ANAIS

PROCEEDINGS

7º SIMPÓSIO BRASILEIRO
SOBRE TUBULAÇÕES E VASOS DE PRESSÃO

7TH BRAZILIAN SYMPOSIUM ON PIPING AND PRESSURE VESSELS

FLORIANÓPOLIS, 07 - 09 DE OUTUBRO DE 1992

TRABALHO Nº

PP. 369-377

3-D MODELS TO ANALYSE A PWR PRESSURE VESSEL HEAD UNDER NON-AXISYMMETRIC LOADINGS

CARLOS ALBERTO DE OLIVEIRA

COPESP/IPEN - CNEN/SP

SUMÁRIO

Neste trabalho estuda-se o tampo superior de um vaso de pressão nuclear tipo PWR (pressurized water reactor). Para isto, foram feitos dois modelos tridimensionais do tampo em elementos finitos, usando o programa ANSYS (rev. 4.4a). Estes modelos utilizam principalmente elementos de casca para modelar o trecho central perfurado, e elementos isoparamétricos solidos para o flange. Os parafusos do tampo foram discretizados em elementos de viga 3-D. Os critérios do código ASME são usados para a discussão dos resultados.

SUMMARY

The upper pressure vessel head of a Pressurized Water Reactor was analysed by using two 3-D finite element models to represent it. The ANSYS program (rev. 4.4a) was used for calculations. The models use mainly quadrilateral shell elements in the perforated center of the head and 3-D 8-nodes isoparametric solid elements for the flange. The flange bolts were discretized with 3-D beam elements. Results were discussed considering the ASME code criteria and limits.

1. INTRODUCTION

The pressure vessel in study is a typical PWR one (design pressure of 16.55 MPa and design temperature of 343°C). The spherical part of the closure head is penetrated by parallel tubes related to the control rod drive mechanisms, and by other tubes, also parallel to the formers, that conduct the in-core instrumentation probes. There are 4 coolant nozzles (two inlet and two outlet ones) that support the pressure vessel and the structures attached to it. The whole design was made in accordance with the rules of the "ASME Boiler and Pressure Vessel Code, section III" [1]. This code requires a detailed stress analysis for various loading conditions that it specifies. For the analyses developed in this paper, the following loads were considered: internal pressure, bolt-tightening, and accelerations due to gravity and/or earthquake, acting in the x and z directions of the coordinate system (see Figure 1.b).

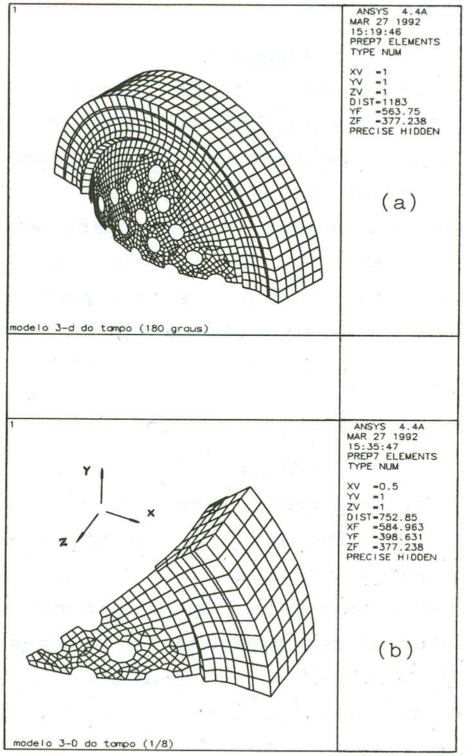

Figure 1: 3-D pressure vessel head models - (a) 180º model
(b) 45º model

Considering the loads mentioned above, two three-dimensional finite element models were developed [3] to analyse the vessel head behavior, that is, to verify the perforated part of the head and a bit about the vessel flanges.

2. THREE DIMENSIONAL MODELS OF THE HEAD

In order to know the behavior of the head, two three-dimensional finite element models were developed: for vertical accelerations a 45° model was developed, and for horizontal ones a 180° model has been used. Internal pressure and weights act in the two models - see Figure 1.

