3-D MODELS TO ANALYZE A PWR PRESSURE VESSEL

C.A. De Oliveira, IPEN/CNEN-SP
G.M.V. De Assis, COPESP

ABSTRACT

The pressure vessel of a PWR was analyzed using two models: a 3-D model to represent the perforated upper torospherical head and an axisymmetric model for the whole vessel. The former model used mainly STIF63 shell elements in the perforated center of the head and STIF45 3-D 8-nodes isoparametric elements for the flanges. The flange bolts were discretized in STIF4 3-D beam elements. Constraint equations were defined to compatibilize the rotations between the shell and the 3-D parts. For vertical loads, the model was developed in 45 degree and for horizontal ones a 180 degree model was used. Convenient symmetries were defined in both models.

INTRODUCTION

The pressure vessel in this study is a typical PWR (design pressure of 16.55 Ma and design temperature of 343 degree C). The spherical part of the closure head is penetrated by 21 parallel tubes related to the control rod drive mechanisms, and by 4 more tubes, also parallel to the formers, that conduct the in-core instrumentation probes.

There have been 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, specified by its indications. For the analyses developed in this paper, the following loads were considered: internal pressure, weights, bolt-tightening, and accelerations, as in an earthquake, acting in the x and z directions of the coordinate system (see Figure 1.b).

Considering the loads mentioned above, three finite element models have been developed [3] for analyzing the vessel behavior; two three-dimensional models for verifying the perforated part of the head, and an axisymmetric model of the whole vessel for verifying the shell and vessel flanges.

PTC nov.
ANSYS Technology Conference and Exhibition, Pittsburgh, Pa, USA, June 8-12 1992

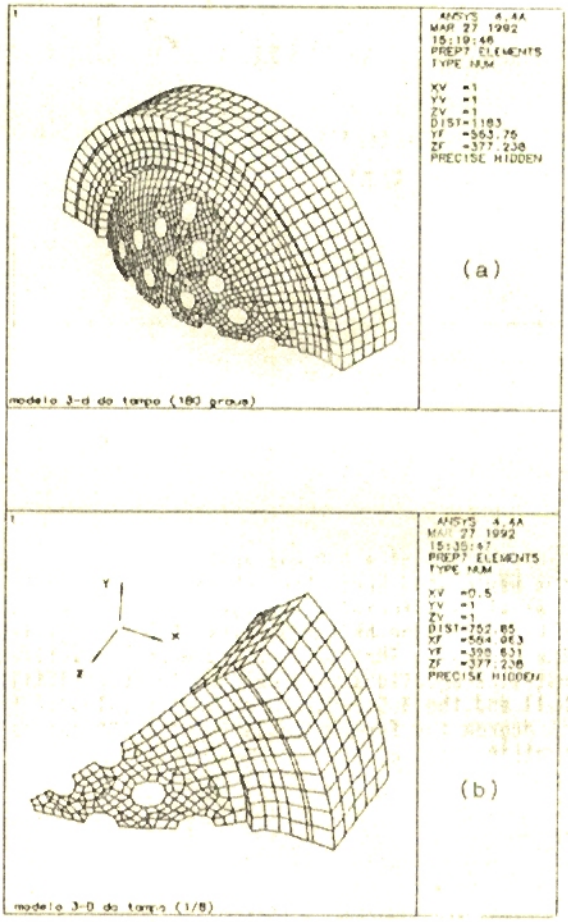

Figure 1: 3-D pressure vessel

head models

(a) 180° model

(b) 45° model

THREE DIMENSIONAL MODELS OF THE HEAD

In order to know the behaviour of the head, two three-dimensional finite element models: a 45 degree model, to study the efect of a vertical acceleration, and a 180 degree model for a horizontal one (internal pressure and weights act in the two models) - see Figure 1.

In these models the flange ring was meshed with STIF45 3-D isoparametric solid elements with 8 nodes, and the spherical shell, with STIF63 elastic quadrilateral shells. The use of shell elements makes the model simpler than if it was made only with solid elements, besides introducing a certain conservatism in the analysis. This is due to the fact that when we use shell elements, the internal pressure is applied at the medium diameter of the shell, while using solid elements, the internal pressure would have been applied at the internal diameter of the head (actual situation). We still have used gap elements (STIF52 3-D interface) to model the contact between the upper and lower flanges, beam elements (STIF4 3-D elastic beam) to model the flange bolts and mass elements (STIF21 generalized mass) 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's Modulus of the used material [2]. For the shell part of the head, all of the existing holes, related to the control rod drive mechanisms and to the instrumentation tubes, were included in the model, but the stiffness of the associated nozzles has not been considered (this is a conservatism when analyzing the shell under the holes influence).

