Study of the mechanical stability of a PWR vessel during a severe accident by a finite element method of homogenization

Diaz V.(1), Gatt J.M.(1), Peseux B.(2)
(1) CEA, France
(2) Ecole Centrale de Nantes, France

ABSTRACT: Within the framework of PWR severe accident analysis, the thermomechanical behaviour of the vessel is studied with a new finite element based on the homogenization method and the shell theory. This allows to take directly into account the vessel melting and the variations of the physical parameters. Since the results are very satisfactory (good comparison with another finite element code and reduction of calculation time) and the model simple enough, we can introduce it into a scenario code.

1 INTRODUCTION

During the course of severe accidents, one of the main uncertainties is the thermomechanical behaviour of the vessel which is submitted to high thermal gradients due to the presence of corium (mixture of molten fuel and materials). These can lead to its melt-through and rupture. The situations in which the vessel could resist have to be determined as well as the ones which could cause its rupture. In this last case, the way in which the vessel fails is very important for the continuation of the scenario.

Up to now, no precise modelling of the vessel behaviour has been introduced into scenario codes such as, for example, MAAP4 [1] and ICARE2 [2]. Our aim is to present an analysis of the mechanical behaviour of PWR vessels by a finite element method of homogenization. It will be an intermediate method between the quick but too simplified analytical methods and the accurate but much too heavy models, so that we can introduce our model into a scenario code.

2 GEOMETRY

The reactor vessel model is made up of a cylindrical structure with an hemispherical bottom head. Both of them have the same mean radius \( R_m \) and the same thickness \( 2h \) (Fig 1).
3 THERMOMECHANICAL THEORY

The equilibrium equation is given by:

$$\text{div}\sigma + \vec{f} = \vec{0}$$  \hspace{1cm} (1)

with $\sigma$: Cauchy Stress tensor,
$\vec{f}$: vector of volumic internal loads.

First, we neglect the damage and consider an elastic, plastic, creep and thermal behaviour. Therefore, the strain tensor is:

$$\varepsilon = \varepsilon^e + \varepsilon^{pl} + \varepsilon^{cr} + \varepsilon^{th}$$  \hspace{1cm} (2)

where $\varepsilon$, $\varepsilon^e$, $\varepsilon^{pl}$, $\varepsilon^{cr}$, $\varepsilon^{th}$ are respectively the total, elastic, plastic, creep and thermal strain tensors.

We note $\vec{u}$ the displacement vector. The total strain is given by:

$$\varepsilon = \frac{\text{grad} \vec{u} + (\text{grad} \vec{u})^T}{2} + \frac{\text{grad} \vec{u} \cdot (\text{grad} \vec{u})^T}{2}$$  \hspace{1cm} (3)

The elastic and thermal strains are written as:

$$\sigma = D\varepsilon^e \hspace{2cm} \varepsilon^{th} = \alpha \Delta T$$  \hspace{1cm} (4)

where $D$: elasticity tensor,
$\alpha$: thermal expansion coefficient tensor.
$\Delta T$: temperature.

If the steady-state operating conditions are reached, the vessel failure can only happen by creeping. We assume here a perfect creep law given by [3]:

$$\dot{\varepsilon}^{cr} = \frac{3}{2} \left( \frac{\sigma_{eq} - \sigma_Y}{\lambda_Y} \right)^{N_Y} \frac{\sigma'}{\sigma_{eq}}$$  \hspace{1cm} (5)

The effective stress $\sigma_{eq}$ follows the Von Mises yield criterion:
\( \sigma_{eq} = \sqrt[3]{\frac{3}{2}} \sigma' : \sigma' \)  

(6)

with \( \sigma' \): deviator of \( \sigma \).
\( \sigma_Y \): yield strength.
\( \lambda_Y, N_Y \): temperature-dependent constants identified by the behaviour law.

During the transient operating conditions, the vessel can also break due to plastic instability. Then, we suppose that plastic strains follow the law of Prandtl-Reuss:

\[
\text{d}e_{pl} = \frac{3}{2} \frac{M_Y}{K_Y} \left( \frac{\sigma_{eq} - \sigma_Y}{K_Y} \right)^{M_Y - 1} \left( \frac{d\sigma_{eq}}{\sigma_{eq}} \right)
\]

(7)

\( M_Y \) and \( K_Y \) depend on the temperature and are given by the behaviour law.

4 FINITE ELEMENT ANALYSIS

We suppose that thermal and mechanical phenomena are uncoupled. The vessel thermal analysis is already performed satisfactorily by specific programs included in the scenario codes. In consequence, we use these results only to define the evolution of the parameters which depend on the temperature and to calculate the thermal stresses.