In these models the flange ring was meshed with 3-D isoparametric solid elements with 8 nodes, and the spherical shell with elastic quadrilateral shell elements. The use of shell elements makes the model simpler than if it was made only with solid elements, besides it is also conservative. This is due to the fact that when shell elements are used, the internal pressure is applied at the medium diameter of the shell, while if solid elements were employed, the internal pressure would have been applied at the actual internal diameter of the head. Also used were gap elements to model the contact between the upper and lower flanges, beam elements (with *per radian* properties) to model the flange bolts and mass elements to represent the structures attached to the head. The weakening effect of the bolt holes in the flange ring is taken into account by reducing the Young Modulus of the ring material [2]. For the shell part of the head, all of the existing holes, related to the control rod drive mecanisms and instrumentation tubes, were included in the model.

All of the analyses performed with the 3-D models are linear, except for the local non-linearities due to the gap elements.

2.1 Boundary Conditions and Coupling of Nodes

The following boundary conditions were used on these models:

- symmetry for faces at $\theta=0°$ and $\theta=45°$ (45° model), and at $\theta=0°$ and $\theta=180°$ (180° model), in the Figure 2 coordinate system.
- full constraint for bolt nodes that also belong to the lower flange ring.
- full constraint for interface element nodes that also belong to the lower flange ring.

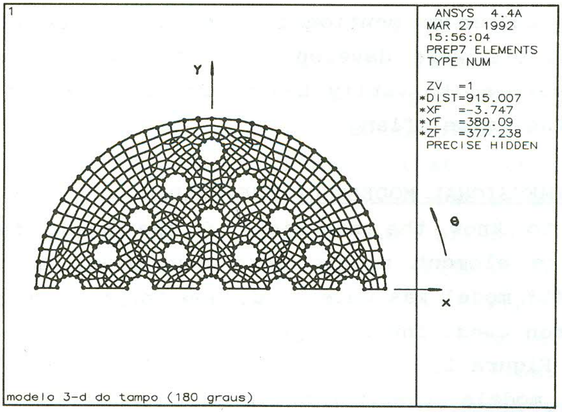

Figure 2: Shell part of the 180º pressure vessel head model

The different finite element types used in these head models have nodes with different numbers of degrees of freedom. In order to keep the consistency of the model, the nodes of different adjacent elements were coupled. By the use of rigid regions, the shell elements were coupled to the solid ones, at the model transition region (excluding the model symmetry face elements, where the boundary conditions are imposed), and the solid elements to the beam ones, at the bolts to upper flange ring union. At the model face planes, where the symmetry boundary conditions were imposed, 3-D massless rigid beams were employed to couple the solid element nodes to the adjacent shell element ones.

2.2 <u>Loadings</u>

The loadings used in these three-dimensional analyses were: bolt-tightening (1.44×10^6 N), internal pressure (16.55 MPa), and accelerations due to gravity and earthquake (49 m/s^2 in x-direction - 180° model, and z- 40m/s^2- 45° model - see Figure 1.b coordinate system).

An iterative process was used to find a suitable initial deformation that, once applied to the beam elements, leads to the actual bolt-tightening.

When applying the x- and/or z-accelerations to the head model, their effect on the internal water is taken as an increase on the internal pressure that varies with the direction considered. In the case of using a longitudinal acceleration (z-axis in Figure 1.b), the loadings related to the masses of the pressure vessel internals are applied to the upper flange ring as forces. In the case of using an x-acceleration, the structures attached to the vessel head cause moments on it. Due to the use of concentrated masses, the models are not capable to capture this effect. Hence, for each structure (control rod drive mechanisms, instrumentation tubes and supporting structure) a set of couples corresponding to a specific moment has been applied, thus reproducing the global original effect. This is done in accordance with the Figure 3, where a moment is represented by some couples that employ a cosinusoidal force distribution. Figure 2 shows the shell element model part. The nodes of this part of the model that are used to apply couples and/or concentrated masses to the model are highlighted in Figure 2.

