrete containment vessels (PCCVs, Ref. 1) for nuclear power plants poses special challenges. PCCVs, which are heavily reinforced structures, are designed to deform beyond the cracking limits of the concrete. Abaqus has been used extensively for analyzing such structures in the nuclear utility industry (Ref. 2) and can be used to assess and improve the performance of these and other similar reinforced concrete structures.

BACKGROUND
Nuclear power plant containment structures are used for safeguarding against accidental release of radiation from nuclear reactors. These containment structures are composed of heavily reinforced and prestressed concrete. Accurate modeling of such structures is essential to ensure that they remain safe and reliable if an accident occurs. To validate the analytical models and finite element codes that are used to predict how much internal pressure is tolerated by such structures, a 1:4 scale model of an actual PCCV was constructed and loaded to failure at the U. S. Department of Energy's Sandia National Laboratories (Ref. 1).

Two tests were performed on the scale model: a Limit State Test (LST) and a Structural Failure Mode Test (SFMT). In both tests the structure was loaded beyond its design pressure. The LST was carried out to determine the internal pressure at which cracks begin to develop in the concrete leading to leakage of gases from inside the PCCV. The SFMT was carried out to determine the internal pressure at which complete structural collapse occurs. The structural response of the model was measured during both tests, and information on the failure mechanisms was collected. These data were used as the benchmark in a round robin test between design engineers for testing their material models and finite element analysis codes.

Dimensions of the PCCV scale model are shown in Figure 3. The structure is cylindrical, with a hemispherical dome. The overall height is 16.4 meters, and the radius of the cylinder is 5.3 meters. The cylinder’s wall thickness is 0.32 meters, reducing to 0.275 meters in the dome after a short transition region. The bottom of the cylinder is joined to a base that is 3.5 meters thick.

The concrete has passive reinforcement as well as prestressed tendons in the hoop and longitudinal (vertical in the cylinder, meridian in the dome) directions. The passive reinforcement is of mild steel, whereas the prestressed tendons use high...
Finite element modeling of a PCCV poses special analysis challenges. A PCCV is a heavily reinforced and prestressed concrete structure and is designed primarily for preventing any gas leakage from the inside of the PCCV to the outside environment. In some situations the PCCV also may be subjected to a much higher internal pressure leading to a full structural collapse wherein the concrete may deform well beyond its cracking limit. Abaqus has been used extensively for analyzing such structures in the nuclear utility industry; a representative example can be found in Ref. 2.

**FINITE ELEMENT ANALYSIS APPROACH**

As part of the pre- and post-test analyses of the containment vessel, a finite element model of the prototype PCCV was constructed by NNC Limited, UK (details of this work can be found in Ref. 3). In this technology brief the NNC model has been upgraded to take advantage of more recent Abaqus capabilities and now uses the concrete damaged plasticity model.
Abaqus provides two different constitutive models for the analysis of concrete. The smeared crack concrete model is capable of modeling compressive crushing as well as tensile cracking and postcracking behavior. The concrete damaged plasticity model provides for the degradation of the elastic stiffness induced by plastic straining, both in tension and compression.

The concrete material is modeled using first-order, reduced-integration, continuum brick elements (Figure 4). The prestressed tendons in the hoop direction are modeled with prestressed rebar. The vertical and meridional tendons are modeled with prestressed trusses.

The liner of the PCCV is thin; hence, it was modeled with membrane elements.

The PCCV model consists of approximately 150,000 elements and 0.5 million variables and is loaded by an internal pressure of 2.0 MPa.

RESULTS AND CONCLUSIONS

Abaqus analysis predicts the following:

Some of the rebar in the PCCV begin yielding at a load level of about 0.88 MPa. Some cracking in the concrete also takes place at this load. At a load level of about 0.997 MPa, rebar yielding and associated concrete cracking are extensive for the rebar in the hoop direction at the base of the PCCV buttresses at azimuths 270° and 90°. Azimuths and locations of the buttresses are shown in Figure 3. Results from the central cylinder of the model PCCV at a load level of 1.52 MPa are shown in Figure 5 and Figure 6. Figure 6 shows a plot of output variable RC YIELD, which is a scalar quantity denoting the onset of yielding. A value of 0 indicates that the rebar material has not yielded, and a value of 1 indicates that the rebar material has yielded. Figure 6 shows a contour plot of output variable DAMAGET, which is a scalar degradation measure used to express the reduced tensile elastic modulus of concrete after it has sustained cracking damage. The regions where the rebar has yielded, as shown in Figure 5, also exhibit associated concrete cracking. In the LST it was
observed experimentally that functional failure of the PCCV occurred due to liner tearing in a region containing repaired weld seams. This leakage associated with the liner tearing was first detected at a pressure of about 2.4 to 2.5 times the design pressure of 0.39 MPa; that is, at about 0.96 MPa. A subsequent increase in pressure during the LST resulted in further tearing at many strain concentration locations and further leakage (Ref. 1).

Structural collapse of the PCCV prototype occurs at an internal pressure of about 1.4 MPa, as inferred from the increase in the rate of change of displacements and the small time increments needed to run the analysis further beyond this load level. The experimentally measured internal pressure value for structural collapse is about 1.424 MPa (Ref. 1).

Contour plots of the displacement magnitude and the tensile damage near the complete collapse load of the PCCV are shown in Figure 7 and Figure 8, respectively. Figure 8 shows a contour of output variable DAMAGET. In Figure 9 and Figure 10 the load-displacement curves at two locations as predicted by ABAQUS are compared with the experimental results.

Figure 9: Values of radial displacement at Location 3 computed using ABAQUS and measured experimentally in the SFMT.

Figure 10: Values of radial displacement at Location 14 computed using ABAQUS and measured experimentally in the SFMT.

Source: ABAQUS Technology Brief
The Combined Method of Dynamic Analysis of the “Soil-Structure” Systems at Aircraft Crash onto Civil Structure of NPP

K. Ilin, V. Korotkov (JSC Atomenergoproekt, Moscow)

Purpose of this paper is to determine response spectra in locations of components in the reactor building of nuclear plant VVER-1000 in case of aircraft crash (AC). Crash of aircraft Phantom is considered here. However methodology taken in this paper can be extended to beyond-the-design basis accident that is crash of large commercial aircraft of Boeing-747 type. Code ABAQUS was applied for solution of the above-mentioned problem. Since aircraft crash causes nonlinear strain in shock area of concrete, we used the specially developed model of concrete in ABAQUS that is Brittle cracking model, which enabled us to consider cracking of concrete and plastic strain of rebars. Spatial model of the building was developed for the analysis; the model duplicates the complex inner structure of the building that enables simulation in true geometry. Soil was analysed on the basis of three-dimension elements of continuous medium that allows simulating soil lamination. Previously the analyses of AC response spectra considered homogeneous soil. It would be impossible to solve the problem with an ordinary personal computer because of demand on enormous main storage. Therefore efficient cluster of JSC Atomenergoproekt was utilized.

