EXPERIMENTAL AND THEORETICAL STRESS ANALYSIS IN A NUCLEAR STEAM GENERATOR HEAD

S. CURIONI, G. FAVRETTI, A. FREDDI,

Istituto di Meccanica Applicata alle Macchine, Facoltà di Ingegneria, Università di Bologna, Bologna, Italy

A. GARRO,

FIAT, Sezione Energia Nucleare, Torino, Italy

ABSTRACT

A multiple approch for elastic stress analysis in a nuclear steam-generator head is presented. Two experimental techniques (photoelastic, electrical strain gages) are used for a first study and the solutions are compared with one another and with those derived from finite-element analysis. It is shown the importance of the application of different complementary experimental methods for prediction of stresses at discontinuities and for cheking of the numerical program which provides the facility for efficient investigations of the effect of changes in the geometric parameters of the structure.

1. INTRODUCTION

In the practice of designing of sodium-cooled nuclear fast reactors the study of steam generators is of great importance for the high temperatures and consequentely for the high working pressures. The most characteristic structural part is the head for its complex shape and for high pressure loading condition. Therefore, for the prevision of the behaviour of this structure and for the definition of optimum shapes a detailed stress analysis in elastic field is necessary. The present analysis refers to a preliminary dimensioning solution carried out at C.N.E.N.-P.R.V:-Bologna. The head of great thickness is shown in fig. 1: in hemispherical part there are a hand-hole with a flat cover and a radial nozzle.
The component is loaded only by internal pressure ($p = 200$ kg/cm^2), being the pre-tightening load negleeible owing to the type of gasket used.

2. SCOPE OF STUDY

In order to predict stresses at discontinuities and to give the designer the possibility of extending the results to similar components, the three following analyses have been carried out:

- a threedimensional photoelastic analysis by "freezing" technique;
- a strain gages analysis;
- a threedimensional numerical analysis by finite element method.

After the photoelastic results have been controlled with strain gages test in the points of higher stresses at the surfaces, the finite element program has been planned and checked: in this way such program can be used for efficient investigations of the effect of changes in the parameters which define the geometry of the structure.

3. EXPERIMENTAL AND NUMERICAL METHODS

A - Photoelastic analysis

The photoelastic research is carried out through the following points:

- model designing
- "freezing" of strains
- analysis of stresses.

The model designing requires the definition of the scales of lengths and forces and the study of schematizations for photoelastic test. The lengths scale (1:4) has been chosen to obtain tolerances on the linear dimensions in order to maintain within acceptable limits the errors on the membrane stresses (see later). The tube plate is simpler than the prototype with 56 non-through holes, on the basis of the validity of the similarity between the bored plates according to the O'Donnel and Langer theory [1], fig. 2. The connection with the sleeve is eliminated. The model is obtained by casting in all-metal mold, according to an already widely used process (see Freddi [2]). In this way the nozzles and the body of the head are integral to avoid disturbances of welding on the isochromatic fringe pattern. This aspect is particularly important since the welding area usually correspnds to the stress concentraion area. The aluminium alloy mold, fig. 3, has been designed to contain within a limit of 1% the maximum percent error on the model thickness. In Freddi [3] are given the details of this study together with tests which contributed to the set up of the pretightening device of the cover, formed of 12 calibrated springs, to the selection of the forces scales and to the optimisation of the welding technique between the two parts of the model.

After "freezing", from the model have been cut the slices according to the diagram of the slicing given in fig. 9 and following. For each of the shown planes the tangential stresses and the normal stresses have been given. In fig. 4 the isochromatic patterns for meridional plane of simmetry are given and in fig. 5 has been plotted the isostatic network on the basis of the isocline pattern. The photoelastic analysis has pointed out the stress concentrations on the internal fillet in the junction of the body with the tube plate, at the internal and external fillets between nozzles and shell in the plane of symmetry and at the conical reinforcement of the nozzle.

B - Extensometric analysis

The model fig. 6, has been cast machined. The material is UNI 3046 G--AL-CV 3 FE-MG-NI aluminium alloy with the following characteristics:

Elastic Modulus E = 8320 kg/mm^2
Poisson Coefficient = 0.374
Ultimate tensile stress σ_R = 20.6 kg/mm^2

This material has been chosen in order to reduce the test pressure; this allows a reduction of errors due to the effect of the pressure acting on strain gages. The model realised on a scale 1:2, presents some semplifications:

- the bottom plate without holes;
- supporting flange simulating the connection between head and structure;
- nozzle with a flange used for the instrumentation.

For the loading of the model an oleodinamic circuit with two different oils have been used in order to avoid the direct contact of the circuit-oil with the strain gages. The outlet of the electric cables of the strain gages inside the model and the extensometric bridge has been realised by a connector. This connector, fig. 7, is realized by means of an araldite plate which the strain gages cables cross; this plate is intermediate between the nozzle flange and a steel plate whith holes to support the internal pressure. The strain gages have been placed in the areas of discontinuity stresses. (Strain gages Hottinger-Baldwin: 6/120 LA 21 for the measurement of the deformations along the parallels, O 6/120 LE 11 for the measurement of the deformations along the meridians). The selection of the measurement points has been suggested by the results of the photoelastic analysis. The tests have been carried out for different values of pressure up to 50 kg/cm^2. With this pressure there is an error of about 40 $\mu\varepsilon$ on the value of the strain gages deformation due to the hydrostatic pressure. Taking into account all the systematic and casual errors it may conclude that the results obtained are effected by an error of about 5% (see Favretti, Curioni [4]).

C - Numerical Analysis

The determination of the stress state in this structure by the finite element method has been reached by developing a hexahedral element with twenty-four degrees of freedom inserted in the AMSA code of FIAT. The subdivision into elements has been made in according with the schematization shown in fig. 8 obtained from the plotter, (594 nodes and 330 elements, utilizing the structure symmetry). The simulation of the bored plate has been made by reducing the local elastic modulus according to O'Donnel and Langer [1] but taking into account a central hole necessary to make the schematization possible through elements of the same type. The loads applied correspond to the internal pressure and to the action of the bolts. The system has been constrained so to make possible the radial displacements at the supporting flange and the vertical displacement at the plate centre. The results in the present paper are limited to the areas of major interest for a direct comparison with the other methods of analysis. The detailed results of analysis are reported in Garro, Verona [5] .