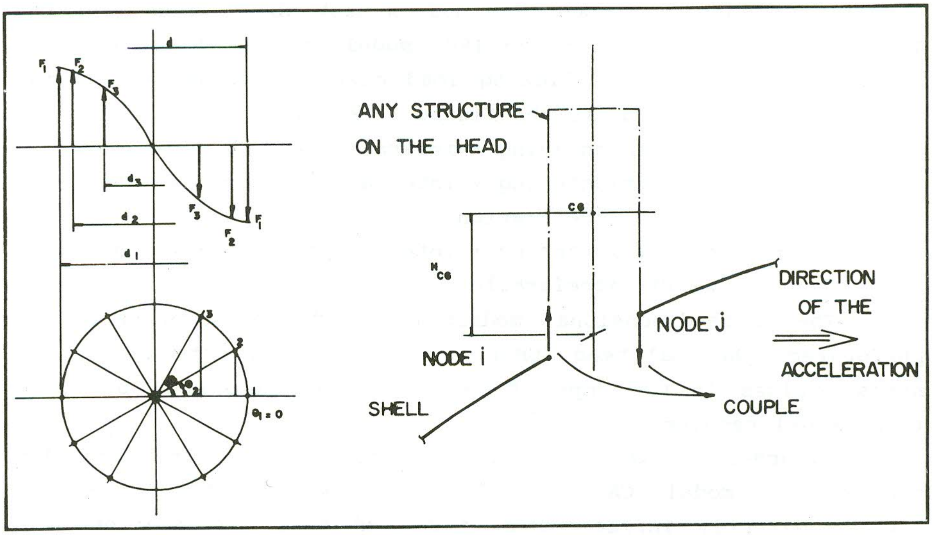

Figure 3: Couple definition for the 3-D closure head models

3. STRESS LIMITS

All of the stress limits mentioned below are based on reference [1]. Maximum-shear-stress theory is employed and all of the stresses refered below are stress intensities.

The three-dimensional models studied consist basically of a solid element part, that will be called the RING, and a shell element part, called the SHELL.

Initially the objective of the analyses was to verify the SHELL stresses, the RING included in the model only to provide rigidity to the SHELL. However, in order to have only an approximate idea of the stresses acting in the RING, some sections have been chosen and their average nodal stresses (SI) were compared with the general membrane stress limit (Sm). For the perforated shell only the primary stresses were studied, since this paper deals only with design procedures. For this shell, in certain conditions, one can classify the membrane stresses near a hole influence region as local ones (limiting stress: 1.5 x Sm).

4. RESULTS

From now on, some of the results obtained from the analyses are shown. Revision 4.4a of the ANSYS program [3], running on a VAX 785 computer, was used. The models took about 0:20h (for the 45° model) and 3:45h (for the 180° model) of CPU time to run a static analysis. The following load cases have been considered (accelerations refered to the Figure 1 axes):

CASE 1: bolt-thightening + internal pressure + weights

CASE 2: bolt-thightening + internal pressure + weights + z-axis acceleration

CASE 3: bolt-thightening + internal pressure + weights + x-axis acceleration

The three-dimensional models provided the stresses in the perforated spherical head (SHELL). A preliminary estimate of the stress values in the upper flange ring was also made by using these model results.

In order to exemplify the upper flange ring stress results from the 45° model (CASE 2) and the 180° model (CASE 3), a ring section directly influenced by the stud hole perturbations was considered. In doing so, the average nodal stress was evaluated for each load case and compared with the material primary membrane stress limit:

CASE 2 : average stress= 81.4 MPa < Sm=184 MPa

CASE 3 : average stress= 89.4 MPa < Sm=184 MPa (critical section)

To elucidate a bit about the stresses in the SHELL, some membrane stress results are shown in the following. Figure 4 shows this kind of stress for CASE 1. Classifying the membrane stresses as local, it comes:

$$SMX = 251.5 \text{ MPa} < 1.5 \times Sm = 276 \text{ MPa}$$

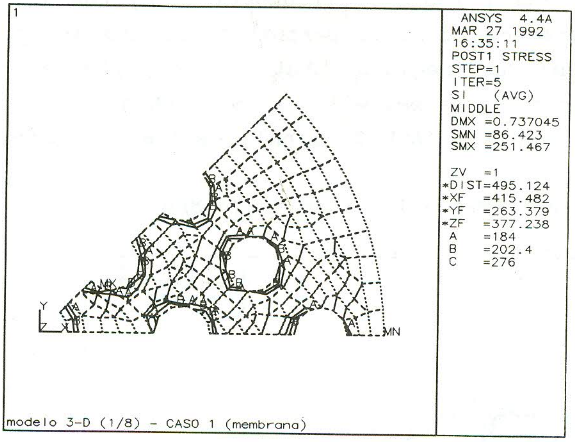

Figure 4: CASE 1 membrane stress results (MPa)