1. METHODOLOGY FOR CALCULATIONS OF SOIL-STRUCTURE

At present in analyses of soil-structure system at aircraft shock the soil basis is modelled as springs and dampers located under the building foundation. Characteristics of such spring-damper supports are determined for homogeneous elastic half-space by the formulas given in ASCE standards [1]. However is soil feature considerably heterogeneous nature, previously applied formulas from ASCE are unsuitable.

For the dynamic study of the reactor building with regard to complicated soil basis and similarly complicated its inner structure we could develop united finite-element model of the soil-structure system and solve problem in one step (implementation of direct solution). Spatial finite -element model of the system consisting of shell/plate elements for external structure and three-dimension elements of continuous medium for simulation of soil basis contains more than a million degrees of freedom and large width of band of stiffness matrix. Therefore solution of such problem with nonlinear behaviour of concrete will take long time.

That is why the idea is to solve the problem in two steps. At the first step we assume that the building is perfectly rigid and feature only inertia, and soil will be simulated by elements of continuous medium, which refer to the complicated structure of soil. At the second step we shall simulate the building on the basis of shell/plate elements, and we shall represent influence of soil as equivalent stiffness (6 springs) and equivalent damping (6 dampers), characteristics of which are determined in the first step.

In order to obtain equivalent linear stiffness as the initial factor, a unit deflection was applied, acting to stamp (see Figure 1), in geometric centre in the directions of linear degrees of freedom. Single turning angles were applied to obtain equivalent angular stiffness.

Figure 1. Stamp.

Source: SIMULIA Community Conference, 2014
In the analysis of equivalent damping in soil, the impulse impact from Phantom was applied to the stamp (see Figure 2) and after consideration of oscillating process the logarithmic decrement of damping was determined, which characterise radiation damping. Noteworthy, that just radiation damping will be determined in such study, since initial data for other types of damping were not used.

Sizes of stamp simulating the perfectly rigid foundation slab are as follows: $L = 76 \text{ m}$, $B = 54 \text{ m}$. Building mass is $m = 254432 \text{ t}$, and inertia moments around axes passing through geometric centre of the foundation slab are: $I_{xx} = 350217000 \text{ t}\times\text{m}^2$, $I_{yy} = 337464000 \text{ t}\times\text{m}^2$, $I_{zz} = 163827000 \text{ t}\times\text{m}^2$.

At the side boundaries of soil parallelepiped, which simulates soil basis, some special infinite elements were installed in order to absorb transient waves, and lower boundaries were restrained rigidly. Taking into account the above-mentioned and also from considerations that wave caused in soil from stamp oscillation must not be distorted because of reflection from boundary of soil parallelepiped at least during two oscillation periods, we took the following size of the soil parallelepiped: $L = B = 600 \text{ m}$, $H = 300 \text{ m}$.

Ballast cushion properties were taken as soil characteristics: $V_s = 500 \text{ m/s}$, $\rho = 2 \text{ t/m}^3$ and $\nu = 0.4$ and homogeneous soil was taken for development of the methodology.

It shall be pointed out, that requirement to have non-distortion of wave is necessary for application of logarithmic decrement method in the course of determination of equivalent damping. Taking into account the above-mentioned, a mathematic model was developed for soil parallelepiped, consisting of eight-node cubic elements of $10 \text{ m}$ size.

As a result of analysis with the use of ABAQUS the equivalent stiffness and equivalent damping presented in Table 1 were obtained for the stamp, vibrating on soil parallelepiped to simulate homogeneous elastic half-space. Here the relevant stiffness and damping are presented, having been obtained by impedance functions from ASCE.

### Table 1. Comparative analysis of equivalent stiffness and damping for homogeneous half-space.

<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Equivalent stiffness</strong></td>
<td><strong>Equivalent damping</strong></td>
</tr>
<tr>
<td></td>
<td>ASCE</td>
</tr>
<tr>
<td>$k_x$, $t/m$</td>
<td>$8.2\times10^6$</td>
</tr>
<tr>
<td>$k_y$, $t/m$</td>
<td>$9.2\times10^6$</td>
</tr>
<tr>
<td>$k_z$, $t/m$</td>
<td>$1.2\times10^7$</td>
</tr>
<tr>
<td>$k_{xx}$, $t\cdot m$</td>
<td>$8.5\times10^9$</td>
</tr>
<tr>
<td>$k_{yy}$, $t\cdot m$</td>
<td>$1.4\times10^{10}$</td>
</tr>
<tr>
<td>$k_{zz}$, $t\cdot m$</td>
<td>$1.4\times10^{10}$</td>
</tr>
</tbody>
</table>
In the Table 1 damping percentage in brackets were determined as share of the critical damping value.

From the comparative analysis of equivalent stiffness it is evident that analysis done with the use of ABAQUS for all the cases give results greater than the equivalent stiffness obtained in ASCE formulas for the semi-infinite space. One among reasons of higher results is that semi-infinite space was confined by actual size of soil parallelepiped. However if we increase this size, then decrease of the equivalent stiffness will be negligible (around 7%). In particular, $k_z$ will be $1.4 \times 10^7$ t/m. In such case it is require to develop mathematic model of soil for large size, but as a result we still have an insurmountable difference of around 15% between results with regard to the equivalent stiffnesses because of approximate nature of ASCE formulas.

Let us review in details method of coming to the equivalent damping given in Table 1. With this aim we studied free oscillations of the stamp after aircraft crash. Figure 4-9 demonstrate damping-time curves.

![Figure 3. General view of soil-structure system.](source)

![Figure 4. Damping at $u_x$.](source)

![Figure 5. Damping at $u_y$.](source)

![Figure 7. Damping at $u_{\phi_x}$.](source)

![Figure 8. Damping at $u_{\phi_y}$.](source)

Source: SIMULIA Community Conference, 2014
Using these curves we define damping logarithmic decrement as follows:

\[ \eta = \ln \frac{A_1}{A_2} = \frac{2\pi \xi}{1 - \xi^2}, \]  

(1)

where \( A_1 \) and \( A_2 \) - amplitudes of the first and second oscillations and \( \xi \) is relative damping. Here with we easily obtain:

\[ \xi = \frac{\eta}{\sqrt{4\pi^2 + \eta^2}}. \]  

(2)

Using Equation 2, the equivalent damping was obtained to be as follows:

\[ b_i = 2\xi_i k_i m_i, \]  

(3)

where \( i = x, y, z, \) \( \phi_x, \) \( \phi_y, \) \( \phi_z \) and \( m_i \) and \( \xi_i \) represent building mass and moment of inertia, and relative damping in the process of translational and rotational displacements, accordingly.