All of the analyses accomplished with the 3-D models are linear, except for the local non-linearities (gap elements).

BOUNDARY CONDITIONS AND COUPLING OF NODES

We have been using the following boundary conditions with these models:

- symmetry for faces at θ=0 degree (45 degree and 180 degree models) and θ=180 degree (180 degree model), in the Figure 2 coordinate system. (θ=45 degree for 45 degree model).
- 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.

The different finite element types used in these head models have nodes with different numbers of degrees of freedom. In order to keep the models consistent, we have coupled the nodes of different adjacent elements that, in other ways, could lead to some wrong result. By the use of rigid regions, we have coupled the shell elements 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, we have used STIF4 3-D rigid beams with no mass to couple the STIF45 solid element nodes to the adjacent STIF63 shell element ones.

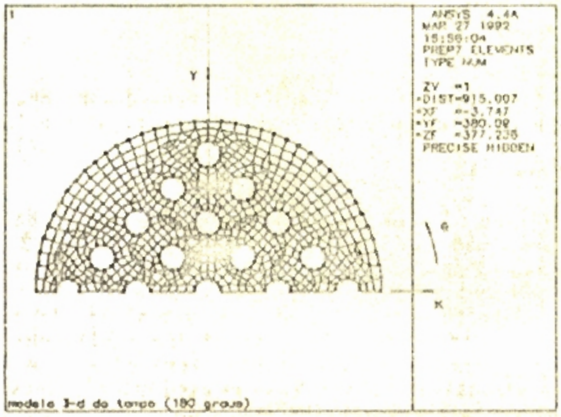

Figure 2: Shell part of the 180° pressure vessel head model

LOADINGS

The loadings used in the three-dimensional model analysis are: bolt-tightening (1.44x10 degree N), internal pressure (16.55 MPa), weights and x- (49 m/s^2 - 180 degree model) and z (40 m/s^2 - 45 degree model) accelerations (Figure 1.b coordinate system).

An interative process has been used to find a suitable initial deformation that, once applied to the beam elements, lead to the actual bolt-tightening.

The weights ae considered in the analyses by imposing a 9.81 m/s^2 vertical acceleration to the structural model.

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 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 the effects of that moment. Hence, for each structure (control rod drive mechanisms, instrumentation tubes and supporting structure) we have applied a set of couples to the models that corresponds to a specific moment, thus reproducing the global original effect. This is done in accordance with Figure 3, where a moment is represented by some couples that consist of a cosinusoidal force distribution. Figure 2 shows the shell element model part. Its highlighted nodes correspond to the ones used to apply the couples to the models and/or define the concentrated mass finite elements.

THREE DIMENSIONAL MODELS OF THE HEAD

In order to know the behaviour of the head, two three-dimensional finite element models: a 45 degree model, to study the efect of a vertical acceleration, and a 180 degree model for a horizontal one (internal pressure and weights act in the two models) - see Figure 1.

In these models the flange ring was meshed with STIF45 3-D isoparametric solid elements with 8 nodes, and the spherical shell, with STIF63 elastic quadrilateral shells. The use of shell elements makes the model simpler than if it was made only with solid elements, besides introducing a certain conservatism in the analysis. This is due to the fact that when we use shell elements, the internal pressure is applied at the medium diameter of the shell, while using solid elements, the internal pressure would have been applied at the internal diameter of the head (actual situation). We still have used gap elements (STIF52 3-D interface) to model the contact between the upper and lower flanges, beam elements (STIF4 3-D elastic beam) to model the flange bolts and mass elements (STIF21 generalized mass) 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's Modulus of the used material [2]. For the shell part of the head, all of the existing holes, related to the control rod drive mechanisms and to the instrumentation tubes, were included in the model, but the stiffness of the associated nozzles has not been considered (this is a conservatism when analyzing the shell under the holes influence).

All of the analyses accomplished with the 3-D models are linear, except for the local non-linearities (gap elements).

BOUNDARY CONDITIONS AND COUPLING OF NODES

We have been using the following boundary conditions with these models;

- symmetry for faces at θ=0 degree (45 degree and 180 degree models) at θ=180 degree (180 degree model), in the Figure 2 coordinate system. (θ=45 degree for 45 degree model).
- 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.

The different finite element types used in these head models have nodes with different numbers of degrees of freedom. In order to keep the models consistent, we have coupled the nodes of different adjacent elements that, in other ways, could lead to some wrong result. By the use of rigid regions, we have coupled the shell elements 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, we have used STIF4 3-D rigid beams with no mass to couple the STIF45 solid element nodes to the adjacent STIF63 shell element ones.