Figure 5 shows the SHELL membrane stresses for CASE 2. Taking them as local, we have:

$$SMX = 290.4 \text{ MPa} > 1.5 \times Sm = 276 \text{ MPa}$$

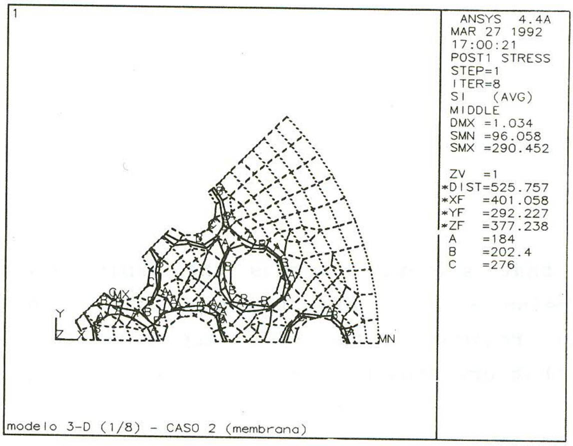

Figure 5: CASE 2 membrane stress results (MPa)

Although the limit above is exceeded, it happens only at a loaded node, corresponding therefore to a localized effect due to the way of application of the load, and is not to be verified. In the same way, all other regions circumscribed by the isostress C (276 MPa) are subjected to local perturbations and are not to be verified. The remaining regions, that are not affected by local effects, are under stresses below 1.5 x Sm (276MPa).

Figure 6 shows, for CASE 3, final results that are similar to CASE 2 ones. So:

$$SMX = 276.6MPa > 1.5 \times Sm = 276 \text{ MPa}$$

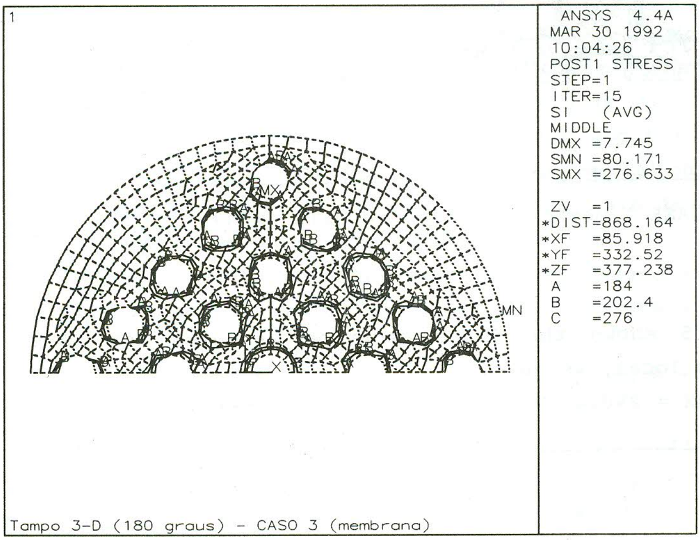

Figure 6: CASE 3 membrane stress results (MPa)

This value that slightly exceeds the limit occurs on a loaded node, so being a local affected one that is not to be verified. All other regions of the SHELL are subjected to local membrane stresses that are smaller than 1.5 x Sm.

5. <u>CONCLUSIONS</u>

From the results obtained with these three-dimensional models, one can say that all of the stresses are smaller than the limits stated in paragraph 4, since local effects due to concentrated loads were neglected.

REFERENCES

[1] "ASME Boiler and Pressure Vessel Code, Section III", American Society of Mechanical Engineers, 1989.

[2] M. B. Bickell, S. H. Dance; "An Elastic Analysis of Axially Perforated Cylinders with Applications to the Design of Reactor Vessel Flanges", First Int. Conf. on Structural Mechanics in Reactor Technology, Berlin, 1971.

[3] G. J. De Salvo, R. W. Gorman; "ANSYS Engineering Analysis System, User's Manual", rev. 4.4a, 1990.