Let us alternatively review obtaining of frequency-dependent characteristics of stiffness and damping for this problem. For this, as initial impact to stamp we apply displacement, varying according the harmonic law with single amplitude, and frequency, varying from 0.5 Hz to 30 Hz through 0.5 Hz steps. As a result we come to dynamic reaction. Phase difference \( (\phi) \) between amplitude of reaction force \( (RF) \) and amplitude of displacement we define graphically. As a result, values of dynamic stiffness \( (k) \) and damping \( (b) \) for the fixed value of external impact frequency \( (\omega) \) was determined by the following formulas:

\[ k = RF \cos \phi \]  

(4)

\[ b = \frac{RF \sin \phi}{\omega}. \]  

(5)

Figure 10 and 11 demonstrate the dynamic stiffness and damping at vertical vibrations of the stamp dependent on frequency of driving force. As seen from Figure 10, stiffness at low oscillation frequencies \( (0.5 \text{ Hz}) \) is \( 1.4 \times 10^7 \text{ t/m} \) that perfectly fits earlier analyses done by static method.

In compliance with Figure 11, damping is \( 8.3 \times 10^5 \text{ t-s/m} \), which by 19% differs from the ASCE results. Swaying of oscillations at high frequencies is also seen in Figure 11. It happens due to FE rough mesh used in simulation of soil that ensures trustworthy solution at frequencies not greater than \( 5-10 \text{ Hz} \). However for this study such FE mesh is quite ample.

Source: SIMULIA Community Conference, 2014
On the basis of the study we can come to conclusion that method of static stiffness and logarithmic damping decrement gives trustworthy results, if to compare with ASCE formulas, and frequency-dependent characteristics of stiffness and damping, and can be applied for calculations of stiffness and damping in laminated soil basis after aircraft crash. This method is more preferable than application of frequency-dependent characteristics, since the resultant equivalent stiffness and damping can easily be applied in timing methods of the dynamic analysis. When we use frequency-dependent characteristics, it is constantly a question about transition of characteristics from frequency into timing space.

With regard to reasons of difference between results of calculations done by static stiffness and logarithmic damping decrement method and calculations according to ASCE, as it was mentioned above, the difference happens because of approximate nature of methods used to determine the equivalent stiffness and damping according to ASCE.

Let us review the methodological aspects of direct integration method applied in the dynamic analysis. The main equation of movement of soil-structure system looks as follows:

$$KU_i + C\dot{U}_i + M\ddot{U}_i = P(t)$$  \hspace{2cm} (6)

In Equation 6, $U_i$, $\dot{U}_i$ and $\ddot{U}_i$ are vectors of full displacements, velocities and accelerations of the system, and $P(t)$ is impact shown in Figure 2.

$K = K_1 + K_2$, where $K_1$ and $K_2$ are partial matrices of structure and soil stiffness. $C = C_1 + C_2$, where $C_1$ is partial damping matrix related to viscous friction in the system and calculated according to Rayleigh’s formula:

$$C_1 = \alpha M + \beta K$$  \hspace{2cm} (7)

and $C_2$ is partial acoustic damping matrix, which characterise energy release from oscillating building into soil. Values $K_2$ and $C_2$ consist of components of the equivalent stiffness and damping presented in Table 1:
\[ K_2 = \{ k_x, k_y, k_z, k_{\varphi_x}, k_{\varphi_y}, k_{\varphi_z} \} \quad \text{and} \quad C_2 = \{ c_x, c_y, c_z, c_{\varphi_x}, c_{\varphi_y}, c_{\varphi_z} \} \]. \[ M \] is the structure mass matrix.

Some details of this approach are set forth in [2].

2. DETERMINATION OF THE EQUIVALENT STIFFNESS AND DAMPING FOR LAMINATED SOIL BASE

Let us apply the above-described approach to determine the equivalent stiffness and damping for actual soil. General view of the soil-structure system is given in Figure 3.

Soil characteristics were determined based on the engineering-geological sections and tables with the calculated physical-mechanical and dynamic properties of soil. Based on these data the most significant layers were defined at depth up to 250 m. Based on the results of above studies the following sizes are obtained for soil parallelepiped under the building: \( L = B = 600 \text{ m} \) and \( H = 250 \text{ m} \).

Soil layers used in calculations feature the characteristics given in Table 2.

<table>
<thead>
<tr>
<th>EGE number</th>
<th>Soil description</th>
<th>Layer thickness, m</th>
<th>Density, ( \rho ), tonne/( \text{m}^3 )</th>
<th>Velocity of shear waves, ( V_s ), m/s</th>
<th>Poisson' s ratio, ( \nu )</th>
<th>Modulus of elasticity, ( E ), tonne/( \text{m}^3 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1b</td>
<td>Filled-up compacted sand-gravel soils</td>
<td>0.95</td>
<td>2.04</td>
<td>230</td>
<td>0.40</td>
<td>30000</td>
</tr>
<tr>
<td>5</td>
<td>Clays, lean clays</td>
<td>0.90</td>
<td>2.02</td>
<td>390</td>
<td>0.45</td>
<td>89000</td>
</tr>
<tr>
<td>6</td>
<td>Clays, lean clays</td>
<td>1.30</td>
<td>1.96</td>
<td>250</td>
<td>0.46</td>
<td>35000</td>
</tr>
<tr>
<td>B(8-a)</td>
<td>Sandy loams, fine sands</td>
<td>1.40</td>
<td>2.02</td>
<td>450</td>
<td>0.45</td>
<td>10300</td>
</tr>
<tr>
<td>9a</td>
<td>Sandy loams, fine sands</td>
<td>1.70</td>
<td>2.00</td>
<td>520</td>
<td>0.44</td>
<td>14100</td>
</tr>
<tr>
<td>9</td>
<td>Dusty sands, fine sands</td>
<td>4.80</td>
<td>1.93</td>
<td>660</td>
<td>0.42</td>
<td>224000</td>
</tr>
<tr>
<td>12,13</td>
<td>Pebble with sand filler</td>
<td>8.30</td>
<td>7.29</td>
<td>700</td>
<td>0.43</td>
<td>267000</td>
</tr>
<tr>
<td>15</td>
<td>Marl</td>
<td>2.50</td>
<td>2.21</td>
<td>900</td>
<td>0.58</td>
<td>470000</td>
</tr>
<tr>
<td>1a</td>
<td>Ballast cushion above groundwater level</td>
<td>4.55</td>
<td>2.25</td>
<td>340</td>
<td>0.38</td>
<td>680000</td>
</tr>
<tr>
<td>1a</td>
<td>Ballast cushion below groundwater level</td>
<td>6.50</td>
<td>2.31</td>
<td>500</td>
<td>0.43</td>
<td>154000</td>
</tr>
</tbody>
</table>

Source: SIMULIA Community Conference, 2014
The equivalent stiffness and damping were calculated for the stamp placed on the laminated soil base with the use of data given in this Table with help of ABAQUS according to the above-described methods. These data are given in Table 3.

<table>
<thead>
<tr>
<th>Equivalent stiffness</th>
<th>Equivalent damping</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k_x^c$, kN/m</td>
<td>$b_x^c$, kN s/m</td>
</tr>
<tr>
<td>$k_y^c$, kN/m</td>
<td>$b_y^c$, kN s/m</td>
</tr>
<tr>
<td>$k_z^c$, kN/m</td>
<td>$b_z^c$, kN s/m</td>
</tr>
<tr>
<td>$k_{phi}^c$, kN m</td>
<td>$b_{phi}^c$, kN s m</td>
</tr>
<tr>
<td>$k_{theta}^c$, kN m</td>
<td>$b_{theta}^c$, kN s m</td>
</tr>
</tbody>
</table>

Table 3. The equivalent stiffness and damping for laminated soil base.