4. RESULTS

Stresses are given in adimensional form with the reference stress equal to membrane stress in the hemispherical part of the head of medium radius r_m and thickness t ($\sigma_o = \frac{p\ r_m}{2t}$).

Tangential and normal stresses in the symmetry plane of the structure at the internal and external surfaces are reported in figs. 9,10, 11. Tangential and normal stresses for the three planes perpendicular to the symmetry plane and containing the axes of the two nozzles and of the cylindrical body are reported in figs. 12,13,14.

5. CONCLUSIONS

In figures 9,10,11,12,13,14 is given a comparison of the results of the three methods used. The three methods of analysis agree in the location of the stress peaks: the maximum stress peak is at the connection of the cylindrical body with the tube plate in the internal fillet for both the tangential stress σ_P and normal stress σ_N and the ratio σ_P / σ_N is of the order of 2.5. In the areas of junction of the nozzles with the hemispherical shell other stress peaks are localised and the maximum values are at the internal surface for the nozzle and at the external surface for the hand-hole. In these areas the highest values occur for the normal stress σ_N. The measurements in the radial planes, figs. 12,13,14, provide information about the circumferential change in the stress state.

The finite elements method and the extensometric analysis closely agree at the stress peaks while in other areas a better agreement exists between the numerical analysis and the photoelastic. The differences found out can be ascribed to some differences in the models: the absence of the external supporting flange and the presence of a preload condition on the hand hole in the photoelastic model.

From the results obtained it can be seen the importance of a multiple approach in order to define the magnitude and location of the maximum stress state, providing therefore the designer with the information required for a reliable design.

REFERENCES

[1] O'DONNEL, W.J. LANGER, M., "Design of perforated plates" Trans. Asme Journ. Eng. for Ind. Vol. 84, S.B., 3 (1962)

[2] FREDDI, A., "Analisi fotoelasticimetrica di un vessel di reattore nucleare". Atti Ist. Mecc. Appl. n°. 39 (1969)

[3] FREDDI, A., "Analisi fotoelasticimetrica di una testata di generatore di vapore" Atti Ist. Mecc. Appl. n°. 40 (1969)

[4] FAVRETTI, G., CURIONI, S., "Analisi estensimetrica su una testata di generatore di vapore". Atti Ist. Mecc. Appl. n°. 46 (1970)

[5] GARRO, A., VERONA, E., "Analisi tridimensionale dello stato di sollecitazione del fondo di generatore di vapore". FIAT-SEN-NT1-N-IN/AG--EV-4 April (1971)

