

T. Hajas, P. Pandazis, L. Lovász and B. Babcsányi

New finite element-based modeling of reactor core support plate failure

ATHLET-CD is the severe accident module of the code system AC² that is designed to simulate the core degradation phenomena including fission product release and transport in the reactor circuit, as well as the late phase processes in the lower plenum. In case of a severe accident degradation of the reactor core occurs, the fuel assemblies start to melt. The evolution of such processes is usually accompanied with the failure of the core support plate and relocation of the molten core to the lower plenum. Currently, the criterion for the failure of the support plate applied by ATHLET-CD is a user-defined signal which can be a specific time or process variable like mass, temperature, etc. A new method, based on FEM approach, was developed that could lead in the future to a more realistic criterion for the failure of the core support plate. This paper presents the basic idea and theory of this new method as well as preliminary verification calculations and an outlook on the planned future development.

Neue Finite-Elemente-basierte Modellierung der Reaktorgitterplatte. ATHLET-CD ist ein Modul des Codesystems AC², welches entwickelt wurde, um die Phänomene während eines schweren Störfalls zu simulieren: Kernzerstörung, Spaltproduktfreisetzung und Transport sowie Phänomene im unteren Plenum. Während eines schweren Störfalls können die Brennstäbe abschmelzen, so dass die dadurch entstehende große thermische und mechanische Belastung nach einer gewissen Zeit auch zum Versagen der Kerngitterplatte führen kann. Wegen dieses Versagens verlagert sich der angesammelte Schmelze in das untere Plenum. In der aktuellen Version von AC² ist das Versagen der Kerngitterplatte durch ein benutzerabhängiges Temperatur- oder Zeitsignal gesteuert. Eine neue Methode, basierend auf dem FEM-Ansatz, wurde entwickelt, die in Zukunft das Versagen der Kerngitterplatte realistischer berechnen kann. In diesem Artikel werden die Grundidee und die ersten Testrechnungen der neuen Methode präsentiert und das weitere Vorgehen dargestellt.

1 Introduction

Understanding the phenomena that can occur during a nuclear accident is very important. Experimental and theoretical research works have been performed for many decades; most of them are still ongoing. Simulation tools were developed in order to model the behavior of a nuclear power plant during an accident. ATHLET-CD is the severe accident module of the code system AC² that can simulate the early and late phases of a nuclear severe accident. The characteristic process of the transition between the early and late phases of a nuclear severe accident is the relocation of the molten material from the core to the lower plenum. This relocation process is

significantly influenced by the condition of the core support plate.

Despite of the important role of the core support plate its failure is currently simulated in AC² via a user-defined signal. A more detailed model could predict the failure of the core support plate with higher accuracy. With the help of more accurate simulations further improvement of severe accident prevention and mitigation methods would become possible.

The research work presented in this paper laid down the foundation of a more detailed model that can simulate the failure of the core support plate. A detailed standalone program was developed that is able to calculate core support plate deformation caused by different mechanical and thermal loads. With further improvements this model could complement the currently existing user-defined failure criterion for the core support plate in AC². This paper presents the theory of the new model and the results of preliminary verification calculations.

2 Finite element-based model for the core support plate failure

The in-vessel core melt progression including the core degradation and followed by the relocation of the molten material results in a large thermal and mechanical load on the core support plate. Analytical examination of the deformation and prediction of the support plate failure with its relatively complex geometry is not a straightforward process or rather an impossible option. Therefore, in this study a finite element-based numerical approach was applied to define failure criteria depending on the heat and mass load distribution on the support plate.

2.1 Geometry of the core support plate

The future aim of the developed model is to accurately simulate the support plate failure of a real reactor core, therefore the mechanical load calculation model was based on a typical reactor support plate geometry. The generally applied material of reactor core support plates is austenitic stainless steel. The core support plate of a generic German PWR has a cylindrical shape with a thickness (t) of 340 mm and a radius (R) of about 1700 mm. The perforated core plate has around 156 cylindrical holes with a diameter (d) of 133 mm. The considered geometry is shown on Fig. 1. The typical number that characterizes the geometry is defined as $2R/t$ which is ~ 10 in the case of the target geometry.

2.2 The Reissner-Mindlin plate theory

There are several unknown parameters regarding the core support plate deformation during a severe accident. In the

present investigation low strains and small displacements are assumed with linearly elastic, homogeneous and isotropic material behavior. Based on previously presented geometric parameters (i.e. in the typical number range for the target geometry) the investigated core support plate can be treated as a “thick plate” [2–4]. The theoretical description of the assumed “thick plate” problem is based on the Reissner-Mindlin [5] plate theory that has the following kinematical assumptions:

1. The normal vector ($\mathbf{n}$) of the mid-surface of the plate does not stretch after the deformation.
2. The normal vector ($\mathbf{n}$) of the mid-surface of the plate stays straight after the deformation.
3. The material points of the mid-surface of the plate only move in the vertical direction.