As is seen from Table 3, all the equivalent stiffness for laminated soil is greater than the equivalent stiffness for ballast cushion given in Table 1. The reason is that under ballast cushion there are layers with higher stiffness properties.

Comparing equivalent dampings it is evident that all the damping components, except for $b_y$ calculated according to ABAQUS are lower than what is obtained according to ASCE. It shall be noted that the equivalent stiffness and damping indicated in Table 3 were obtained according to ABAQUS with the use of three-dimensional model of soil base with account of soil lamination. For determination of the equivalent damping we used curves of stamp oscillations on laminated soil, which are shown in Figure 12-17.

Source: SIMULIA Community Conference, 2014
3. SPECIAL COMPUTERS USED FOR ANALYSIS

As it was noted in the Introduction, creation of this paper would not have been feasible without the efficient cluster of JSC Atomenergoproekt, because multiple solutions of the large size problem with account of nonlinear factors were necessary.

This cluster consists of four nodes, servers DELL PowerEdge 2950 of the following configuration: 2 four-nuclear processors Intel Xeon 3 GHz with main memory of 64 Gbyte;

Infiniband bus is used for organization of operative inter-processor interaction between cluster nodes;

For storage of big volume of data generated in the course of calculations, cluster is connected to storage disc system EMC CX4-120 via bus FiberChannel.

Cluster is controlled via Gigabit Ethernet network. Operative system RedHat Enterprise Linux 4.6 operate in cluster nodes.

4. MATHEMATIC MODEL OF THE SOIL-STRUCTURE SYSTEM

In order to carry out dynamic analyses the spatial mathematic model of the building was developed with the account of its complicated spatial structure, with due respect for inertia, and in some cases stiffness properties of equipment. Soil with regard to AC was accounted through springs and dampers, which simulate the equivalent stiffness and damping. The mathematic model of the reactor building features the following:

- total number of finite elements (FE) is \( 78898 \);
- total number of nodes is \( 72114 \);
- total number of degrees of freedom is \( 429840 \).

The following FE types were used in the simulation:

- shell/plate element \( S4R \) for simulation of walls, floors and shells;
- beam element \( B31 \) for simulation of the reactor;
- concentrated mass element \( MASS \) for account of equipment components not developed in the model in details,
- spring element \( SPRING2 \) for account of soil stiffness,
- damper element \( DASHPOT2 \) for account of energy dispersion into soil.
- noteworthy that in this study the integral springs and dampers were connected to surface of the foundation slab accounted as rigid body, in its geometric centre.

Sections along row B and along axis 3 of finite-element scheme of the building are shown in Figure 18 and 19, and points of aircraft shocks are shown in Figure 20, 21 and 22.

Source: SIMULIA Community Conference, 2014
Figure 18. Fragment of finite-element scheme, section is cut along row B.

Figure 19. Fragment of finite-element scheme, section is cut along axis 3.

Figure 20. Points of aircraft shocks.

Figure 21. Points of aircraft shocks.

Figure 22. Points of aircraft shocks.

Source: SIMULIA Community Conference, 2014
For reinforced concrete structures there was used concrete $B50$ for outer structures (shells and building annex), $B25$ (extra-heavy concrete) for hermetic area and $B25$ for the other structures as well as Class $A500$ rebars. Characteristics of concrete are indicated in Table 4.

<table>
<thead>
<tr>
<th>Class of concrete</th>
<th>Modulus of elasticity (tonne/m$^2$)</th>
<th>Calculated concrete resistance $R_e$ (t/m$^2$) for axial tension</th>
<th>Reinforced concrete density (tonne/m$^3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$B50$</td>
<td>$3.9 \times 10^6$</td>
<td>$8.3 \times 10^3$</td>
<td>2.5</td>
</tr>
<tr>
<td>$B25$ (extra-heavy concrete)</td>
<td>$3.25 \times 10^6$</td>
<td>-</td>
<td>3.6</td>
</tr>
<tr>
<td>$B25$</td>
<td>$3.06 \times 10^6$</td>
<td>-</td>
<td>2.5</td>
</tr>
</tbody>
</table>

Rebars feature the following: pitch is 200 mm, diameter is 1.26 cm, elastic modulus is $2.1 \times 10^7$ t/m$^2$, Poisson’s ratio is 0.3, yield point is 50000 t/m$^2$, ultimate limit is 62000 t/m$^2$ when strain reaches 0.005. It shall be pointed out that in ABAQUS the longitudinal reinforcement is accounted as layers with the reduced characteristics of material.

5. MAIN RESULTS

The model of nonlinear strain of concrete ‘Brittle Cracking Model’ was applied only for outer surfaces of the structure, internals of the model were in elastic realm of strain. The Brittle Cracking Model was applied in ABAQUS/Explicit (calculation according to the explicit integration scheme) is recommended in case cracking occurs mostly during tension, and compression behaviour is of elastic nature. Functionality of the concrete model discussed here was checked in the test task. Plate of 25×25 m size, 0.5 m thick restrained along the perimeter was studied. Impulse caused by crash of Phantom was applied to the centre of plate, and timing characteristic of displacement was defined in the centre of this plate. Comparison between results of ABAQUS calculation and calculation done according to UDAR code [3] is shown in Figure 23. One can see good concurrence of results for nonlinear behaviour of material in Figure 23.

Figure 23. Comparative analysis of displacements obtained in both ABAQUS and UDAR within the test task.

Source: SIMULIA Community Conference, 2014
Response spectra in equipment locations

Figure 24. Non-hermetic area. Expanded enveloping response spectra caused by AC. Foundation slab. Elevations -7.200 m and -4.200 m in axes 1p-5p.

Source: SIMULIA Community Conference, 2014
Figure 25. ALA. Expanded enveloping response spectra caused by AC at elevation +17.100 m. Reactor bottom support ring.

Source: SIMULIA Community Conference, 2014
Figure 26: ALA. Expanded enveloping response spectra caused by RC at elevation +31.700 m. Except for R SCPF accumulators and pressurizer room.

As a result of calculation according to the explicit integration scheme, the response spectra were determined in equipment locations. In Figure 24, 25 and 26 comparisons of response spectra are given for 2% damping for three analyses:

- elastic concrete and laminated soil;
- elastic concrete and homogeneous soil;
- non-elastic concrete and laminated soil in case of shock 1 shown in Figure 21.

Source: SIMULIA Community Conference, 2014
Comparative review of results proves that accounting of non-elastic behaviour of concrete leads mainly to essential drop of spectral accelerations. In some cases some growth of spectra still occurs. It happens because of varying frequency composition of the structure in case of inelastic deformation of concrete, which causes resonances. The most conservative results are obtained for the homogeneous soil and elastic behaviour of concrete.

6. CONCLUSIONS

1. Here we propose the method of analysis of response spectra in civil structures of nuclear plant caused by aircraft crash with account of laminated soil basis and nonlinear strain of reinforced concrete in the shock area.

