Thermomechanical Analysis of a Steam Generator Bottom

G. Debruyne, S. Ignaccolo
Electricité de France, Clamart, France

1. INTRODUCTION.

During an in-service inspection in french Tricastin 900 MW PWR unit 4, a defect has been detected by ultrasonic testing in a steam generator (SG) bottom. It was located in the area of the weld joining the channel head, the tube plate and the partition plate. This latter cannot be modelized by usual axisymmetric models of the SG. To estimate the nocivity of this defect, it is necessary to perform a 3D thermomechanical analysis by F.E.M. of the whole SG bottom. This study is also very helpful to know the mechanical behaviour of such an important component under transient thermal and pressure loadings. The computed stress distribution in the SG can then make up a data base for any kind of investigation or crack propagation analysis.

2. BASIC ASSUMPTIONS.

In the SG modelization, we consider the tube plate, a piece of the secondary shell, and the primary head with nozzle, manway and partition plate. Preliminary 2D studies have been carried out to check the modelization parameters. They shown that a linear thermoelastic analysis was sufficient. The multiperforated part of the tube plate is not directly modelized but, using homogenization techniques, replaced by an homogeneous equivalent thermal and elastic medium. Because of the thinness of the partition plate (34 mm), it was practically impossible to have an unique 3D mesh for the whole structure. Therefore, we built two models:

- A global model representing one half part of the SG, devoted to symmetrical loadings. In this case the partition plate is in plane stress and modelized with membrane elements, others components are dicretized with 3D elements.

- A local model representing the partition plate and its vicinity, devoted to non symmetrical loadings (from hot to cold leg) which have local effects. The geometry is discretized with 3D elements including the partition plate which bends.

3. MODELIZATION AND CALCULATIONS.

3.1 Modelization of the symmetrical loading cases (model A).

3.1.1 Modelization of the geometry.
The mesh of this model is composed of 5820 bricks and wedges (cf fig.2 the deformed mesh under primary pressure loading) and 250 membrane elements for the

partition plate. Some simple axisymmetric studies have shown that the sharpness of this mesh is sufficient for linear elements in elastic analysis and leads to 19500 degrees of freedom. On the other hand, transient thermal analysis with the same mesh needs quadratic discretization and leads to 27000 d.o.f.

3.1.2 Loadings and boundary conditions.
The structure is subjected to transient primary and secondary pressures and temperatures (cf fig.4 one of the transients considered). We assume that the thin partition plate gets the same temperature as the surrounding water at each step of the transient. Others boundaries have convection conditions.

3.1.3 Special treatment of the tube plate.
The multiperforated tube plate is a good example of periodic medium which can be treated by homogenization process. We briefly resume here the background of this technique and the application to our problem. The basic concept [1] is the distinction between the local behaviour in a periodic elementary cell , and the global behaviour of the whole structure. In the cell, we can find a relation between the local fields, depending on both local and global variable , and the global fields depending only on the global variable. Roughtly speaking, the local variable is the position inside the cell, and the global one the position of the cell in the structure. There are three steps in the process:

1. First, we look for an homogeneous material whose behaviour is equivalent to the heterogeneous one. In our case, this material is linear thermoelastic but anisotropic. For that, we solve elementary problems in the cell which give the local fields in terms of the global one. We can then deduce the thermal and elastic global characteristics.

2. Then, we substitute the homogenized material to the real one in the global structure and compute thermomechanical global fields of the whole structure submitted to the real loading.

3. Finally, if necessary, we can compute the local fields in every cell, for the real loading.

In our application, the cell is a square surrounding each tube hole, and forms a pavement of the perforated plate (cf fig.1).