The above assumptions that are based on the Reissner-Mindlin theory lead to the following form for the displacement field in the plate [5]:

$$\underline{u}(\underline{r}) = \begin{bmatrix} u(\underline{r}) \\ v(\underline{r}) \\ w(\underline{r}) \end{bmatrix} \approx \begin{bmatrix} z * \theta_x(x, y) \\ z * \theta_y(x, y) \\ w_0(x, y) \end{bmatrix} \quad (1)$$

In Eq. (1) $\underline{r}$ is the coordinate vector, u and v are the in-plane displacements of the mid-surface, while w is the mid-surface displacement in the z direction. θ_x and θ_y designate the angles which the normal to the mid-surface makes with the z axis, therefore θ_x is the rotation of the mid-surface in xz direction, while θ_y is the same but in the yz direction. Based on the assumptions of the theory, the zz components of the strain and stress tensors, i.e. the components that act normally to the mid-surface are zero. The geometry equation [5–7] describes the relation between the strain and the displacement fields as follows:

$$\underline{\varepsilon}(\underline{r}) = \begin{bmatrix} \varepsilon_{xx}(\underline{r}) \\ \varepsilon_{yy}(\underline{r}) \\ \gamma_{xy}(\underline{r}) \\ \gamma_{xz}(\underline{r}) \\ \gamma_{yz}(\underline{r}) \end{bmatrix} = \begin{bmatrix} \frac{\partial u(\underline{r})}{\partial x} \\ \frac{\partial v(\underline{r})}{\partial y} \\ \frac{\partial u(\underline{r})}{\partial y} + \frac{\partial v(\underline{r})}{\partial x} \\ \frac{\partial u(\underline{r})}{\partial z} + \frac{\partial w(\underline{r})}{\partial x} \\ \frac{\partial v(\underline{r})}{\partial z} + \frac{\partial w(\underline{r})}{\partial y} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \end{bmatrix} \begin{bmatrix} u(\underline{r}) \\ v(\underline{r}) \\ w(\underline{r}) \end{bmatrix} \quad (2)$$

In Eq. (2) $\underline{\varepsilon}$ denotes the strain vector of which only the axial components ε_{xx} and ε_{yy} are considered, while γ components denote the shear angle in the given xy , xz and yz directions. Based on Eq. (2) the strain vector can be determined. Depending on the temperature the strain induces different kind of stresses in the material. To describe the temperature-dependent stress vector the generalized Hooke's law [5–7] was applied (see Eq. (3)).

$$\underline{\sigma}(\underline{r}) = \begin{bmatrix} \sigma_{xx}(\underline{r}) \\ \sigma_{yy}(\underline{r}) \\ \sigma_{xy}(\underline{r}) \\ \sigma_{xz}(\underline{r}) \\ \sigma_{yz}(\underline{r}) \end{bmatrix} = \begin{bmatrix} \frac{E(T)}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} & \begin{bmatrix} 0 & 0 \\ 0 & 0 \\ 0 & 0 \end{bmatrix} \\ \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} & \frac{E(T)}{2*(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \end{bmatrix} * \underline{\varepsilon}(\underline{r}) \quad (3)$$

In Eq. (3) the σ components are the elements of the stress tensor, $E(T)$ is the temperature-dependent Young's modulus and ν is the Poisson ratio.

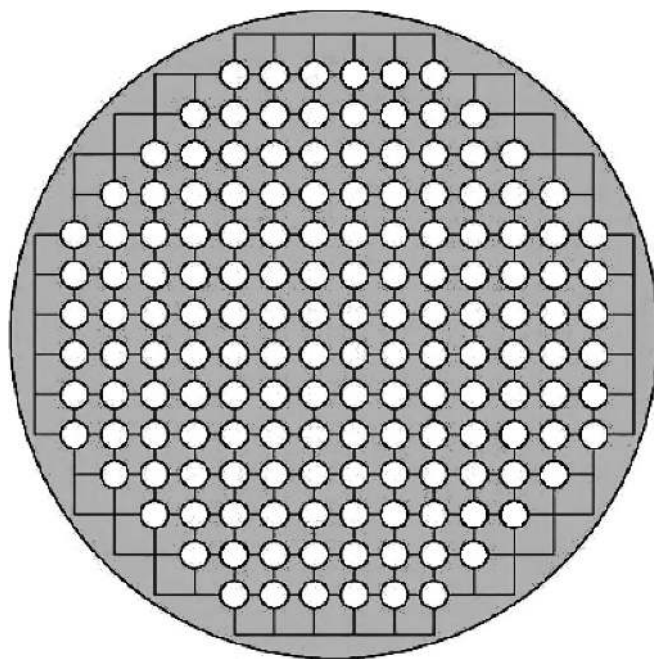


Fig. 1. Typical core support plate geometry considered in the mechanical model development [1]

2.3 The principle of minimum total potential energy

The calculation of the displacement field is based on the theory of minimum total potential energy. If there is a linear elastic solid supporting external distributed loads which is creating internal stresses and strains in the volume, and displacements on the surface (in the examined case on the mid-surface) of the solid, one can define several physical quantities, including the so called total potential energy function (π) that is given by [6, 7].

$$\pi(\underline{r}) = U(\underline{r}) - W(\underline{r}) \quad (4)$$

In Eq. (4) U is the internal strain energy, while W is the virtual work of the external loads and forces leading to deformations.

$$U(\underline{r}) = \frac{1}{2} \iiint_V \underline{\sigma}^T(\underline{r}) * \underline{\varepsilon}(\underline{r}) dV \quad (5)$$

$$W(\underline{r}) = \iiint_V \underline{u}^T(\underline{r}) * \underline{q}(\underline{r}) dV + \iint_A \underline{u}^T(\underline{r}) * \underline{p}(\underline{r}) dA \quad (6)$$

In Eq. (6) $\underline{q}(\underline{r})$ and $\underline{p}(\underline{r})$ are the volume and surface load vectors of which the latter can be obtained as the product of the surface stress vector and the surface normal vector. There are several displacements that are consistent with the given loads and reactions, but among them the so called correct state of displacement is that which minimizes the total potential energy. The variation of the total potential energy can be considered as the result of the difference between the consistent displacements and the correct state of displacement [6].