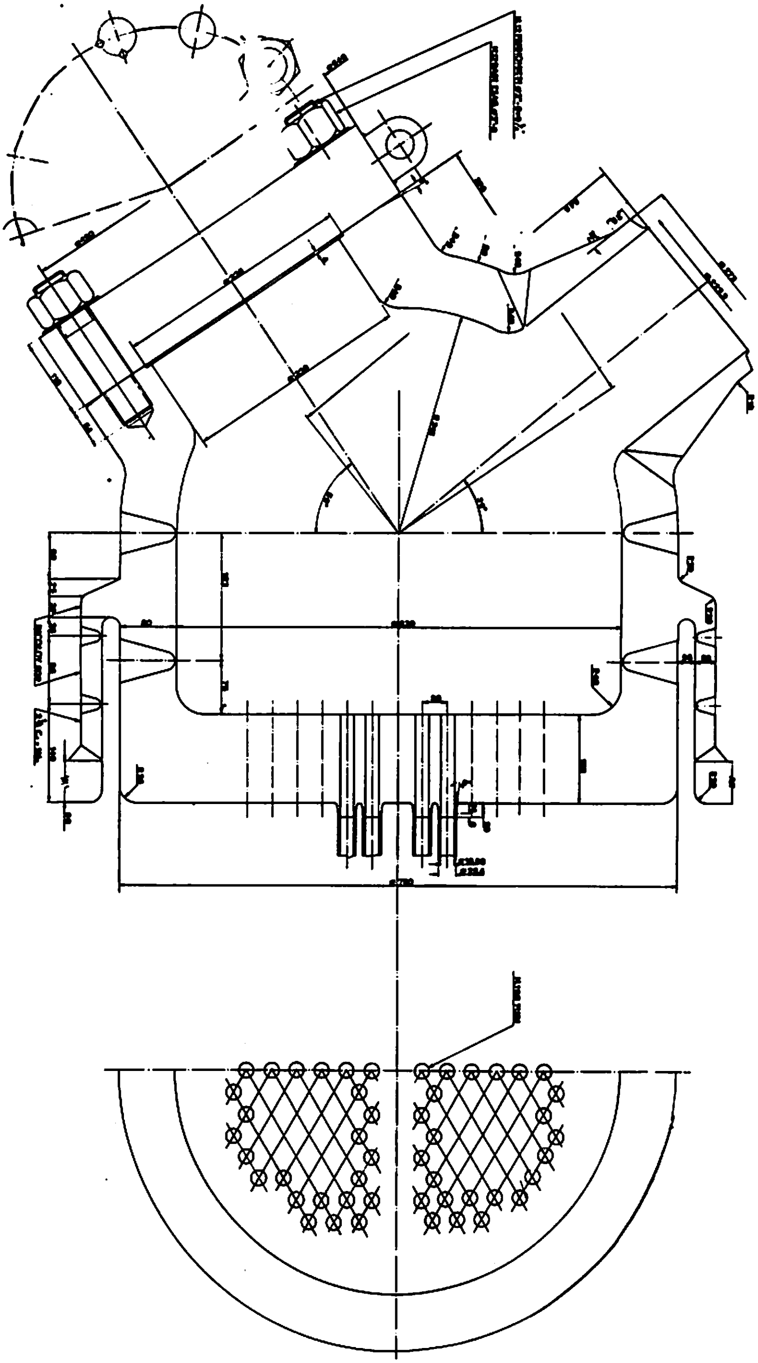

fig. 1 : Steam-generator head

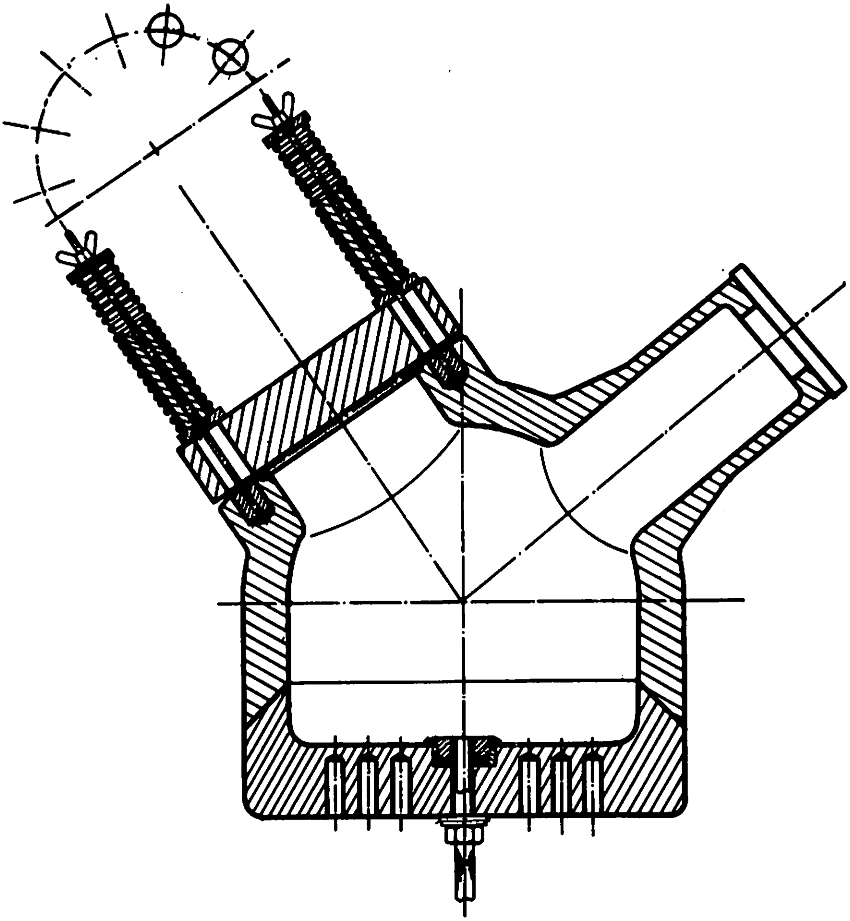

fig. 2 : Photoelastic model

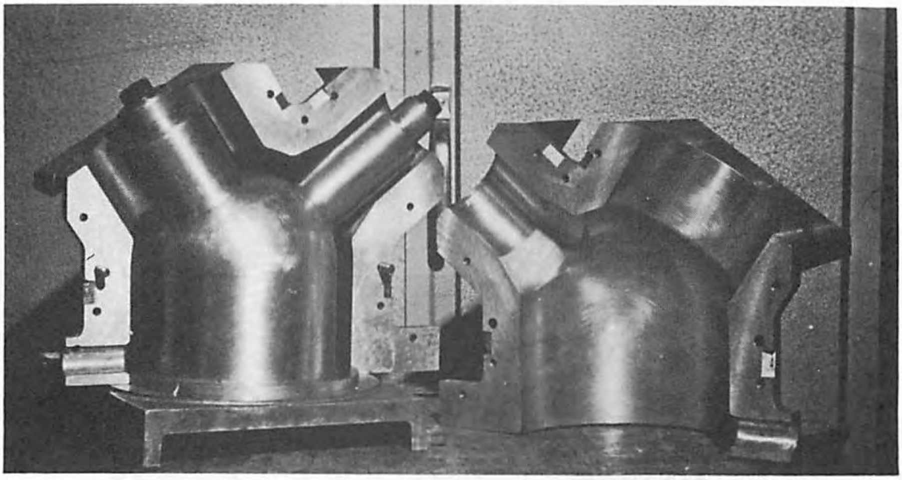

fig. 3 : All-metallic mold