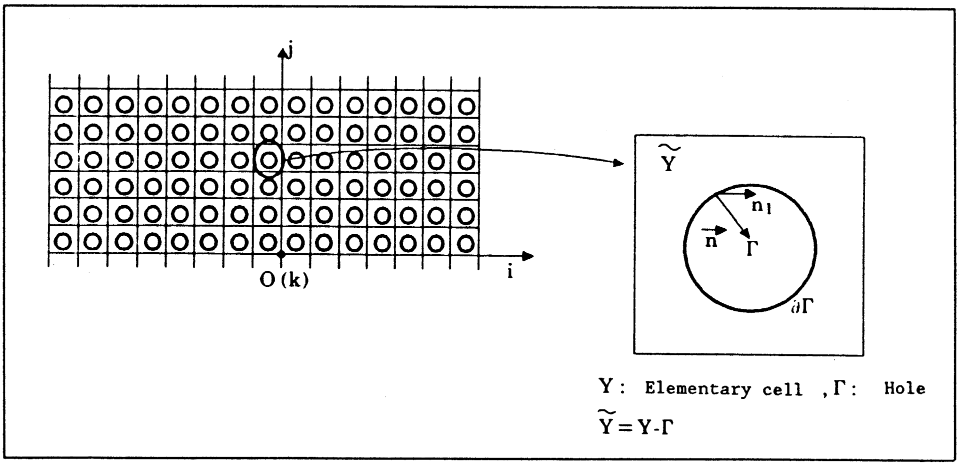


Fig.1 : Elementary cell and pavement of the tube plate

The temperature T is solution of the homogenized problem [2] :

$$\begin{cases} C \dfrac{\partial T}{\partial t} + \operatorname{div} \vec{q} + A(T - T_f) = 0 , \\ \vec{q} = - \lambda \, \overrightarrow{\operatorname{grad}} \, T , \\ \vec{q}.\vec{n} = H \, (T - T_f), \end{cases}$$

where q and C are the average heat flow and capacity on a cell. λ is the transversly isotropic conductivity tensor, ($\lambda_{11} = \lambda_{22} = 18.5$ W /m°C, $\lambda_{33} = 25.2$ W /m°C). The source term A is due to the homogenization of heat transfert on tube surface Γ, T_f is the temperature of the primary fluid, n the normal to Γ. H is the heat transfert on primary and secondary faces of the plate.

For the elastic problem [3], we only replace the usual isotropic stress-strain tensor by an orthotropic one.

3.2 Modelization of the non symmetrical loading cases (model B).

3.2.1 Modelization of the geometry.
We consider here the partition plate and its embedding (a part of the head and its connection with the tube plate, not perforated at this place). The boundaries of the embedding have been chosen, thanks to the 2D models, so that stresses due to non symmetrical loadings are negligible beyond them. Because of the symmetry we only represent one half of the geometry. The mesh of this model is composed of 2100 bricks and wedges (cf fig.8 the deformed mesh under pressure loading) and 9800 nodes. Both the thermal and the elastic problems are discretizated with quadratic elements.

3.2.2 Loadings and boundary conditions.
The loadings considered here are a pression and a temperature gradient from hot to cold leg (cf the correspondent diagrams on fig.4). Because the plate is very thin, we do not process a real transient analysis but compute the stationary thermal fields and adjust them proportionally to the real transient temperature gradient. This way is correct if the transient loading is not too rought, in this case the temperature distribution is linear in the plate section at each step of the transient. The boundary conditions on cutting faces are prescribed temperatures or displacements, starting from the results on simplified models. Others boundaries have convection conditions.

4. RESULTS.

4.1 Symetrical loadings.

4.1.1 Hydrostatic test.
To validate our computations, we compare the results of the primary pressure loading analysis with the hydrostatic test carried out on Bugey plant. Figure 2 shows for a pressure of 15 MPa the deformed mesh. We see that the most deformed component is the tube plate. The experimental curve pressure-deflection of this plate during the loading-unloading test (cf fig.3) shows that there is plastic deformation at the beginning of the test and after two cycles there is an elastic shakedownn that validates an elastic analysis. The experimental deflection of the plate is 2 mm and the numerical one 1.8 mm. The numerical value obtained without partition plate modelization is 3.3 mm which shows that this one has an important effect on the structure behaviour.

4.1.2 Transient thermomechanical loading.
Among the five transients analyzed, we present here the case of a transient with a load increase (see the diagram on fig.4). On fig. 5 we see the Mises stress field on the whole structure. The maximal stress occurs on the weld in

the bottom of the channel head and the field is not axisymmetric. The stress decreases toward the symmetry plane. This is due to the dropping effect of the partition plate . The same analysis without the partition plate shows that the stresses field is axisymmetric (except near the man hole and the nozzle): as we can see on a zoom of the cutting symmetry plane (fig. 6 and 7), the magnitude of the Mises stress on the weld is about 50% higher near the symmetry plane than for the real case, with a strong concentration. The maximal value is 216 MPa for the analysis without partition plate analysis and only 153 MPa for the complete analysis.

4.2 Non symmetrical loadings.

For this kind of loadings, we perform an elastic analysis with a differential pression on the partition plate, and a stationnary thermoelastic analysis with a temperature gradient of 40 °C prescribed on the hot face of the plate.