2.4 The finite element discretization

By spatially discretizing the analyzed geometry, M number of elements are generated. According to the principle of the total potential energy, the sum of the potential energy of each element gives the total potential energy of the analyzed sys-

tem. In the following paragraphs the theoretical overview is limited to an arbitrary element e . The correct state of displacement (i.e. the solution of the strain-stress-displacement calculations) – based on the applied finite-element theory – can be approximated by the combination of the nodal displacement coefficient values (u_b, v_b, w_b) and the element specific shape functions (f_i) as follows:

$$\underline{u}(r) \approx \begin{bmatrix} \sum_{i=1}^N u_i f_i(x, y) \\ \sum_{i=1}^N v_i f_i(x, y) \\ \sum_{i=1}^N w_i f_i(x, y) \end{bmatrix} \quad (7)$$

$\underline{u}_e$ is a hyper vector containing the nodal displacement coefficients (u_b, v_b, w_b) of the displacement components according to Eq. (7) for all the nodes N within the element e . Hereinafter bold letters designate hyper vectors and hyper matrices being present and operating in the coordinate and the nodal space. The dimension of this hyper vector equals to the dimension of the displacement field (three) times the number of nodes N within an element (the latter depends on the type of the applied elements within the spatial discretization process). In the followings the element-wise potential energy (π_e), stiffness matrix ($\underline{K}_e$) and force vector ($\underline{F}_e$) are defined. The potential energy is a scalar value, while the dimension of the stiffness matrix is $3N \times 3N$ and that of the force vector is $3N$. Eq. (8) describes the relation between the mentioned values [6]:

$$\pi_e = \frac{1}{2} \underline{u}_e^T * \underline{K}_e * \underline{u}_e - \underline{u}_e^T * \underline{F}_e \quad (8)$$

Based on the principle of the total potential energy, the total potential energy of the complete system – without considering interaction between the elements – can be calculated as the sum of the potential energy of each element [6]:

$$\pi = \sum_{e=1}^M \pi_e = \frac{1}{2} \underline{U}^T * \underline{K} * \underline{U} - \underline{U}^T * \underline{F} \quad (9)$$

In Eq. (9) $\underline{U}$ vector contains the nodal displacement values for the whole system (i.e. it is the global variable with the unknown displacement coefficients), $\underline{K}$ is the global stiffness matrix and $\underline{F}$ is the global force vector. The necessary criterion for the minimization of the total potential energy based on variation calculus is that the first variation of the total potential energy equals to zero as follows [6]:

$$\delta\pi = 0 = \delta \left(\frac{1}{2} \underline{U}^T * \underline{K} * \underline{U} - \underline{U}^T * \underline{F} \right) = \underline{K} * \underline{U} - \underline{F} \quad (10)$$

Taking into account specific boundary conditions as well, the displacement field of the system can be obtained according to the following equation:

$$\underline{U} = (\underline{K} + \underline{K}_{\text{boundary}})^{-1} * (\underline{F} + \underline{F}_{\text{boundary}}) \quad (11)$$

The spatial discretization can be performed by using triangular prisms for example, where the elemental volume can be defined as $V = z x \Omega_e$; $z \in \left[-\frac{t}{2}, \frac{t}{2}\right]$, $(x, y) \in \Omega_e$, and t is the height of the triangular prism. According to this definition Ω_e is a two dimensional triangular element. In the present study, the unknown variables depend only from x and y (see Eq. (1)) thus it is enough to model only the mid-surface of the geometry (i.e. that of the targeted core support plate geo-

metry). For stress calculations the application of isoparametric triangular elements (six-node triangles) is suggested [8] by approximating the unknown variables in a quadratic (parabolic) form.

For the calculation of the element stiffness matrices and force vectors numerical integration was applied based on the appropriate Gaussian quadrature rule. By this numerical integration method the integral of the functions having a polynomial form – in this case – of degree 4 (which is obtained by the product of two quadrilateral functions) over the isoparametric 2D triangular domain can be exactly calculated using minimum 3 integration points [8]. The general form of triangular quadrature rules is as follows:

$$\int_0^1 \int_0^{1-r} f(r, s) ds dr \approx \sum_{i=1}^n w_i * f(r_i, s_i) \quad (12)$$

In Eq. (12) $f(r, s)$ describe a function depending on the r and s variables as local coordinates, n is the number of the Gaussian quadrature points, w_i the Gaussian weight, while r_i and s_i are the local coordinates of the i^{th} quadrature point. As highlighted above, a properly constructed n -point Gaussian quadrature yields to exact integration of maximum a $(2*n-1)$ degree polynomial. As in the present study the maximum polynomial degree is four, the three-point quadrature is a suitable choice to obtain exact integration results.

2.5 The solution algorithm

Spatial discretization was performed by the GMSH free three-dimensional finite element mesh generator which is coupled with the program developed for the analysis of the core support plate failure. During one calculation the temperature of the core support plate is considered constant, which enables to reduce the computational costs taking symmetry considerations and resulting boundary conditions into account. In this way solely one quarter of the core support plate needs to be modelled.

The algorithm starts with the extraction of the mesh information (i.e. global numbering of the nodes, local numbering of the nodes within an element and the global coordinates of each node) from the output generated by GMSH, then – based on the obtained information and along with the application of a material parameter database – calculates the element vectors and matrices.