2. In the framework of this method the review of comparisons between the equivalent stiffness and damping of soil and the results obtained in ASCE, and review of comparisons between nonlinear strain of reinforced concrete obtained in ABAQUS and UDAR codes confirm veracity of the proposed approach.

3. Review of comparison between the resulting response spectra obtained with inelastic behaviour of concrete and laminated soil base and the response spectra obtained for elastic model of concrete and homogeneous soil basis proves that spectral accelerations decreases up to three times. Influence of soil lamination is also essential, since being compared with homogeneous soil (ASCE), spectral accelerations decreases down to 40%.

4. According to opinion of authors the presented approach based on static stiffness and logarithmic damping decrement for account of soil basis lamination can also be used in seismic analyses.

7. REFERENCES

Stress and fatigue analyses of a PWR reactor core barrel components

L. Mkrtchyan, H. Schau, H. Eggers (TÜV SÜD ET Mannheim, Germany)

Abstract: The integrity of the nuclear reactor core barrel and its components is one of the most critical issues of the reactor pressure vessel internals due to the unacceptable consequences of their failure and due to the difficulty of their replacement. Especially for such components thermal fatigue is one of the significant long-term degradation mechanisms, due to the fact that thermal loadings lead to most fatigue relevant stresses and strains.

In present paper nonlinear structural FE- analyses with subsequent fatigue evaluations are performed in order to predict crack initiation or failure of the reactor core barrel (RCB) components in spite of a failed bolt, revealed in one of the support brackets mounted to the RCB. The elastic-plastic analyses are aimed to determine the cyclic stresses in the support brackets, originated from constrained thermal deformations as well as to compute the cumulative usage factor by performing fatigue analyses according to the ASME Pressure and Vessel Code [2] and the corresponding German Nuclear Safety Standard KTA3204 [5].

The elastic-plastic analyses are performed with the help of the finite element program Abaqus in two separate models. Different possible boundary conditions are analyzed and the one yielding the largest stresses is used for further fatigue analysis and computation of the cumulative usage factor.

The investigations show, that the absence of the bolt has an influence on the neighbouring support brackets, leading to increase in stresses in those components. However, the stress evaluations according to the German nuclear safety standard KTA3204 [5] show that those increased stress values still lie within the safety limits. The fatigue cumulative usage factor is computed to have the value less than unity, which provides the safety of the reactor core barrel in spite of the damaged bracket.

Keywords: Fatigue prediction, cumulative usage factor, reactor barrel, thermal stresses, nonlinearities

1. INTRODUCTION

In the Class 1 components of operating nuclear power plants (NPPs) fatigue cycling may occur due to temperature variations. Temperature transients during operation can cause local or global temperature gradients, resulting in thermal cycling at the interface of material and environment. Smooth and sharp temperature transients result in slow or rapid thermal cycling, both being sources for accumulation of fatigue usage. Especially for these components, thermal fatigue is one significant long-term degradation mechanism, due to the fact that thermal loadings lead to most fatigue relevant stresses and strains.

Causes for smooth transients are generally start up and shutdown procedures or load following operation modes.

The nuclear plant design specifications, such as ASME Boiler and pressure Vessel Code [2] or the corresponding German Nuclear Safety Standard 3201.2 [4] identify that the design cycles for fatigue evaluation should be based on a 40-year life expectancy of the plant. This method ensures freedom from fatigue cracking during the mentioned period. Using the code fatigue curves, the cumulative usage factor must be computed based on estimated number of cycles for the postulated period, and has to be equal or less than unity.

In present paper fatigue assessment is made in order to predict crack initiation or failure of the mounting brackets, attached to the reactor core barrel (RCB) in the pressurized water reactor of an NPP.

Four irradiation channels are mounted to the outer wall of the mentioned RCB and each of them is clamped to the RCB with the help of 15 support brackets.

These brackets are mounted to the RCB with cheese head bolts and alignment pins.

Recently, during the inspection of the RCB with underwater camera, damage in a support bracket of one of the irradiation channels was discovered. The inspection revealed that one of the bolts of the support bracket No. 9 was broken and the spacer was missing.

Several studies following the inspection proved that the failure cause for the affected support bracket could not be the flow-induced vibrations [1]. Hence, further investigations were needed to study whether the cause of the bolt failure was fatigue induced damage and whether the lack of the mentioned bolt would affect the other support brackets and safety of the core barrel during the further operation of the NPP.

Source: SIMULIA Community Conference, 2010
In present paper detailed nonlinear structural analyses with subsequent fatigue evaluations are performed in order to predict failure of the reactor core barrel taking into account the failed bolt of the support bracket in the NPP. The evaluations are performed by determining the cyclic stresses appearing in the support brackets originated from constrained thermal deformations as well as the cumulative usage factor. The cumulative usage factor is determined by performing fatigue analyses according to the ASME Pressure and Vessel Code [2] and the corresponding German Nuclear Safety Standard KTA 3201.2, section 7.8 [4].

The study is performed with the help of finite element method (FEM) with the Software Abaqus [6].

The progressive increase in computational resources and the recent extensive FEM development has enabled the use of these codes in the mechanical calculation of class 1 components such as the “Design by Analysis” in nuclear calculation codes due to the high level of accuracy and reliability of these methods.

The effects of the thermal expansion of the RCB on the support brackets mounted to its outer wall were examined in two separate FE models by means of nonlinear elastic-plastic structural analyses. Based on results of the structural analyses the cyclic stresses were evaluated with the help of fatigue analyses according to the above-mentioned standard.

2. STRUCTURAL ANALYSIS OF THE RCB SUBJECTED TO THERMAL LOADINGS

The first step in the evaluation of the support brackets integrity is the assessment of thermal strains and stresses of the core barrel during each transient.

The reactor core barrel, the major structural member among the reactor internals, is a cylinder including an external ring flange at the top end and an internal ring flange at the lower end.

Due to radiation the RCB is heated up to 340°C. The heat, generated in the reactor is released to the primary coolant circuit, which consists of 4 identical parallel connected circuits. Water as a coolant is pumped through the reactor to carry away the heat. The coolant enters the reactor pressure vessel (RPV) at the temperature of 291.3 °C via 4 coolant inlet nozzles, flows downward through the downcomer space between core barrel and pressure vessel wall (outer wall of the RCB), is rotated through 180° and re-routed upwards through the reactor core, meanwhile warmed up to 328.3°C. The temperature of the surrounding coolant is variable from point to point causing inhomogeneous temperature distribution along the boundary surface of the core barrel.

During the loading and unloading cycles the temperature increases from 23 °C to the actual temperatures on the inner and outer surfaces of the core barrel and cause thermal expansion. This inhomogeneous expansion leads to cyclic thermal stresses and strains arising in the mounting regions of the support brackets of the irradiation channels, attached to the RCB.

The variable temperature values along the surface of the RCB are measured experimentally and consist of totally 2304 regions [1]. The temperature values are measured on both: the inner and outer walls of the RCB.