4.1 Calculation characteristics

Due to the dimensions of the vessel, we can consider it as a thin shell and we apply the corresponding linear theory. Our aim is to find a compromise between speed and accuracy. Then, the geometry is simplified as much as possible:
- the problem is considered as axisymmetric.
- each element has only 2 nodes with 3 degrees of freedom: 2 displacements \( w, u \) and 1 rotation \( \beta \) around the axis perpendicular to the element plan, as usual in the axisymmetric thin shell theory.

In order to gain accuracy, we develop analytical formulae and apply the homogenization method: we then take into account, on one hand, the variations of the physical parameters (Young's modulus...) with the temperature and, on the other hand, the vessel melting, directly into the elements. With such a method, we can consider only one element through the thickness.

4.2 Temperature and Young's modulus expressions

The thermal program gives a mesh of temperatures \( T(N, M, i) \): \( M \) corresponds to the element number and \( i \) to the position \( p(i) \) of the point through the thickness as it can be seen in Figure 2. \( N \) is the total number of temperature values through the thickness.
For each temperature value $TN(M,i)$, the Young's modulus $E(M,i)$ is calculated by a linear interpolation of the material characteristics. Then we define the Young's modulus for each element by a bilinear interpolation:

$$\forall \ p \in [p(i),p(i+1)] \quad E(M,i) = a_i s + b_i + p (c_i s + d_i) \quad (8)$$

where $s$ is the curvilinear coordinate along the neutral axis (axis of the flexion centers).

We will see in the next paragraph that the displacements are known if we know those of the neutral axis. As the Young's modulus varies through the thickness, the neutral axis varies too and its position $Y_0$ is given by:

$$Y_0 = \frac{\int_b^h E_p \, dp}{\int_b^h E \, dp} \quad (9)$$

After development we can write $Y_0$ as:

$$Y_0 = YA + \frac{YB}{\lambda}s + \mu \quad (10)$$

$YA, YB, \lambda$ and $\mu$ are constants defined for each element and depend on $a_i, b_i, c_i, d_i$.

### 4.3 Displacements

To define the displacements we use 3 references (Fig 3):

- the 'tangential reference' characterized by a 't' in subscript corresponds to the tangential and normal displacements $u_t$ and $w_t$.
- the 'local reference', with a 'l' in subscript, is defined by the axis $x$ which links together the 2 nodes of the element. The stiffness matrix of each element is written in this reference.
- the 'global reference' is used to write the global stiffness matrix of the structure. It is defined by the axis $z$ and the perpendicular one. The displacements are noted $u$ and $w$.

We suppose that the displacements are small on a time step. In consequence, the normal and tangential displacements $w_t$ and $u_t$ and the rotation $\beta$ are given by equations (11). A '1' above a variable indicates that the value is taken at the node. $q$ is the position of the point through the thickness relative to the neutral axis coordinate (Fig 4). $R_{NS}$ is the radius of curvature in $s$.

$$\begin{align*}
    u_t &= \bar{u}_t - q\bar{\beta} \\
    w_t &= \bar{w}_t \\
    \beta &= \frac{d\bar{w}_t}{ds} + \frac{\bar{u}_t}{R_{NS}}
\end{align*} \quad (11)$$

422
The angle between the axis of symmetry $z$ and the normal direction to the neutral axis is $\gamma$. The angle between the tangent to the neutral axis and the axis $x$ is $\Phi$ (Fig 3).

![Fig 3: Displacements](image1)

![Fig 4: Coordinates](image2)

To a first order we can write:

$$\frac{ds}{dx} = \frac{dx}{\cos \Phi}, \quad d\Phi = dy = -\frac{ds}{R_m + Y_0} \tag{12}$$

We assume that $\Phi$ is small. We can then interpolate the displacements by the equations [4]:

$$\bar{u}_1 = a_1 + a_2 x$$

$$\bar{w}_1 = a_3 + a_4 x + a_5 x^2 + a_6 x^3 \tag{13}$$

4.4 Strains

Since we assume small strains, $w_t$ does not depend on the radial coordinate $r$ and so the radial strain $\varepsilon_r$ is nil. In consequence we obtain the general expressions [5]:

$$\varepsilon_\theta = \frac{u_t - w_t}{R_g \theta - R_{N\theta}} \frac{1 - q}{R_{N\theta}}$$

$$\varepsilon_s = \frac{\partial u_t}{\partial s} - \frac{w_t}{R_{NS}} \frac{1 - q}{R_{NS}} \tag{14}$$

with:

<table>
<thead>
<tr>
<th>Sphere</th>
<th>Cylinder</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_\varepsilon$</td>
<td>$(R_m + Y_0) \tan \gamma$</td>
</tr>
<tr>
<td>$R_N$</td>
<td>$-(R_m + Y_0)$</td>
</tr>
<tr>
<td>$R_{NS}$</td>
<td>$-(R_m + Y_0)$</td>
</tr>
</tbody>
</table>

$R_g$ is named the geodesic radius and $R_N$ the principal radius. The subscript 's' refers to the neutral axis and $\theta$ to the curvilinear line perpendicular to the plan of the element.