During the calculations temperature-dependency of the material properties is taken into account, however in this preliminary model calculation of the thermal strain is not yet considered. In order to avoid the continuous decrease of the net weight of the support plate, a constant density of 7864.5 kg/m^3 was assumed which is the density of the austenitic stainless steel at a temperature of 20°C [9]. The following table shows the applied values of the temperature-dependent Young's modulus:

Between 1200°C and 1600°C the Young's modulus is determined by extrapolation assuming that above 1600°C the lower support plate failure definitely occurs due to reaching its melting temperature.

By the application of the calculation method presented in the previous subchapters, the element stiffness matrices and force vectors were calculated for each element. As presented in Eq. (12) the numerical integration is usually performed in the local coordinate system which is defined – in the present case – on the above mentioned triangular domains. To be able to perform the integration in such a way, the associated coor-

dinate transformation from the global coordinate system to the local coordinates needs to be performed. The coordinate transformation necessitates the calculation of the related Jacobian matrix ($\underline{J}$), while the calculation of the element stiffness matrices – beside the Jacobian matrices – necessitates the calculation of the matrices ($\underline{B}$) containing the derivate of the shape functions as the function of local coordinates (see Eq. (13)). With the aid of matrices containing the local shape functions ($\underline{N}$) and the predefined volume- and surface load vectors ($\underline{q}$ and $\underline{p}$) the element force vectors can also be calculated (see Eq. (14)). The formulae applied in the algorithm are the followings:

$$\underline{K}_e = \sum_{i=1}^3 w_i * \underline{B}^T(r_i, s_i) * \underline{C}(T) * \underline{B}(r_i, s_i) * |det(\underline{J}(r_i, s_i))| \quad (13)$$

$$\underline{F}_e = \sum_{i=1}^3 w_i * \underline{N}^T(r_i, s_i) * \left(\int_{-\frac{t}{2}}^{\frac{t}{2}} \underline{q}(r_i, s_i, z) dz + \underline{p}(r_i, s_i) \right) * |det(\underline{J}(r_i, s_i))| \quad (14)$$

In Eq. (13) the matrix gives the relation between the external load and forces and the material behavior according to Eq. (2) and Eq. (3). Both Eq. (13) and Eq. (14) contains a summation for Gaussian point including the corresponding weights (w_i). In Eq. (15) $E(T)$ is the temperature-dependent Young's modulus (see Table 1), ν is the Poisson's ratio (considered to be 0.3 [9]), while $k = \frac{5}{6}$ is a recommended correction factor [10].

$$\underline{C}(T) = \begin{bmatrix} \frac{t^3 * E(T)}{12 * (1 - \nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1 - \nu}{2} \end{bmatrix} & \begin{bmatrix} 0 & 0 \\ 0 & 0 \\ 0 & 0 \end{bmatrix} \\ \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} & \frac{k * t * E(T)}{2 * (1 + \nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \end{bmatrix} \quad (15)$$

Table 1. Applied values of the Young's modulus at different temperatures [9]

Temperature $T [^{\circ}\text{C}]$	Young's modulus $E [\text{GPa}]$
20	200
100	193.9
200	186.2
300	178.9
400	172.0
500	165.7
600	159.7
700	153.7
800	147.3
900	139.7
1000	130.2
1100	117.9
1200	101.6

The expressions in Eq. (13) and (14) are calculated for all the M elements. With the aid of the nodal numbering information (that connects the local numbering at the element level to the global numbering at the system level) the global matrix and vector can be built up from the element matrices and vectors. This process, called assembling in the finite element terminology, might be the most complex step of the finite element solution process but not due to its mathematical complexity rather because of the difficulty of handling large matrices [6–8]. As a result of the assembling procedure the global stiffness matrix and force vector are obtained.

2.6 Failure limit calculation

In order to examine the failure state of the core support plate, the obtained displacement field can be compared to a user-defined, even time-dependent displacement limit, e.g. according to the following equation:

$$|u|_{tot} = \sqrt{u^2 + v^2 + w^2} \leq LIMIT(t) \quad (16)$$

The norm of the displacement vector can be calculated for each node according to Eq. (16) and it can be decided whether the predefined condition is met. If the norm of the displacement vector reaches the limit at any node, the mechanical damage of the core support plate at the position of that specific node can be set true.

3 Calculation results

The developed method was tested for one- and two dimensional problems. In this section the calculations performed with the developed finite element-based program are presented and compared to analytical solutions. The following subsections show the result of this preliminary verification.

3.1 One-dimensional verification

As a first step in the verification process a one-dimensional fixed beam problem (Fig. 2.) [11] was recalculated. This problem can be considered one-dimensional because the length of the system in one direction is more extensive than that in the other dimensions. If holes are not considered on the target geometry (i.e. core support plate) it can be regarded similar to a fixed beam problem. The applied geometric parameters of the fixed beam problem in the Reissner-Mindlin model are: length (L) of 10 m, width (Y) of 1 m and thickness (t) of 0.2 m. Two different cases were calculated: the effect of 100 N concentrated force (F) in the end of the middle axis (case 1) and the effect of constant 100 N/m distributed load (p) (case 2). Similarly to that in the model description in section 2, the mid-surface is examined and the mesh is generated over its middle axis. For the calculations aluminum beam is considered with the following properties [11]:

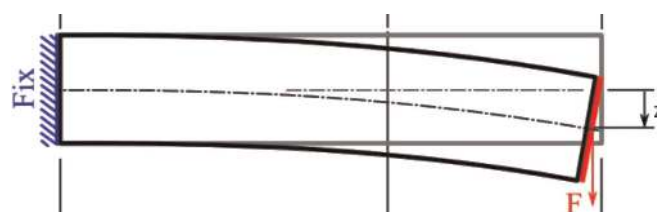


Fig. 2. Fixed beam geometry

Using the above mentioned assumptions and properties, the numerical calculations were performed and the obtained results were compared to the analytical solution of the same problem. The relative error was calculated based on the following formula:

$$\text{RELATIVE ERROR}[\%] = \frac{|\text{numerical result} - \text{analytical result}|}{|\text{analytical result}|} * 100 \quad (17)$$

The analytical solution (see Eq. (18) and (19)) is based on the solution of the differential equation of the bending curve with fixed boundaries (the values for the Young's modulus (E) and second order area (I) can be found in Table 2).

$$w_0(x)_{\text{concentrated}} = -\frac{F}{I * E} * \left(\frac{L * x^2}{2} - \frac{x^3}{6} \right) \quad (18)$$

$$w_0(x)_{\text{distributed}} = -\frac{P}{I * E} * \left(\frac{x^4}{24} - \frac{L * x^3}{6} + \frac{L^2 * x^2}{4} \right) \quad (19)$$

The numerical calculation was performed using a one-dimensional mesh with 365 nodes. Mesh independence study was performed and the obtained results confirmed that the calculation at this level of mesh resolution can be considered independent from the number of nodes. The results of the comparison are shown in Fig. 3. and Fig. 4.

Table 2. Material properties of the aluminum beam applied for one-dimensional verification [11]

Poisson's ratio ν [-]	Young's modulus E [GPa]	Second moment of area I [m ⁴]
0.33	71	$\frac{Y * t^3}{12} = 0.0000833333$

Figure 3. and Fig. 4. show that the numerical results agree well with the results of the analytical solutions in this one-dimensional verification problem based on a fixed beam calculation. The relative error of the results is between 1–5 %.

3.2 Two-dimensional verification

The analytical form of the Reissner-Mindlin model is a system of partial differential equations which consists of material laws, strain-displacement equations, equilibrium equations and the assumptions given in Section 2.3. For simple geometries in more dimensions the analytical solution of the system of partial differential equations can be also given.

In order to perform a preliminary two-dimensional verification, a two-dimensional square plate was considered. The plate is at all sides simply supported [12] and has a specific

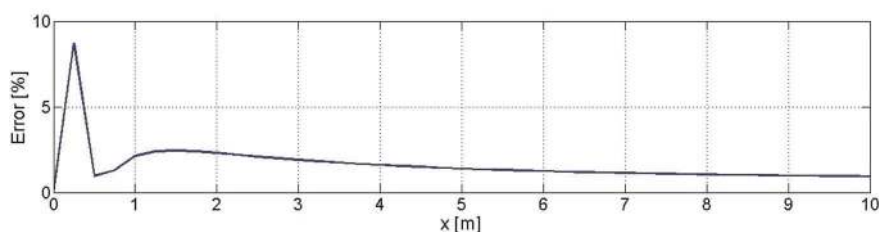
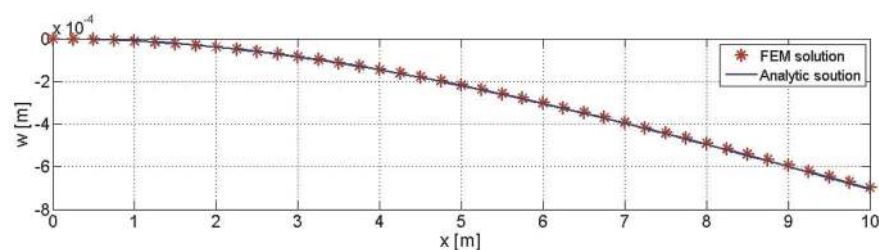


Fig. 3. Results and relative error of the comparison between numerical and analytical solutions in case 1 of the one-dimensional fixed beam problem

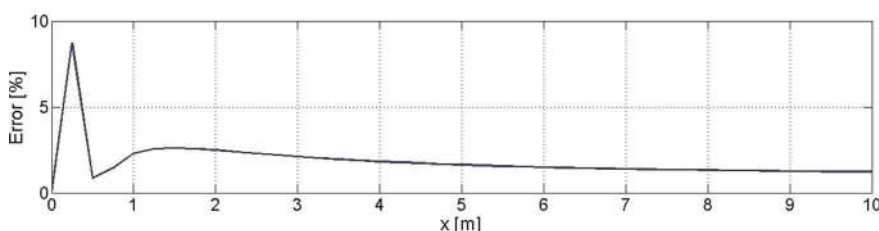
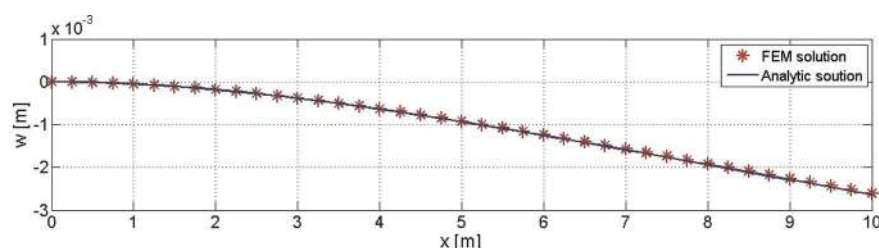


Fig. 4. Results and relative error of the comparison between numerical and analytical solutions in case 2 of the one-dimensional fixed beam problem

distributed load function. The applied geometrical, material and load properties are shown in Table 3.