In order to obtain the thermal stresses and strains, at first the FEM model of the RCB is constructed in Abaqus. In this FEM model the RCB is examined under thermal expansion taking into account the measured inhomogeneous temperature distribution along the boundary, its inhomogeneous distribution through the thickness of the RCB wall as well as its linear increase and decrease in time during a loading and unloading cycle respectively.

The finite element model represents one loading and unloading cycle of the RCB. Due to the cosine shape temperature distribution along the inner and outer walls of the cylindrical barrel only one eighth model of it is considered. The barrel is modeled with the help of 2D shell elements of the type S4R.

The temperature distributions are applied along the wall of the barrel at the extent positions given in [1], modeling 297 different temperature regions for the one eighth of the model.

Also the variation of the temperature across the wall thickness has been taken into account in the model. For that purpose 2 different temperature points across the thickness have been defined in the model, at which the measured inner and outer temperature values have been prescribed. The variation has been taken to be linear between these points.

The shell of the core barrel is made of stainless steel type 1.4550, for which multi-linear elastic-plastic behavior with kinematic hardening was taken. The temperature-dependent stress-strain curves are shown in Figure 1.

Source: SIMULIA Community Conference, 2010
The mentioned curves represent the true stress versus true strain curves, which have been obtained from the experimental curves with the following formulae:

$$
\varepsilon = \ln(1 + \varepsilon_e) \quad \sigma = \sigma_e(1 + \varepsilon_e)
$$

From the elastic-plastic analysis of the shell with the account of the geometric non-linearities the thermal stresses and strains during the loading-unloading cycle are computed at the nodes where the bolts of the support brackets are mounted. The temperature distribution along the outer wall of the shell is shown in Figure 2.
The calculation results reveal the expected increase in axial displacement downwards the barrel. The obtained different thermal strains at different mounting brackets may cause fatigue damage and their influence is studied further in the second FEM model.

The second FEM model represents the model of the irradiation channel together with its support brackets (see Figure 4), with the help of which the channel is attached to the RCB.

The displacements at the attachment points of the mounting brackets which are obtained from the first FEM model of the RCB, subjected to thermal loads are used further to model the influence of the RCB on the support brackets of the irradiation channel.

3. THE IRRADIATION CHANNEL WITH 15 SUPPORT BRACKETS

The irradiation channel essentially consists of a long pipe with 15 support brackets, which are mounted to the shell of the RCB. This is done with the help of cheese head bolts and alignment pins that are screwed into the holes tapped into the RCB shell. The bolts are preloaded and made of Type 1.4571 stainless steel.

Depending upon geometry of the support brackets two types of mountings are used: loose point (with 2 hold-down bolts) and fixed point (with 4 bolts and 2 alignment pins) mountings. The alignment pins in the fixed point mounting brackets serve for the positive locking of the bolts and the shell.

The bracket, where the broken bolt has been found, represents a loose-point mounting bracket and is the ninth one, counting from the top. The loose-point and fixed-point brackets with different geometries are shown in Figure 3.

![Figure 3. Two types of support brackets, a) fixed-point, b) loose-point](image)

For determination of the fatigue level of the support brackets, several elastic-plastic nonlinear structural analyses with subsequent fatigue analyses are performed according to the ASME Boiler and Pressure Vessel Code [2] and the corresponding German Nuclear Standard KTA 3204 [5].

![Figure 4. Mises - stresses in the model without boundary condition on the bracket Nr. 9](image)

Source: SIMULIA Community Conference, 2010
For that purpose the 3D-model of the irradiation channel together with its 15 support brackets is constructed in Abaqus. The support brackets as well as the irradiation channel are modeled with solid elements of type C3D8R. Surface-to-surface tied contact is defined between the brackets and the channel.

Due to the symmetry of the geometry and the loading, it is sufficient to model just the half of the channel in the FEM (see in Figure 4).

The material (stainless steel of Type 1.4550) of the support brackets is modeled to have multi-linear elastic-plastic behavior with kinematic hardening. Similarly, as in the above-discussed thermal analysis of the RCB shell, true elastic-plastic stress-strain curves are used to model the behavior of the material. The irradiation channel is assumed to be elastic.

The thermal displacements, determined in the first model, which are transferred to the bolts and the alignment pins, are taken into account in the second model. They are applied linearly in the temperature interval from 20°C to 291.3°C, the latter representing the temperature of the cold loop. One close loading-unloading cycle is modeled.

In order to reveal the influence of the broken hold-down bolt on the neighbouring ones and also on the RCB, 8 different boundary conditions are analyzed and for each of them an elastic-plastic analysis is performed considering a close loading-unloading cycle. The considered four boundary conditions cover all the possible cases when the bolts still hold in different directions.

In the considered boundary conditions the alignment pins of the fix-point bolts are fixed in ‘x’ and ‘y’ directions, while being free to move along the ‘y’ direction. For the bolts of the fix-point brackets as well as for the hold-down bolts of the loose-point bracket four cases are considered. In BC1 the bolts are loose in both directions, BC2: the bolts are loose in ‘y’ direction, BC3: they are loose in ‘x’ direction, and BC4: they hold in both directions. These boundary conditions with the account of the damaged bracket are given in Table 1.

<table>
<thead>
<tr>
<th>Boundary Conditions</th>
<th>BC1</th>
<th>BC2</th>
<th>BC3</th>
<th>BC4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alignment pins of the fix-point brackets</td>
<td>$u_x$, $u_y$</td>
<td>$u_x$, $u_y$</td>
<td>$u_x$, $u_y$</td>
<td>$u_x$, $u_y$</td>
</tr>
<tr>
<td>Bolts</td>
<td>$u_y$</td>
<td>$u_x$, $u_y$</td>
<td>$u_y$, $u_z$</td>
<td>$u_x$, $u_y$, $u_z$</td>
</tr>
</tbody>
</table>

The other four boundary conditions are similar to those, given above. The only difference is that they are not valid for the defective support bracket Nr. 9, simulating the case of missing bolts for this bracket.

The finite element analyses show that at different boundary conditions increase in stresses and strains occur when the defective bracket is not mounted to the RCB shell. Especially the brackets, neighboring the defective one are affected and show larger stress values (increase with 1%) as compared with the case when the bolts of the defective bracket still hold.

For most boundary conditions the maximal strain values are reached in the fourteenth support bracket at boundary condition 1 with missing bolts for bracket No.9.

The computed cyclic strains from the FEM model in the worst case are used for the subsequent fatigue analyses. In Figure 5 the maximum principal logarithmic strain in the worst case is shown, for the case when the defective bracket is missing, yielding larger strains in the neighboring ones.

Source: SIMULIA Community Conference, 2010
4. FATIGUE ANALYSES

In order to evaluate the long-time fatigue life of the core barrel and the irradiation channel with its support brackets, the accumulated fatigue damage is calculated according to ASME standard [4].