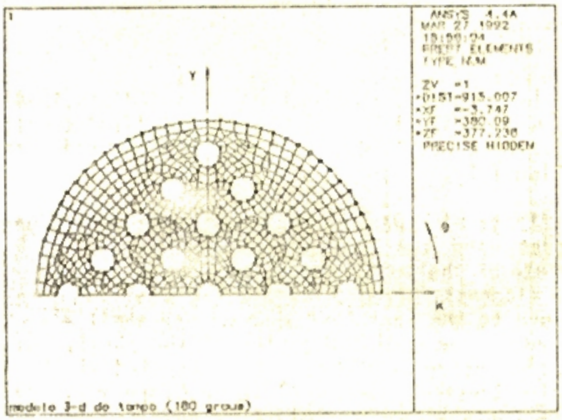

Figure 2: Shell part of the 180° pressure vessel head model

LOADINGS

The loadings used in the three-dimensional model analysis are: bolt-tightening (1.44x10 degree N), internal pressure (16.55 MPa), weights and x- (49 m/s^2 - 180 degree model) and z (40 m/s^2 - 45 degree model) accelerations (Figure 1.b coordinate system).

An interative process has been used to find a suitable initial deformation that, once applied to the beam elements, lead to the actual bolt-tightening.

The weights ae considered in the analyses by imposing a 9.81 m/s^2 vertical acceleration to the structural model.

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 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 the effects of that moment. Hence, for each structure (control rod drive mechanisms, instrumentation tubes and supporting structure) we have applied a set of couples to the models that corresponds to a specific moment, thus reproducing the global original effect. This is done in accordance with Figure 3, where a moment is represented by some couples that consist of a cosinusoidal force distribution. Figure 2 shows the shell element model part. Its highlighted nodes correspond to the ones used to apply the couples to the models and/or define the concentrated mass finite elements.

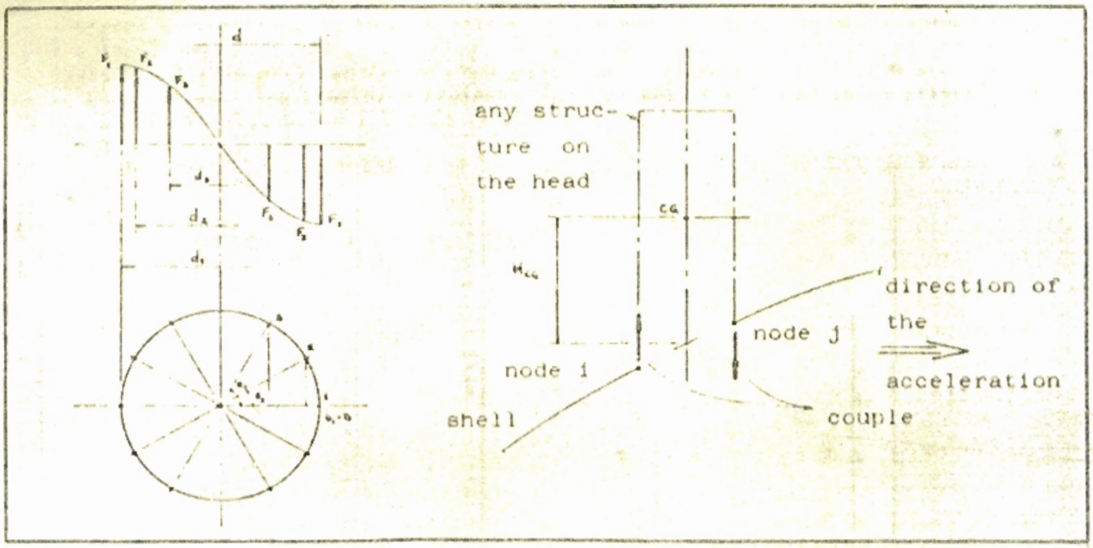

Figure 3: Couple definition for the 3-D closure head models

STRESS LIMITS

All of the stress limits mentioned below are based on Reference [1].

THREE-DIMENSIONAL MODELS

The models shown in paragraph 2 consist basically of a solid element part, that will be called RING, and a shell element part, that we will call SHELL.

At first, since the objective of ours is to verify the SHELL stress, the RING has been included in the model only to make the SHELL rigid. Meanwhile, in order to only have an approximate idea of the stress acting in the RING, we have chosen some sections of it and have compared their average model stresses (SI) with the general membrane stress limit (S_m). For the perforated shell we have studied only the primary stress, since this paper deals only with design procedures. For this shell, in certain conditions, one can classify the membrane stsresses near a hole influence region as local ones (limiting stress: 1.5 x S_m.).