According to [12], if the load function defined in Eq. (20) is applied, the solution of the z -directional displacement is the Navier-solution in the upper criterion's case shown in Eq. (21). The values needed for the calculation of the analytical solution can be found in Table 3.

$$p(x, y) = p_0 * \sin\left(\frac{\pi * x}{X}\right) * \sin\left(\frac{\pi * y}{Y}\right) \quad (20)$$

$$w_0(x, y) =$$

$$\frac{p_0 * X^4}{\frac{E * t^3}{12 * (1 - \nu^2)} * \pi^4 * \left(1 + \frac{X^2}{Y^2}\right)^2} * \left[1 + \frac{\pi^2 * t^2 * (2 - \nu) * \left(1 + \frac{X^2}{Y^2}\right)}{10 * (1 - \nu) * X^2}\right] * \sin\left(\frac{\pi * x}{X}\right) * \sin\left(\frac{\pi * y}{Y}\right) \quad (21)$$

The numerical calculation was performed using a two-dimensional triangular mesh with 545 nodes (Fig. 5.). Mesh independence study was also performed and the obtained results confirmed that the calculation at this level of mesh resolution is independent from the number of nodes. The results of the comparison are shown in Fig. 6.

Figure 6 shows the results obtained by the finite element solution as well as the analytical values for the z -directional displacement in every node. In Fig. 6 the envelope curves for the calculated displacements are also presented. It can be assessed that the numerical results agree well with the results of the analytical solutions in the two-dimensional verification problem based on the square plate calculation. However, in

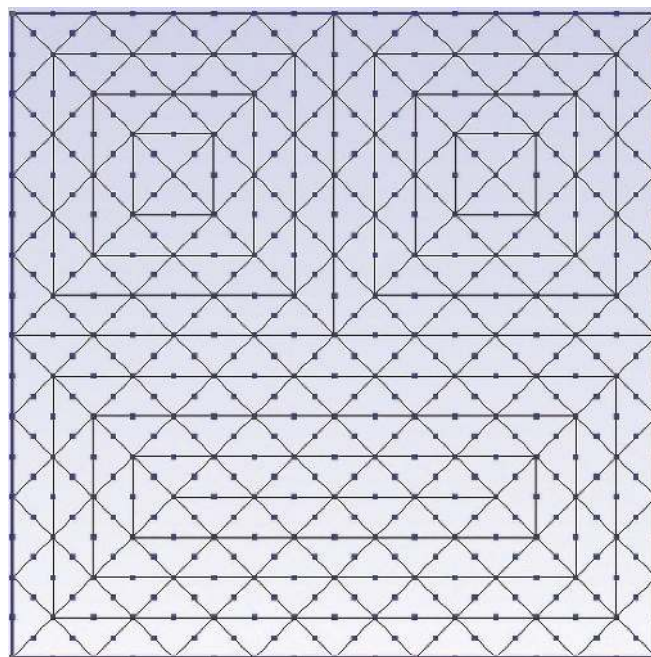


Fig. 5. Mesh of the analyzed 2D square geometry

the case of inner nodes the relative error of the results falls between 15–22 %. The distribution of the relative errors shows an increasing tendency towards the boundaries while lower relative error values can be found in the direction of the center of the plate.

Table 3. Geometrical, material and load properties applied for the two-dimensional verification in the case of the square plate [12]

Geometrical properties		
Length X [m]	Width Y [m]	Thickness t [m]
1.716	1.716	0.343
Material properties		Load properties
Young's-modulus E [GPa]	Poisson's ratio ν [-]	Amplitude of load function p_0 [N/m ²]
200	0.3	-10000

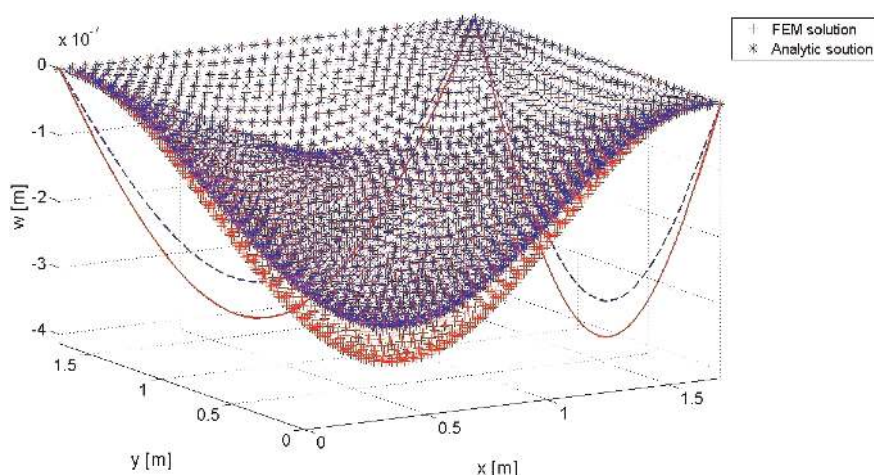


Fig. 6. Results of the comparison between numerical and analytical solutions in the case of the two-dimensional square plate problem

Based on the experience gathered during the verification of the developed method a test calculation was performed on a core support plate to simulate its deformation at 1200 °C and 190 tons of distributed load. In order to reduce computational cost symmetry condition was used and only one quarter of the core support plate was modelled. The mesh for the model is depicted in Fig. 7.