If the material is subjected to $m$ different cycles of frequency $n_i$ and corresponding to stress ranges $S_i$ ($i=1,2,...,m$) then the cumulative usage factor $D$, is given by

$$D = \frac{n_1}{N_1} + \frac{n_2}{N_2} + ... \frac{n_i}{N_i} \leq 1$$

where $D$ is the cumulative usage factor, $N$ - the number of design allowable fatigue cycles at a given total equivalent strain range and temperature, $n_i$ - the specified number of cycles.

Safety from fatigue requires: $D \leq 1$.

The fatigue analysis is based on the determination of the half of the equivalent strain range $\varepsilon_a$ which is defined with respect to changes in the strain components from the starting point to each point in the cycle.

From the amount of the considered loading conditions two appropriate ones are chosen in such a way, that the equivalent strain computed from the difference of the corresponding strains, becomes maximum. This maximum represents the range of the equivalent strain (KTA 3201 Section 7.7.3.3 [8]):

$$\varepsilon_a = \frac{\sqrt{2}}{2(1 + \nu)} \left[ (\Delta \varepsilon_x - \Delta \varepsilon_y)^2 + (\Delta \varepsilon_y - \Delta \varepsilon_z)^2 + (\Delta \varepsilon_z - \Delta \varepsilon_x)^2 + \frac{3}{2} (\Delta \gamma_{xy}^2 + \Delta \gamma_{yz}^2 + \Delta \gamma_{zx}^2) \right]^{\frac{1}{2}}$$

In our case the Poisson’s ratio is assumed to be $\nu = 0.3$ also in the plastic region.

The half of the effective or equivalent stress range $S_a$ is obtained by multiplying the equivalent strain range with the elasticity modulus $E$:

$$S_a = 0.5 E \varepsilon_a$$

Source: SIMULIA Community Conference, 2010
Whereas the elasticity modulus is taken the one for the temperature $t = 300^\circ C$.

The occurring strains in the bracket No. 14, which are calculated with the help of elastic-plastic nonlinear analyses, are used in the latter fatigue analysis study. In this bracket the stresses and strains reach their highest values.

For each loading-unloading cycle with the account of the equilibrium temperature, the following ranges of the equivalent strains have been calculated for different boundary conditions (see Table 2).

In the present study the largest value of the equivalent strain range is computed to be at the support bracket Nr. 14 and occurs at the boundary conditions BC1 with missing bolts for the support bracket Nr. 9. It has the value 0.0091.

The design fatigue curve for the material 1.4550 is obtained from [5] and the number of the admissible load alternations is determined. In the considered case the maximum equivalent strain value corresponds to an admissible load alternation of 791.

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>$\varepsilon_a$ [1]</th>
<th>$S_a$ [MPa]</th>
<th>$f_i$ [1]</th>
<th>$D_{per}$ Cycle *E-03</th>
</tr>
</thead>
<tbody>
<tr>
<td>BC1</td>
<td>0.0098</td>
<td>809.41</td>
<td>794</td>
<td>1.26</td>
</tr>
<tr>
<td>BC2</td>
<td>0.0090</td>
<td>804.16</td>
<td>810</td>
<td>1.23</td>
</tr>
<tr>
<td>BC3</td>
<td>0.0083</td>
<td>742.18</td>
<td>1046</td>
<td>0.96</td>
</tr>
<tr>
<td>BC4</td>
<td>0.0079</td>
<td>708.96</td>
<td>1223</td>
<td>0.82</td>
</tr>
<tr>
<td>BC1 without s. b. Nr.9</td>
<td>0.0110</td>
<td>810.53</td>
<td>791</td>
<td>1.26</td>
</tr>
<tr>
<td>BC2 without s. b. Nr.9</td>
<td>0.0090</td>
<td>803.87</td>
<td>811</td>
<td>1.23</td>
</tr>
<tr>
<td>BC3 without s. b. Nr.9</td>
<td>0.0083</td>
<td>742.62</td>
<td>1044</td>
<td>0.96</td>
</tr>
<tr>
<td>BC4 without s. b. Nr.9</td>
<td>0.0079</td>
<td>710.03</td>
<td>1217</td>
<td>0.82</td>
</tr>
</tbody>
</table>

For the mentioned boundary condition, which appears to be the most critical concerning the number of load alternations among those 8, further fatigue analysis is performed and the cumulative usage factor is computed (See Table 2).

The fatigue cumulative usage factors are computed for the time periods of warming up and down of the core barrel, during operation period from 1985 to 2007 and extrapolated for the period 2007-2025. During the warming up of the RCB 10 loading and unloading cycles are registered. The given 39 loading cycles in the period of 1985-2007 are also considered for the usage factor calculation. Furthermore, these loading cycles are extrapolated for further 17 years and the conservative total fatigue value is computed to have the value: 0.1 which does not appear to be critical neither for the shell nor for the support brackets. Hence, the failure of the damaged support bracket is not expected.

**5. EVALUATION OF THE BOLTS AND THE ALIGNMENT PINS**

Besides the support brackets, also the bolts and the pins require a detailed stress and fatigue evaluation which is out of the scope of the present study.

However, for different boundary conditions for the support brackets of the irradiation channel also the tensile and shear forces were evaluated at the nodes of the bolts and the alignment pins.

The consideration of stresses in the bolts is that thermal loads applied to the bolts are classified as primary loads. The reason for this classification is because these loads, although secondary in the containment, are considered to be applied non-self-limiting loads for the bolts. Hence, the stresses in the bolts are compared to the allowable stress $S_{m}$.

For the stainless steel Type 1.4571, from which the bolts are fabricated, the allowable stress is $S_{m,300} = 131.8 \text{ MPa}$ at 300 °C.

Source: SIMULIA Community Conference, 2010
The maximum shear stress for the alignment pins is computed for the boundary condition BC2 without support bracket Nr9 and has the value of 39.7 MPa. This value lies below the allowable stress, since

$$\tau_{\theta} = \sqrt{3} \tau_{\text{max}} = 68 \text{ MPa} \leq S_{m300}$$

The computed tensile stress in the bolts reaches its maximum at the boundary condition 2 and has the value of $\sigma_z = 30.3 \text{ MPa}$ and in the hold-down bolts: $\sigma_z = 22.2 \text{ MPa}$, which still lie within the allowable limits.

The FEM analyses show that the forces in the neighboring brackets increase for the studied cases, when the bolts at bracket are missing. However, the computations show that failure of further bolts and pins is not expected.

6. CONCLUSIONS

Degradation of core internal structures usually involves increases in clearances due to wear or misalignment, or loss of clamping forces. While these changes are inherently small, significant degradation can occur through thermal fatigue of large structures. This type of degradation may be difficult to detect visually before the structure’s function or integrity are compromised.

The elastic-plastic fatigue analyses of the irradiation channel together with its 15 support brackets show that the fatigue induced stresses due to their low values will have no influence on the safe operation of the NPP. No damage of the core barrel shell and its remaining support brackets due to fatigue and thermal ratcheting or failure of the defective bracket itself are expected. The absence of the bolt has an influence on the neighbouring support brackets, leading to increase in stresses in those components. However, the stress evaluations according to the German nuclear safety standard KTA3204 [5] show that those increased stress values lie still within the safety limits.