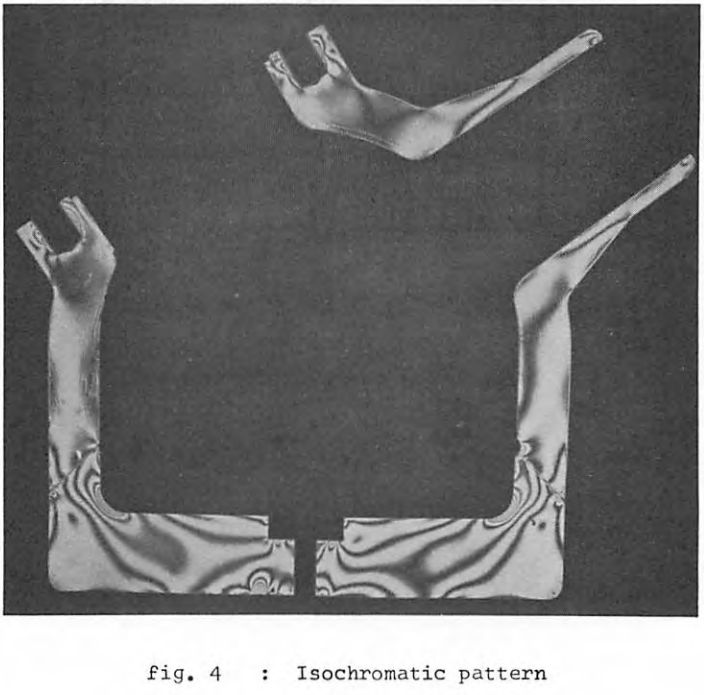

fig. 4 : Isochromatic pattern

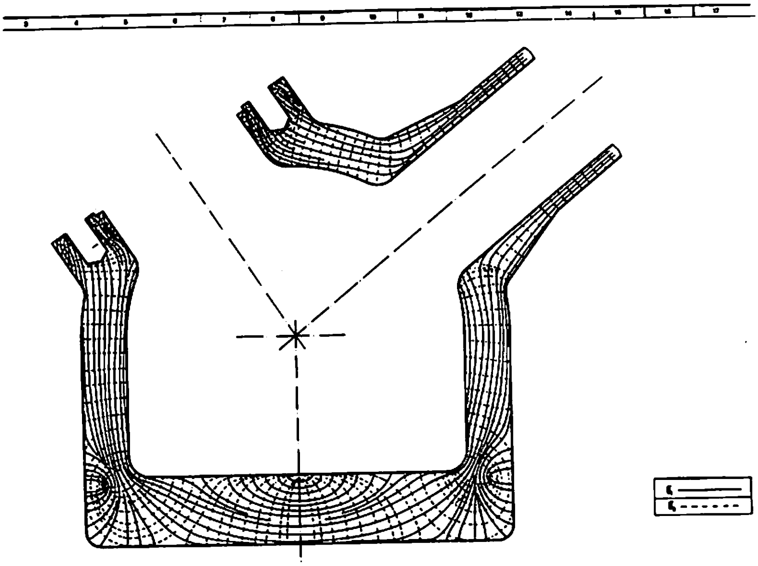

FIG.5

fig. 5 : Isostatic network

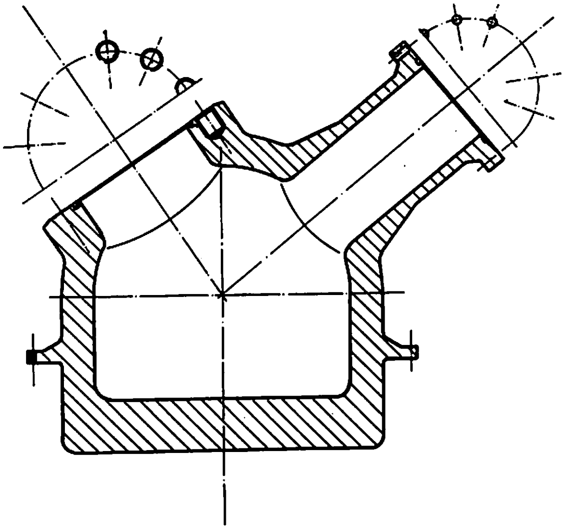

fig. 6 : Strain gage model