4.2.1 Pression analysis.

Figures 8 and 9 show the deformed mesh and the Mises stress field for a pression of 0.21 MPa. The deflection of the partition plate is 4 mm, the rest of the model is practically rigid. The stress field is concentrated in the middle of the plate, in the area joining the tube plate and below of the embedding level. The maximal Mises stress is about 150 MPa. In the vicinity of the weld joining the channel head and the tube plate the stress is very low (about 20 MPa).

4.2.2 Stationnary thermomechanical analysis.

For the temperature gradient loading, the deflection of the partition plate is only 0.6 mm. In opposition to the pression case, there is a deflection of the embedding (but less than 0.5 mm). The maximal Mises stress (120 MPa) is reached at the jonction of the channel head and the tube plate. The maximal stress in the weld is about 50 MPa.

4.3 Application to a defect analysis.

We applied the results of this study to estimate the nocivity of a defect detected in Tricastin plant. This defect is located in the weld near the partition plate, and it is necessary to combine the results of models A and B to estimate the real stresses in the area around of the crack. We can then apply the fracture concepts of RCCM [4] to evaluate the propagation risk and the stability of this defect. That leads to the non nocivity of this latter: its initial length of 3 mm will increase of 0.4 mm at end-of-life and the stability is assured with a safety factor of 5.6.

5. CONCLUSION.

We can draw three conclusions from the results of this study:

- The main one is the dropping effect of the partition plate on stress level, specially in the welded area joining the tube plate and the channel head where defects may appear. This point cannot be exhibited with 2D analysis.

- The second point is that for any transient analyzed (except for the accidental primary piping break) the Mises stress is below plastic yield anywhere in the SG, which validates an elastic analysis (the hydrostatic test shows a plastic deformation at the beginning of the test but an elastic shakedown appears after two deformation cycles).

- The large stock of those results can form a database for a stress analysis of any part of the SG. If the area analysis is near the partition plate, it is necessary to combine the results of models A and B, otherwise the non symmetrical loadings have no effect and the model A is sufficient.

REFERENCES.

[1] Sanchez-Palencia E., : Comportement local et macroscopique d'un type de milieux hétérogènes, Int. J. of Eng. Sc., Vol 12, 1974.

[2] Andrieux S., : Comportement thermique homogénéisé des plaques à tubes des générateurs de vapeur, EDF/DER HI/5785-07, 1987.

[3] Wadier Y., : Plaques à tubes : calcul des constantes élastiques équivalentes par une méthode d'homogénéisation, EDF/DER HI/4818-07, 1984.

[4] RCCM - 1983 edition.