7. REFERENCES

3. Section III, Division 1, Subsection NB, Class 1 components
5. KTA3201.2 Fassung 04.2009 „Komponenten des Primärkreises von Leichtwasserreaktoren“
8. Abaqus CSE Version 6.9, 2009. Dassault Systèmes, RI, USA
Realistic Simulation Assists Nuclear Power Plant Certification

From the onset of the civilian nuclear era, there has been a strong awareness of the importance of safety within the nuclear energy industry. Experts have devoted much time and effort to ensuring the integrity of reactor cores and facility containment.

Global cooperation on nuclear safety issues is widespread. The U.N.’s International Atomic Energy Agency (IAEA) has established mandatory benchmarks for nuclear plant siting, design, construction, operation, resourcing, assessment, and verification of safety, quality assurance, and emergency preparedness. All countries with operating nuclear power facilities are expected to bring their plants up to the latest IAEA standards.

AGING NUCLEAR FACILITIES

An integral part of reactor safety assurance is the mitigation of facility aging. Designed for 30- to 40-year operating lives, the systems, structures, and components of nuclear plants can change with time and use. Components can wear out, corrode, or degrade; instrument and control systems may become obsolete as technologies evolve. Complicating the issue, the properties of critical materials may change through heat and neutron irradiation.

Identifying and correcting longevity issues can extend the operating license of a plant by several decades, which is why upgrading older facilities is a major focus of nuclear regulatory bodies and plant operators. In addition, new facilities are held to the highest standards of quality to ensure a lifetime of safe operation.

“The structural integrity and operational management of nuclear facilities must be secured far into the future—whatever the type or age of the plant,” says Wolfgang Hienstorfer, head of the department of structural analysis at TÜV SÜD ET, a leading global technical service corporation in Filderstadt, Germany.

Hienstorfer’s team independently tests, inspects, and certifies nuclear facilities for licensing by the German government. He is also chairman of the advisory group on nuclear facility aging management to Germany’s Nuclear Safety Standards Commission, and a technical consultant to the IAEA on nuclear facility aging. Many of his recommendations developed during his work at TÜV have been incorporated into existing international standards.

“On behalf of the regulatory bodies, we encourage the power utilities to follow the latest relevant research findings whether they are maintaining an older plant or designing and building a new one,” says Hienstorfer.

FEA ASSISTS SAFETY EVALUATION

To assist in the evaluation of nuclear plant integrity, Hienstorfer’s group employs Abaqus FEA software. “Abaqus is a very useful and powerful tool for many aspects of our work,” says Hienstorfer. “The processes of sensitive industrial facilities are very complex, and FEA helps us evaluate the safety margins in a more sophisticated way.”

TÜV uses Abaqus to analyze stress loads over a wide range of scenarios such as rapid temperature and/or pressure changes, airplane impact, earthquakes, and radiation embrittlement. The software is used to analyze everything from key mechanical components—including pumps, piping systems, vessels, supports, and tanks—to fuel assemblies, building structures, and lifting devices.

STRICT STANDARDS FOR NUCLEAR REACTORS

An ongoing focus of regulators is the reactor pressure vessel (RPV), the steel “heart” of the power plant that houses the nuclear fuel rods (Figure 1). A nuclear power plant using fission to produce steam that drives electric generators is subject to temperature and pressure stresses similar to those at any kind of steam facility. But the possibility of pressurized thermal shock (PTS) affecting a radiation-embrittled RPV is unique to the nuclear industry: bombardment from neutrons can, over time, alter the molecular makeup of the metal from which an RPV is built, making the vessel more prone to structural
damage under stress. In a classic loss-of-coolant (LOC) scenario, a broken pipe in the primary system deprives the reactor core of vital coolant, and the hot vessel (300°C) is then subjected to extreme PTS as colder water (at 30°C) is rapidly piped into the vessel to cool the core and shut the reactor down.

IAEA standards require that RPVs have a proven ability to withstand this kind of event in order to receive certification for operation. “You have to document the damage tolerance of the systems, structures, and components of a plant to pass inspection,” says Hienstorfer. “FEA is integral to that analysis. FEA can be used for virtual testing to provide guidance for new designs in the early stages of product development, as well as for performance assessment of existing components under simulated stress conditions.”

A typical FEA analysis of an RPV takes into account temperature transients, internal pressure fields, and radiation embrittlement behavior of the vessel during a simulated LOC event. The simulations examine stresses at vessel walls and entry points of the hot and cold water nozzles feeding into the RPV.

MODELING AN RPV WITH ABAQUS

To create their FEA models, TÜV engineers first obtained component condition data for the vessel and nozzles from nondestructive x-ray and/or ultrasound testing. Every vessel is plant-specific—in the case described here, the material was ferrite steel coated with austenitic cladding to protect the load-carrying ferrite layer from corrosion. Embrittlement of the metal over time was represented by end-of-life calculations based on existing data from irradiated material.

Next, Abaqus/CAE was used to build and mesh computer models of the vessel and the four water pipe nozzles that fed into it. Using larger, linear hexahedral elements reduced computation time for solving the global model (Figure 2), while smaller, quadratic hexahedral elements were used in the submodels (Figure 4) for more accurate depiction of stresses at the edges of nozzles.

SIMULATING PRESSURIZED THERMAL SHOCK

The TÜV team then used Abaqus/Standard for linear elastic simulation of the rapid cooling of the vessel, calculating the effects of a large increase in tensile stresses on the inner vessel wall. This increase is the result of two phenomena. First, the thermal conductivity of the two materials is different, so each reacts differently to the rapid temperature change. Second, the emergency injection of colder water creates a temperature plume that produces stress buildup at its leading edge (Figure 3).

The effect of the high pressures under which the system would operate was also incorporated into the models; an elastic/
plastic Abaqus simulation predicted where the greatest surface and/or volumetric stresses would occur in the system. The simulations were run beyond the required tolerance levels to the point at which cracking would occur. Such data is useful for fracture mechanics analyses, and can be used in the future by inspectors, says Hienstorfer.

**FEA FACILITATES REGULATORY COMPLIANCE**

The RPV in this example passed TÜV’s simulation testing, indicating that its walls and nozzles would withstand the extreme conditions of an LOC event over a 40-year lifespan. “The Abaqus FEA calculations helped evaluate compliance of the vessel to regulatory safety requirements,” says Hienstorfer.

Successful design, development, and maintenance of nuclear power facilities are challenges that must be managed from both an organizational and an engineering viewpoint, says Hienstorfer. He sees FEA as playing an integral role in both operational evaluation and ongoing monitoring of nuclear facilities to help comply with regulations designed to ensure the world’s growing energy needs can be met safely.

“We depend on FEA for computer modeling and virtual testing of reactor pipelines, vessels, and materials under extremes of stress and time,” he says. “It definitely provides guidance to engineers building safety and longevity into their nuclear power plant designs.”
Our 3DEXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the 3DEXPERIENCE® Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes’ collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 190,000 customers of all sizes in all industries in more than 140 countries. For more information, visit www.3ds.com.