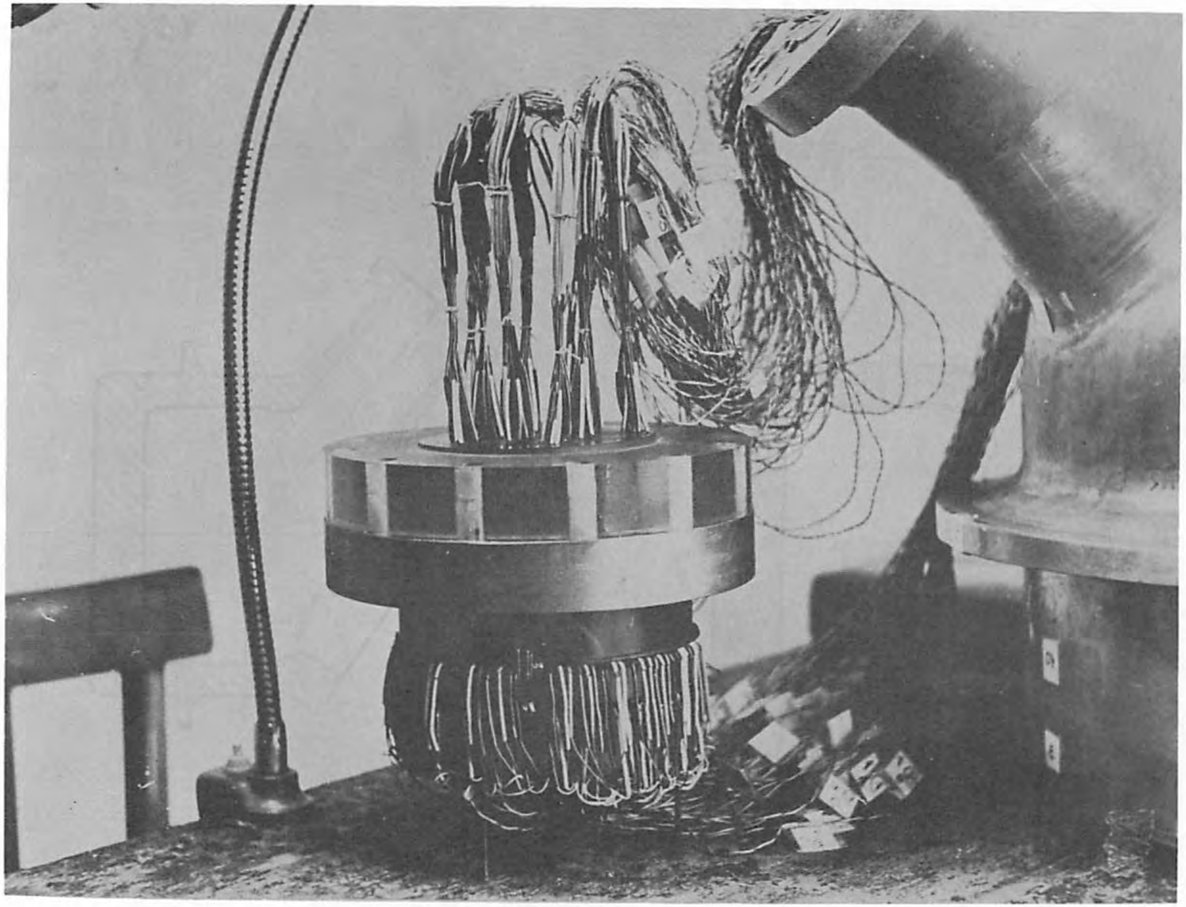

fig. 7 : Connector

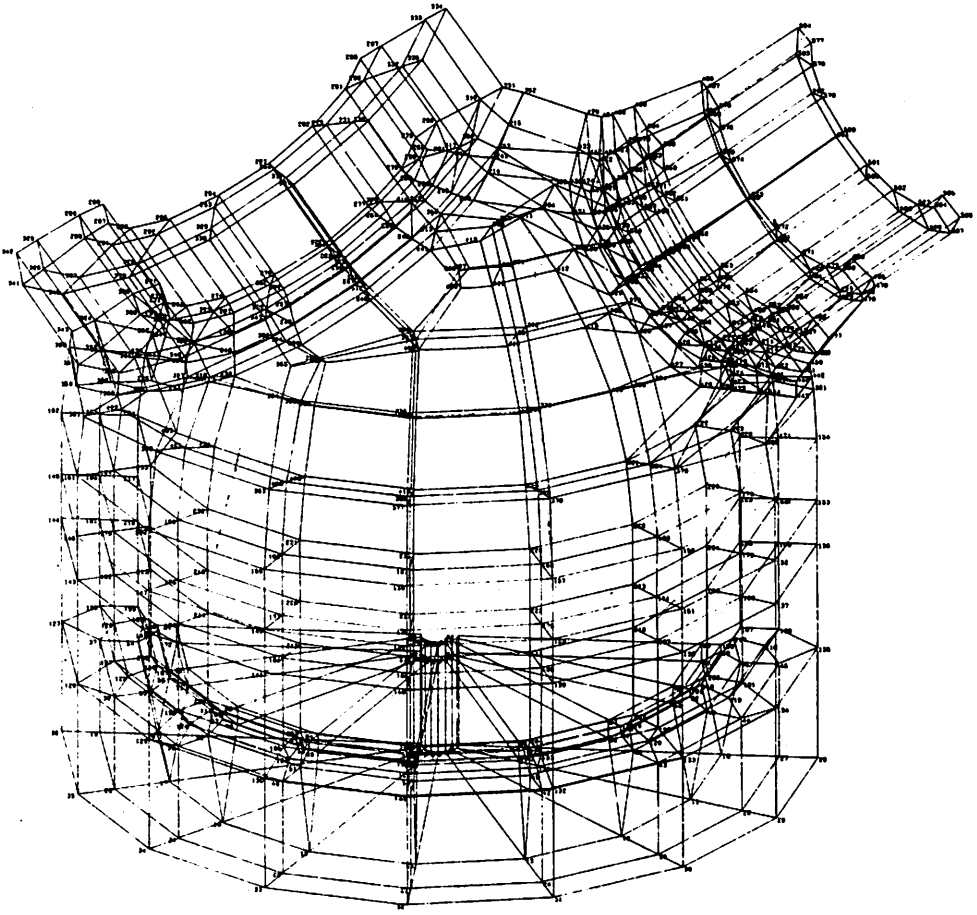

fig. 8 : Element model

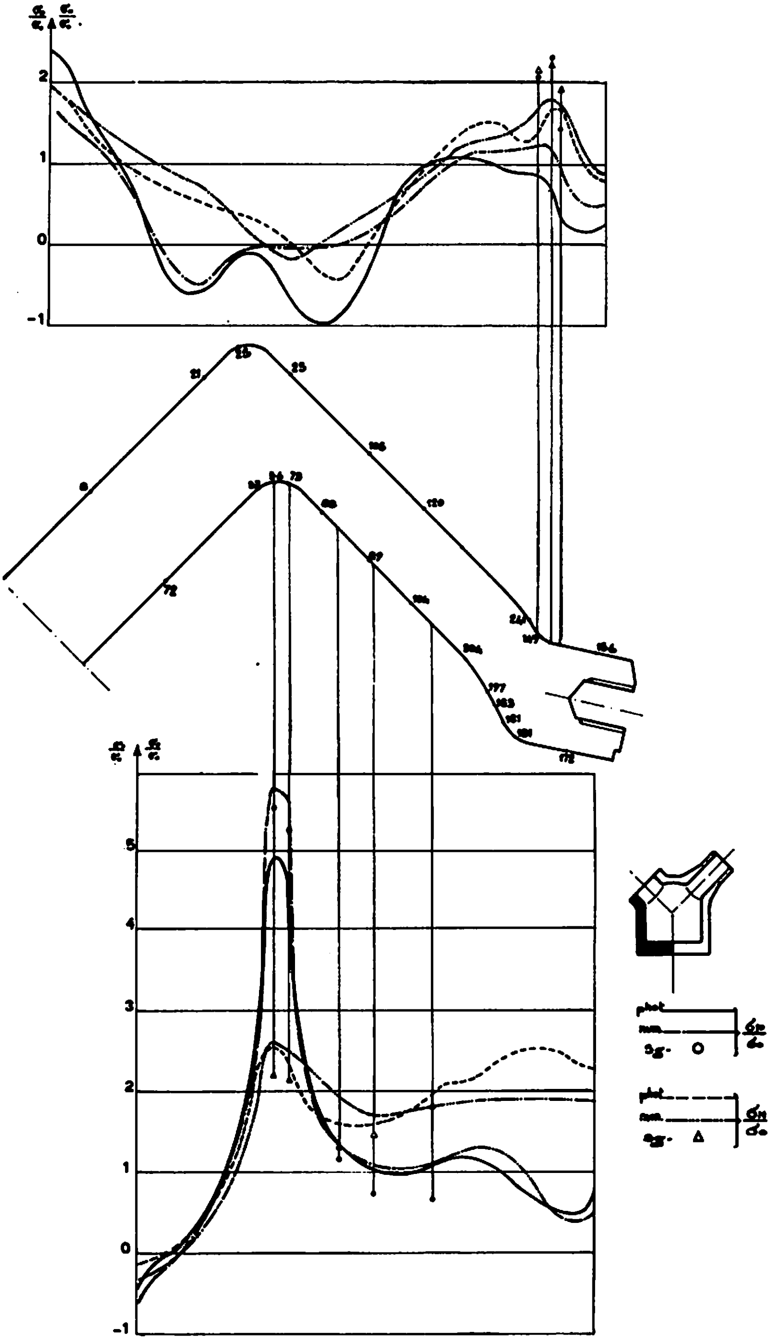

fig. 9 : Comparision between measured and computed surface stress.