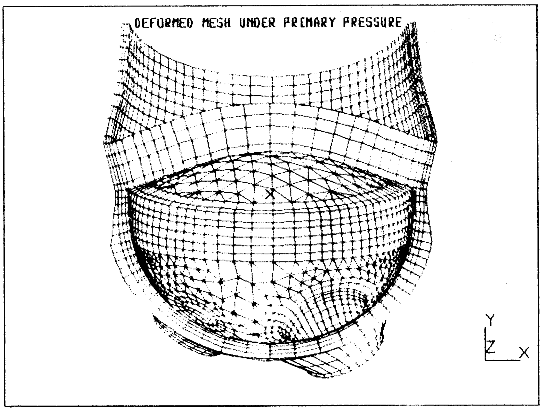


Fig.2 : Deformed mesh under 15 MPa

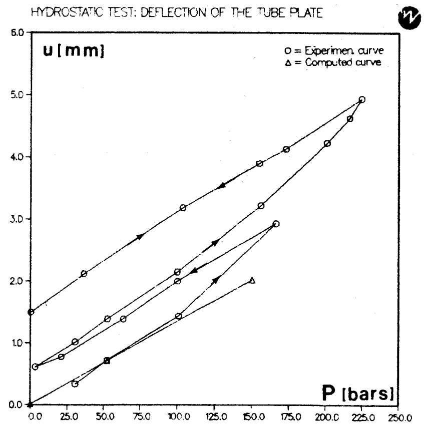


Fig.3 : Deflection of the tube plate

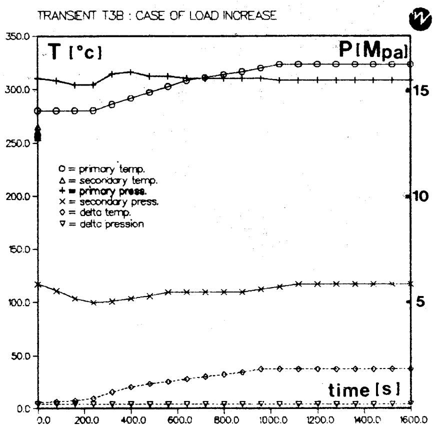


Fig.4 : Transient T3B

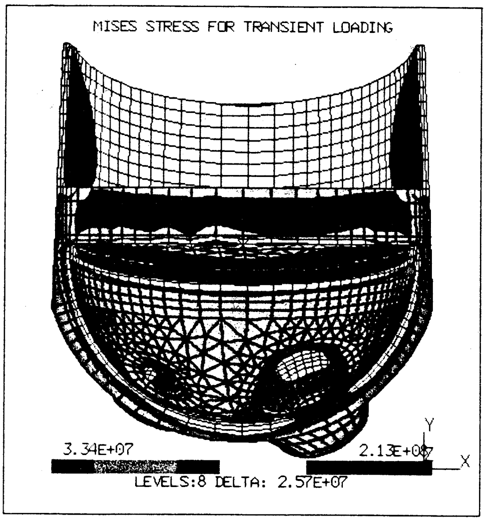
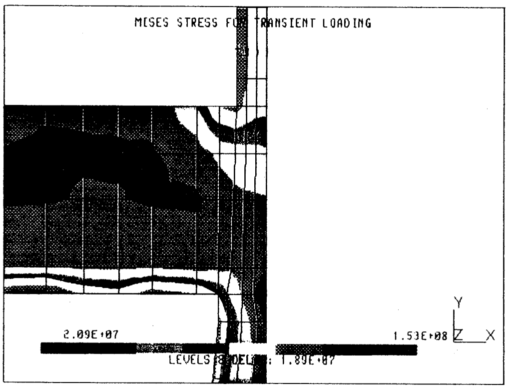


Fig.5 : Mises stress field for T3B



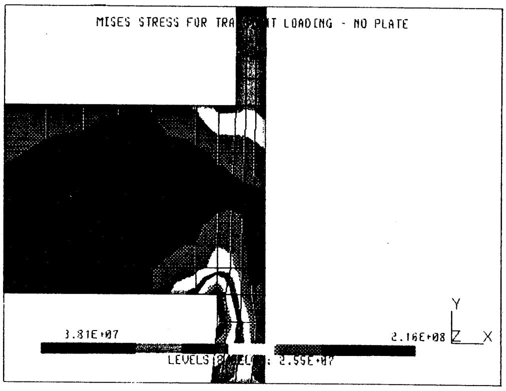
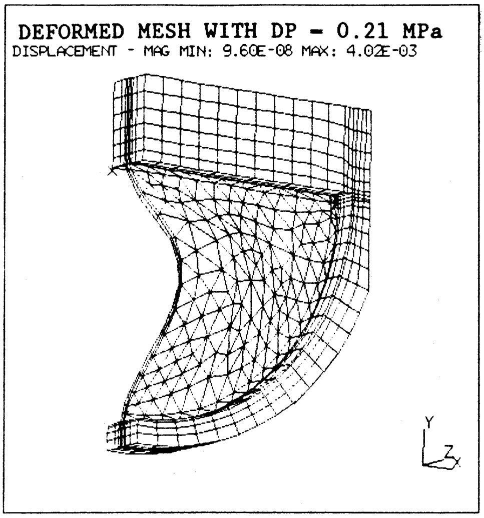


Fig.6 : Mises stress for T3B.
Zoom in the symmetry plane

Fig.7 : Mises stress for T3B.
Model without partition plate



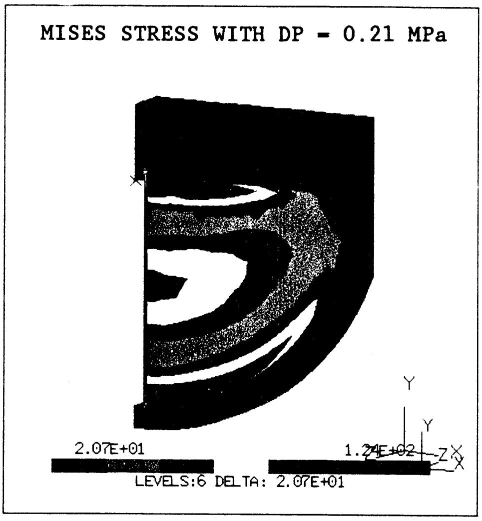


Fig.8 : Deformed mesh.
DP = 0.21 MPa

Fig.9 : Mises stress field .
DP = 0.21 MPa