The first results are shown in Fig. 8. As expected, the biggest deformation can be seen at the inner side of the core support plate. The test calculations showed the potential for future use of the developed program. However, before further development a detailed validation is planned.

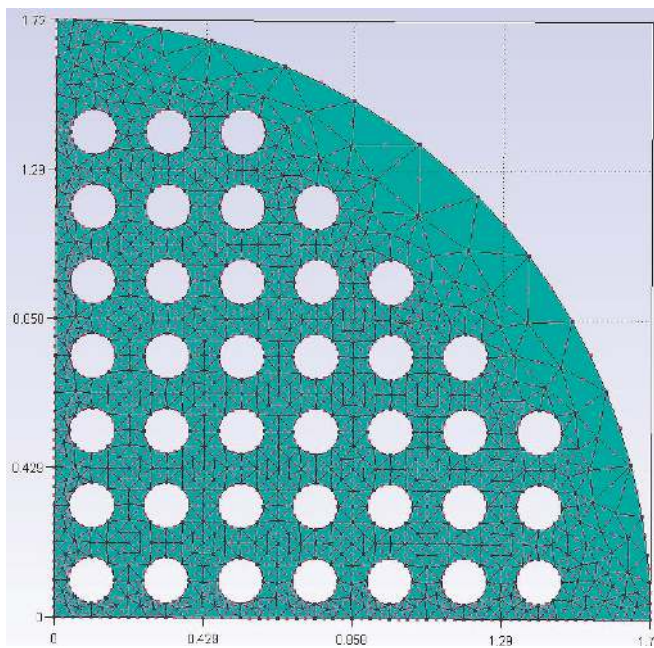


Fig. 7. Mesh of the analyzed quarter core support plate

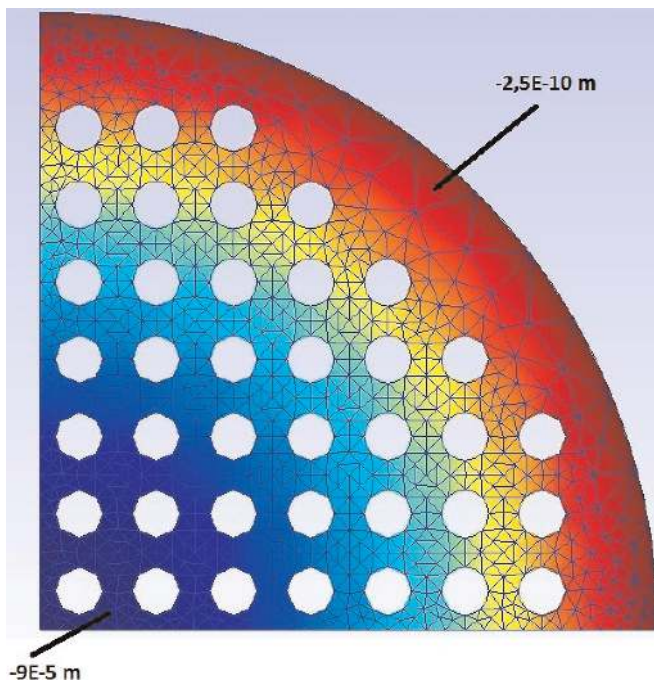


Fig. 8. Deformation of the analyzed quarter core support plate ($T = 1200^\circ\text{C}$ and load = 47.5 t)

4 Conclusions and outlook

The introduced model is still under development, but the first steps towards the calculation of a more realistic core support plate failure criterion were achieved. One and two dimensional sample calculations were performed based on the implemented theoretical failure model using finite element solution and compared with analytical results. The numerical simulations showed good agreement with the expected theoretical results even considering that solely a preliminary model was implemented. The one dimensional case shows a minimal relative error potentially caused by the finite element discretization process and it can be stated that the obtained results overestimate the analytical ones. In the two dimensional problem it is the contrary, the obtained results overestimate the analytical solution. The received relative errors for the two cases are not in same scale and the experiences show that their behavior is opposite.

The developed model provides a good basis for further improvement and for its future implementation of its results into ATHLET-CD. There are still many phenomena that have to be analyzed: the temperature-dependency of the Young's modulus, plasticity, nonlinear behavior, as well as other mechanical and creeping models. The present model itself needs to be further developed in order to handle different types of boundary conditions.

One big advantage of the implemented model is that it is possible to define different material parameters, mechanical load or even thermal load in the future for each element. The model also enables the consideration of temperature distribution in the core support plate. With the presented method node-wise failure criterion can be determined thus the evolution of the in-vessel core degradation process can be more accurately modelled.

Currently, the implemented model only calculates the displacement values in the defined nodes, however in general the failure of the support plate is determined by the equivalent stress. In future the result of the displacement calculations have to be converted into an equivalent stress field which could provide an improved failure criterion, as well as the possibility of the application of more sophisticated failure models, like that of Larson and Miller [13]. It is also planned to take the temperature distribution into account which would make the coupling between the thermal and mechanical field more realistic.