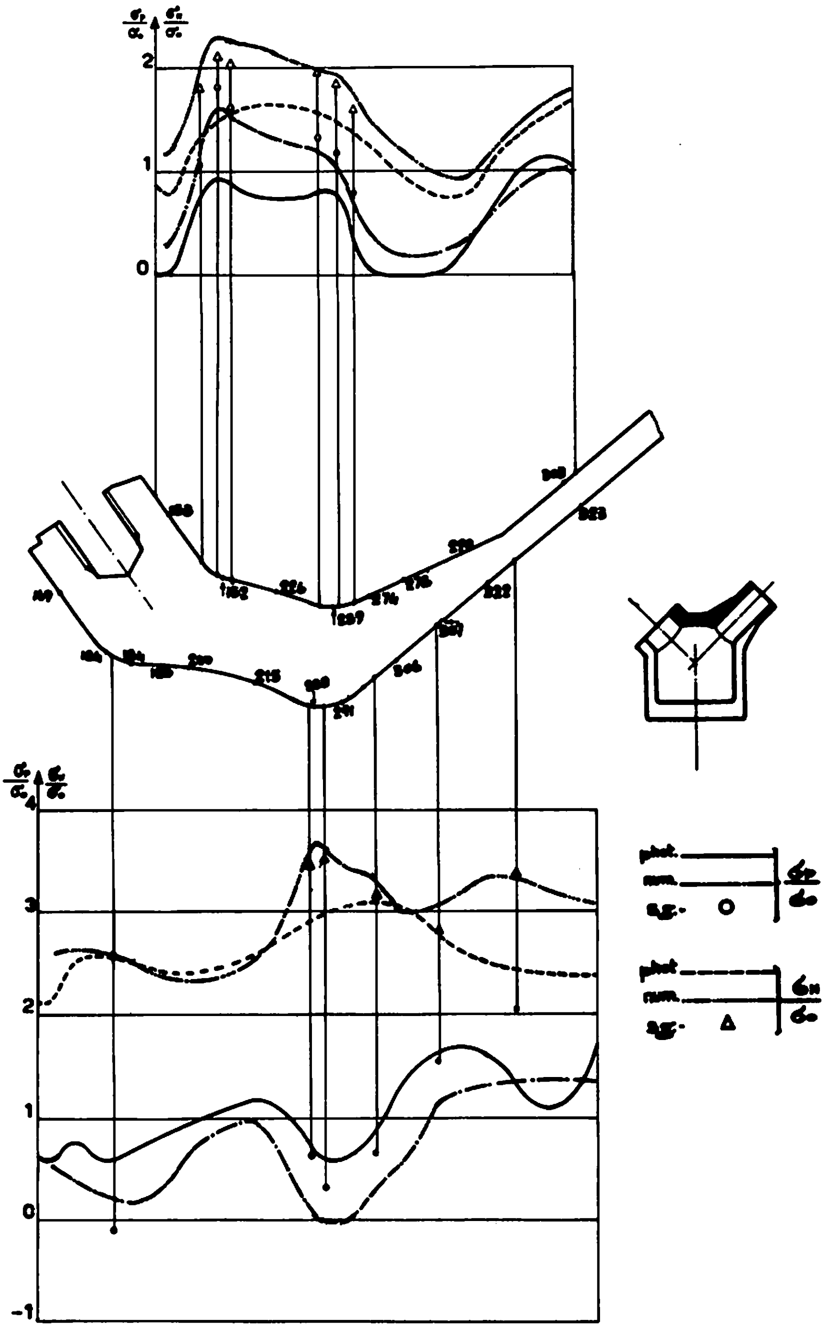

fig. 10 : Comparision between measured and computed surface stress

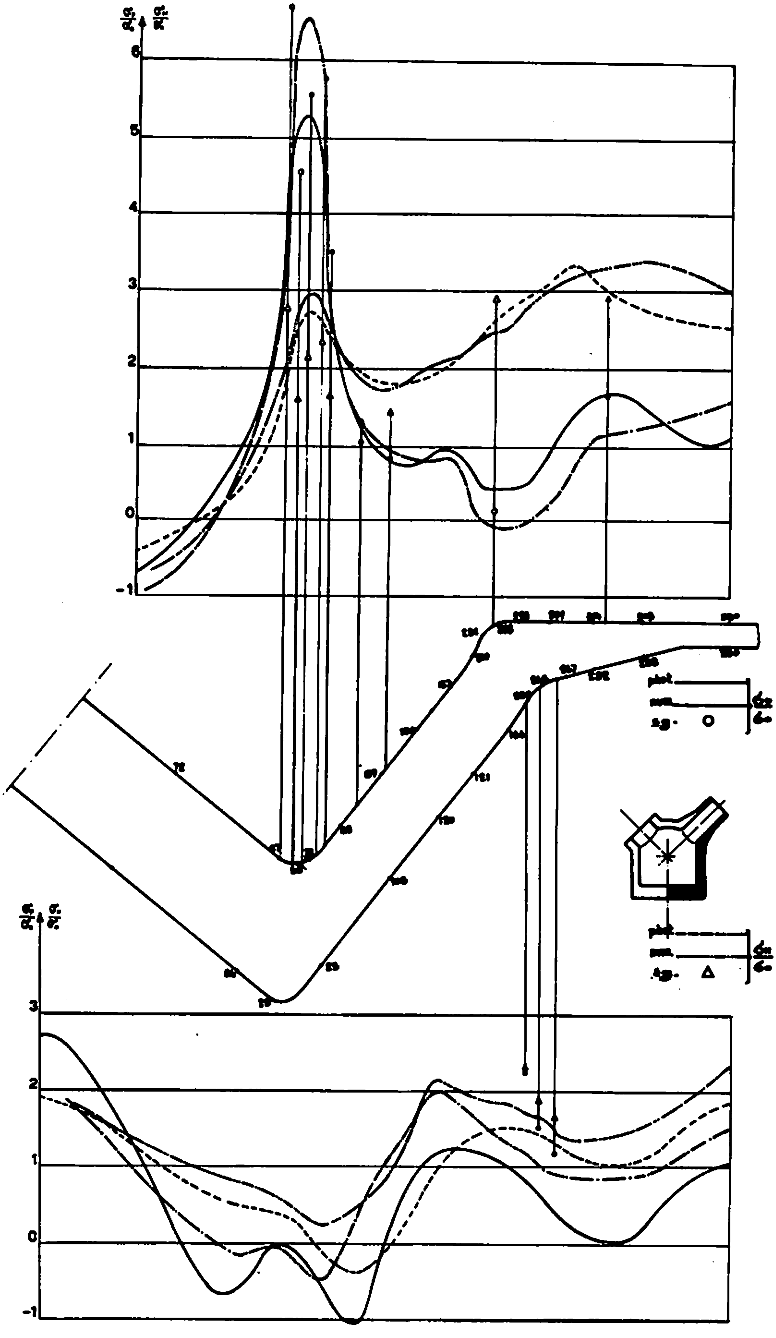

fig. 11 : Comparision between measured and computed surface stress

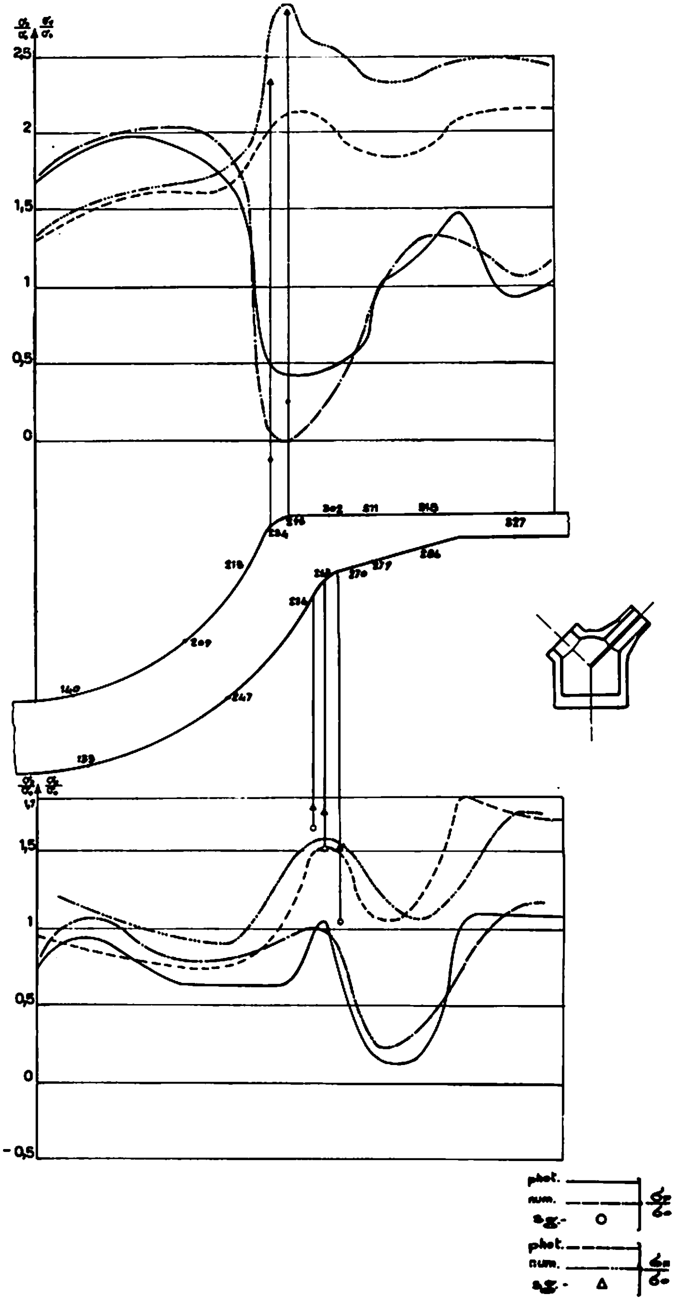

fig. 12 : Comparision between measured and computed surface stress

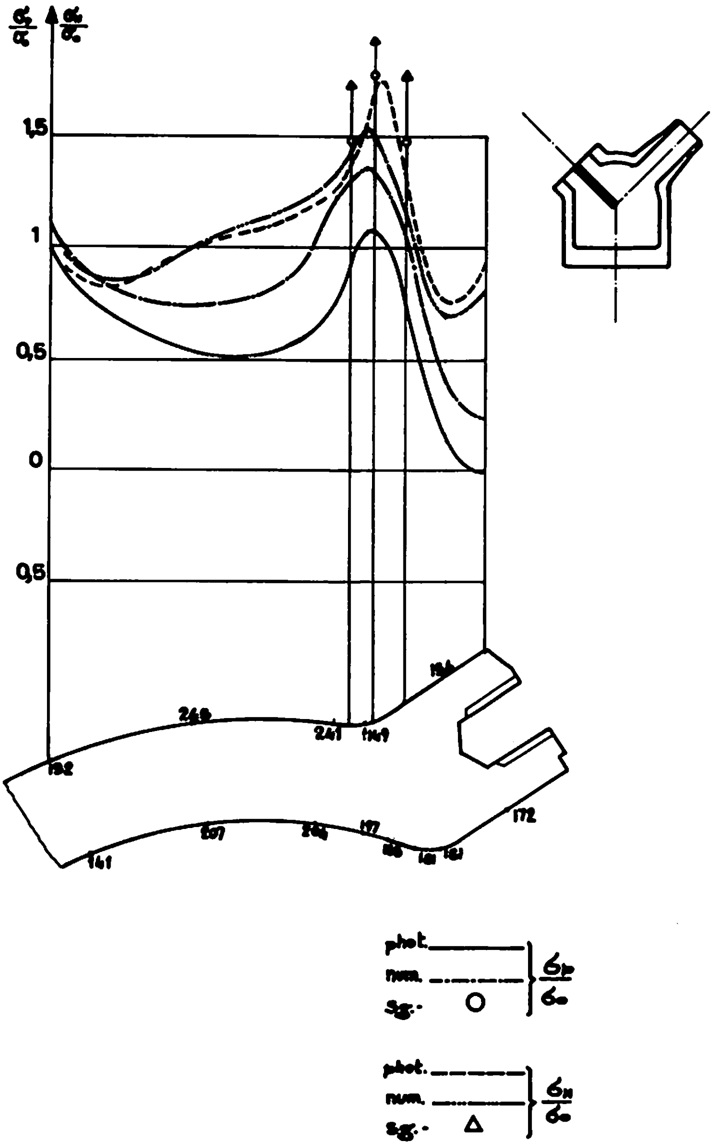

fig. 13 : Comparision between measured and computed surface stress

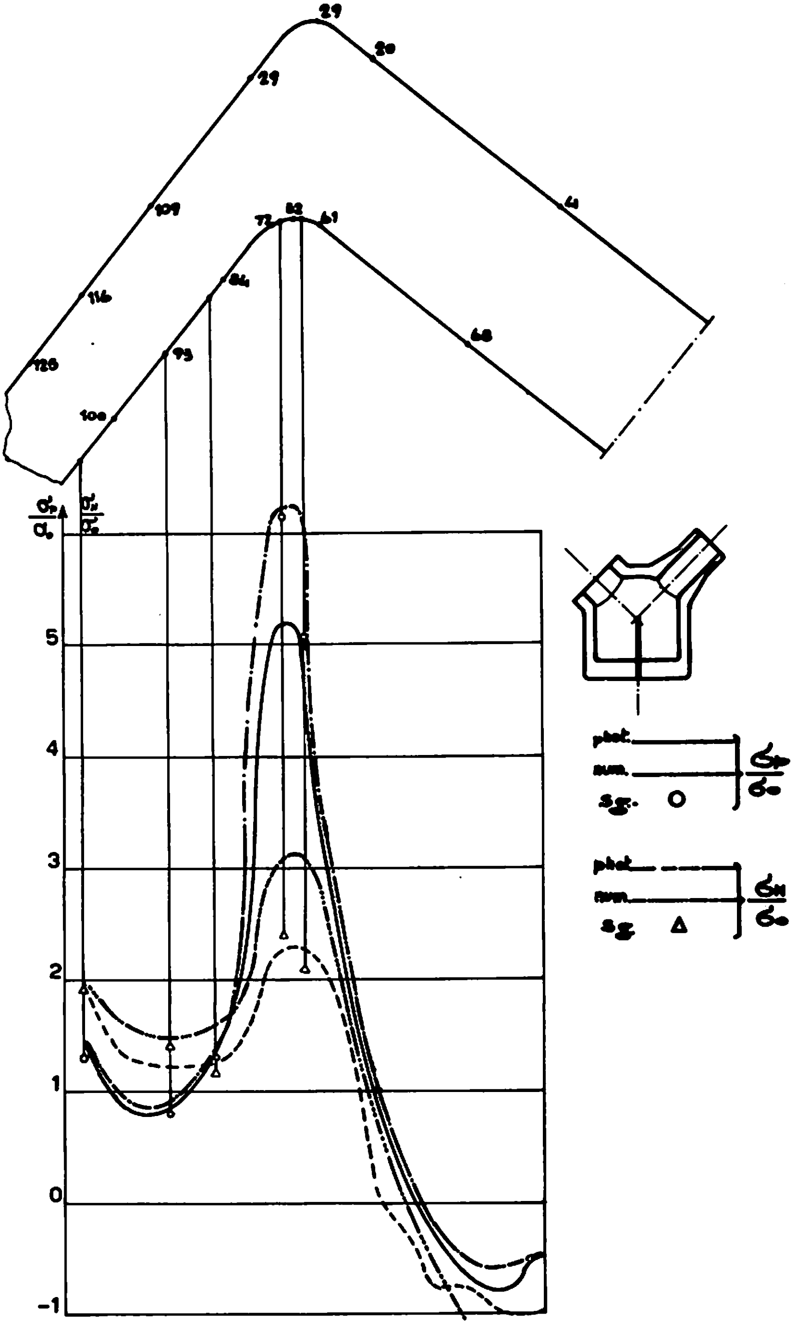

fig. 14 : Comparision between measured and computed surface stress.

DISCUSSION

Q H. BONIN, France

Le modèle utilisé ne prend en compte que la plaque et le fond à l'exception du corps cylindrique. Il semble que la présence d'une telle liaison sur la seconde face de la plaque doive en modifier la rigidité. Dans ces conditions, les résultats peuvent-ils être suffisamment significatifs ?