In order to exemplify the upper flange ring stress 45 degree model (CASE 2) and 180 degree model (CASE 3) results, we have taken a ring section directly influenced by the stud hole perturbations. In doing so, we have evaluated the *average modal stress for each load* case, and compared them 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=1184 MPa (critical section)

To elucidate a bit about the stresses in the SHELL, we have only shown in sequence some membrane stress results. Figure 7 shows this kind of stress for CASE 1. Classifying the membrane stresses as local, it comes:

SMX = 251.5 MPa < 2.5 x Sm = 276 MPa

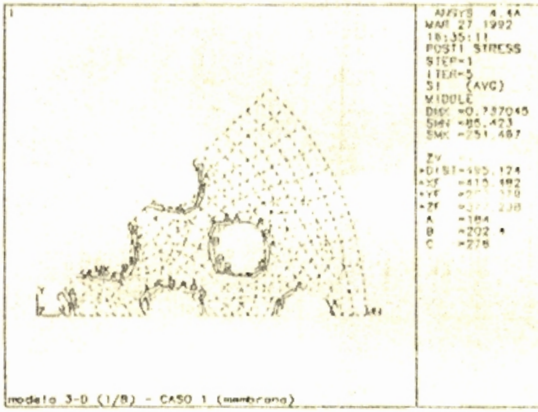

Figure 7: CASE 1 membrane stress results (MPa)

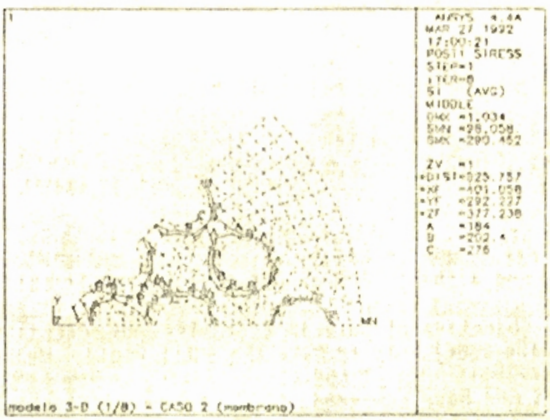

Figure 8: CASE 2 membrane stress results (MPa)

Although the limit above is exceeded, that happens only in a loaded node, therefore, corresponding 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 circumscribed isostress C (276 MPa) regions are subjected to local pertubations, 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 9 shows, for CASE 3, final results that are similar to CASE 2 ones. So:

SMX = 276.6MPa > 1.5 x Sm = 276 MPa

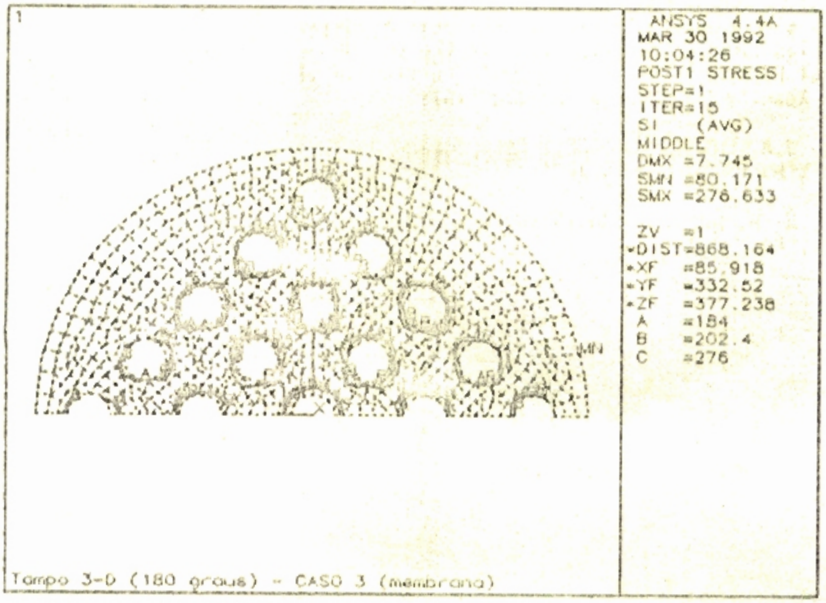

Figure 9: CASE 3 membrane stress results (MPa)

This value slightly exceeding 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 subject to final local membrane stresses that are smaller than 1.5 x Sm.

It is to be noted that in no analysis case, SMXB (SMX plus the estimated error) was used, since the estimated error is very influenced by the large stress gradients that exist among the loaded nodes and the adjacent model regions, and that the use of concentrated loads is just a loading technique whose local influence is not important for the final results.

CONCLUSIONS

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

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. DeSalvo, J.A. Swanson: "ANSYS Engineering Analysis System, User's Manual", Rev. 4.2b

[4] G.J. DeSalvo, R. W. Gorman, "ANSYS Engineering Analysis System, User's Manual, Rev. 4.4a