(Received on 25 September 2017)

References

- Hoffmann, M.: Beitrag zur Modellerweiterung des Systemcodes ATHLET-CD zur Analyse des Einflusses der Kernschmelzeverlagerung auf ausgewählte Unfallszenarien. Bochum, Dissertation zur Erlangung des Grades Doktor-Ingenieur (2015)
- Source: <https://forum.solidworks.com/thread/68643> (11.09.2017)
- Steele, C. R.; Blach, C. D.: Introduction to the Theory of Plates. Division of Mechanics and Computation Department of Mechanical Engineering, Stanford University (2009)
- Rusten, T.: Chapter 4. In The Finite Element Method in Solid Mechanics II. Department of Mathematics University of Oslo (2008)
- Hosseini, S. B.: Implementation and analysis of error estimates for hp MITC finite elements of Reissner-Mindlin plates. Thesis. Espoo School of Engineering, (2016)
- Moharos, I. et al.: Végeselem-módszer. Budapest, Typotex Kiadó (2011)
- Washizu, K.: Variational Methods in Elasticity & Plasticity. 3rd edition. Oxford-New York, Pergamon Press (1982)

- 8 Zhang, G.: Introduction to Integrative Engineering: A Computational Approach to Biomedical Problems. 1st edition. New York, CRC Press (2017)
- 9 Spittel, M.; Spittel, T.: Steel symbol/number: X6CrNiMoTi17-12-2/1.4571. In Ferrous alloys (2009)
- 10 Dr. Cirak, F.: Finite Element Formulation for Plates. Presentation – Handout 4. Source: http://www-g.eng.cam.ac.uk/csml/teaching/4d9/4D9_handout4.pdf
- 11 Source: http://www.mogi.bme.hu/TAMOP/vegeselem_modszerek/index.html (11.09.2017)
- 12 Valle, J. M. M.: A new refined theory of plates with transverse shear deformation for moderately thick and thick plates. Latin American Journal of Solids and Structures Vol. 12, No. 2, Rio de Janeiro (2015)
- 13 Source: <https://caeai.com/blog/creep-rupture-and-larson-miller-approach>

Master of Sciences *Boglárka Babcsányi*
 Budapest University of Technology and Economics
 Institute of Nuclear Techniques
 Műegyetem rkp. 9. R.216/6
 1111 Budapest, Hungary

The authors of this contribution

Tamás Hajas
 1 a., 9., Erkel Ferenc street
 8800, Nagykanizsa
 Hungary

Dipl.-Ing. *Péter Pandazis* (corresponding author),
 Master of Sciences, *Lóvász Lovász*
 Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH
 Forschungszentrum, Boltzmannstr. 14
 85748 Garching, Germany
 E-mail: peter.pandazis@grs.de

Bibliography

DOI 10.3139/124.110845
 KERNTECHNIK
 82 (2017) 6; page 685–692
 © Carl Hanser Verlag GmbH & Co. KG
 ISSN 0932-3902

Books • Bücher

Safety of Nuclear Fuel Cycle Research and Development Facilities. Published by the International Atomic Energy Agency, 2017, IAEA Safety Standards Series No. SSG-43, ISBN 978-92-0-103116-7, 93 p., 44.00 EUR.

The objective of this Safety Guide is to provide up to date guidance on engineering actions, conditions and procedures to meet the requirements established in NS-R-5 (Rev.1) based on experience gained in Member States. This Safety Guide is intended to be of use to researchers, designers, operating organizations and regulatory bodies for ensuring the safety of R&D facilities.

This Safety Guide includes guidance on radiation protection measures to meet the safety requirements specified in Radiation Protection and Safety of Radiation Sources: International Basic Safety Standards, IAEA Safety Standards Series No. GSR Part 3. GSR Part 3 and the associated guidance in Occupational Radiation Protection, IAEA Safety Standards Series No. GSG-7, also present measures for personnel dosimetry, optimization of protection, measures to control and limit the discharge of radioactive materials to the environment and radiation monitoring of the workplace as well as contamination monitoring of workers.

This Safety Guide provides examples of the application of a graded approach to nuclear fuel cycle R&D facilities. The graded approach in itself is a requirement in many IAEA standards, e.g. Requirement 1 of Safety Assessment for Facilities and Activities, IAEA Safety Standards Series No. GSR Part 4 (Rev. 1), and Requirement 6 of GSR Part 3. Application of a graded approach ensures that safety measures and safety related activities are proportionate to the ha-

zards of a facility.

This Safety Guide contains guidance specific to nuclear fuel cycle R&D facilities based on relevant IAEA safety requirements cited in this publication. The recommendations in this guide have been referenced to the corresponding requirements, where consistent with the readability of the text. This Safety Guide covers all stages in the lifetime of an R&D facility, including site evaluation, design, construction, commissioning, operation and decommissioning. It also provides specific guidance on modifications, maintenance, calibration, testing and inspection as well as emergency preparedness, where such guidance is appropriate.

General safety guidance for an R&D facility is provided in Section 2. The safety aspects to be considered during the process of evaluating the site for a facility are described in Section 3. Section 4 deals with safety in the design stage and Section 5 deals with safety aspects in the construction stage. Section 6 describes the safety considerations that arise during commissioning. Section 7 contains guidance on practices to ensure safety during facility operation. It also covers the management of facility operations and emergency preparedness and response. Section 8 provides guidance on meeting safety requirements in the decommissioning of an R&D facility. Annexes I and II show the typical process route for the two classes of R&D facilities covered by this guidance. Annex III gives examples of structures, systems and components (SSCs) important to safety in R&D facilities, grouped by process areas. Examples of operational limits and conditions for R&D facilities are provided in Annex IV.