Due to the well determined geometry of our model and since we want to be as simple as possible, we develop specific formulae for the cylinder and the sphere. But for both of them, we obtain in the end the well known relation:

$$\{\varepsilon\} = B U_N \tag{15}$$

with

$$\{\varepsilon^T\} = \{\varepsilon_\theta, \varepsilon_s\} \quad U^T_N = \{\bar{u}_{l1}, \bar{w}_{l1}, \bar{\beta}_1, \bar{u}_{l2}, \bar{w}_{l2}, \bar{\beta}_2\}$$

423
4.5 Stresses

From the principle of virtual work, we obtain the standard equilibrium equation [6]:

\[ \int_V B^T \sigma \, dV - F = 0 \]  \hspace{1cm} (16)

And we have:

\[ \sigma = D (\epsilon - \epsilon^{cr} - \epsilon^{pl} - \epsilon^{th}) + \sigma_0 \]  \hspace{1cm} (17)

where \( \sigma_0 \) is the initial stress.

Substituting (17) in (16) gives the following relation:

\[ KU_N - \int_V B^T D (\epsilon^{cr} + \epsilon^{pl}) \, dV - \bar{F} = 0 \]  \hspace{1cm} (18)

with

\[ K = \int_V B^T DB \, dV \]

\[ \bar{F} = F + \int_V B^T D \epsilon^{th} \, dV - \int_V B^T \sigma_0 \, dV \]

K is the standard stiffness matrix and \( \bar{F} \) the prescribed force vector.

We obtain then a discretized system of differential equations, transient with respect to time, whose solution can be obtained in a time-stepping manner.

5 APPLICATION

5.1 General problem

To begin our validation we only consider an elastic behaviour and neglect the weight of the corium. Since we have developed specific formulae for the cylinder and the sphere, we present two examples: a cylinder embedded on its two ends and the whole vessel embedded at its top. Both of them are submitted to an internal pressure \( P = 5.10^7 \) Pa and a bilinear temperature as depicted by figures 5 and 6. The geometrical values are approximately those of a PWR vessel, that is: \( R_m = 2m, h=0.1 \text{ m}, LT=4 \text{ m} \). The material characteristics are those of the 16MND5 steel which is used for PWR-type vessels.

![fig 5: cylinder boundary conditions](image)

![fig 6: Vessel boundary conditions](image)

We use the CEA finite element code Castem 2000 to validate our model. Since Young's modulus vary through the thickness we must use eight-node quadrilateral elements. They have also the advantage of introducing no hypothesis on the strains and stresses. So the Castem 2000 solution is closer from the exact one which enables us to check our hypotheses. We need for Castem 2000 8 elements in thickness to obtain an accurate solution. The same number of elements in length is used for Castem 2000 and our model.
5.2 Results

For the cylinder, the stresses $\sigma_\theta$ and $\sigma_z$ versus the curvilinear coordinate $s$ are depicted by Figures 7 and 8. They are relative to the internal wall of the structure. For the maximum stresses the error is of 0.2% for $\sigma_\theta$ and of 10% for $\sigma_z$. Figure 9, which represents the stress $\sigma_r$ versus the curvilinear coordinate $s$, enables us to check that it is negligible.

In the case of the vessel, Figures 10 and 11 give the displacements $w$ and $u$ of the mean axis versus the curvilinear coordinate $s$. We can see a very good agreement between Castem 2000 and our model. For the maximum displacements the error is of 1% for $w$ and of 2% for $u$.

Notation: simple lines correspond to the results of our model and lines with crosses to those of Castem 2000.

![Fig 7](image1.png)

![Fig 8](image2.png)

![Fig 9](image3.png)

![Fig 10](image4.png)

![Fig 11](image5.png)
6 CONCLUSION

A new finite element has been developed to study the mechanical behaviour of a PWR vessel during a severe accident. The objective is to implement the model into a scenario code. So it must be simple and accurate.

Simplicity is obtained by using standard shell elements with two nodes and three degrees of freedom per node. To gain accuracy we develop analytically as much as possible the equations of equilibrium applied to the finite element method. This enables us to take into account the variations of the parameters in function of the temperature and the vessel melting. The results in elasticity are very good. Plasticity and creep have now to be considered.

REFERENCES


2. ICARE 2. User’s manual. CEA/IPSN Cadarache