A S. CURIONI, Italy

The head is connected with the principal shell by mean of a sleeve (Fig. 1). The stiffness of this sleeve is much lower than that of the thick cylindrical part of the head. However, in the extensometric analysis has been included in the model of the upper part of the sleeve. The substantial agreement of the results obtained from the different analyses confirms the validity of the simplification utilized in neglecting the presence of the sleeve.

Q R. L. ROCHE, France

If I have well understood, stress values are obtained on basis of an elastic analysis. I suppose it is intended to use these stress values for predicting the steam generator behaviour under loads. It seems to me very difficult to achieve that second task. For instance, elastic analysis cannot allow to share global stress between primary stress and secondary stress. Moreover, in your case, there are creep phenomena and a viscoelastic analysis could be required in order to get better information. My question is : "What is the way you intend to take in order to predict the structure behaviour from the results of these elastic stress analyses ?".

A S. CURIONI, Italy

This elastic analysis is intended to determine essentially the peak stresses at the geometrical discontinuity of the structures. A second task of this analysis is checking the numerical finite element analysis that in the future can be generalized to include without appreciable difficulty different behaviour laws of materials. Of course, the problem of utilizing these results in the field of temperature foreseen for the component is still open.
In any case in the philosophy of the actual ASME code the elastic analysis remains the first fundamental aspect.