3D Finite Element Stress Analysis of the Primary Side and Tube Sheet of a NPP Steam Generator

Dinggeng Xu, Weijuan Ye, Baisong Wang
Shanghai Nuclear Engineering Research and Design Institute
Shanghai, PRC

ABSTRACT

This report presents the results of a 3D finite element stress analysis of the primary side and the tube sheet of a NPP steam generator, which is subjected to internal pressure load and external load at the safe ends of the nozzle. The interacting effects of different components, maximum peaks stress and ligamemt stress of the tube sheet are obtained. The results of 3D are compared with those of axisymmetrical finite element analysis. At major locations the results have been evaluated in comparison with the stress limits of ASME code sectionⅢ.

1. 3D finite element stress of the primary side

1.1 Mathematical model

The divider of the water chamber is a symmetric plane in the chennel head. For simplification, the following assumption were made:

(1) The four nozzles are symmetrically located on the channel head.

(2) The tube sheet is freated as an equivalent orthotropic solid plate

(3) Only external symmetric nozzle loads are considered.

Two type of mathematical models for the primary side (i.e Channel head with the divider and channel head without the divider as shown in Figure 1 and Figure 2) had been considered. The premary and secondary pressures are equal to 17.5 Mpa and 7.7 Mpa respectively.

1.2 Numerical results

(1) Numerical resuts for the primary side with the divider

At section B, maximum stress appears at the inside of the head wall. In the are connecting the tube sheet and the channel head, the maximum principal stress σ_1 =367.5 Mpa. At section A, the principal stress decreases to σ_1 =180.3 Mpa due to the effect of the divider as shown in Figure 3. At the junction of the nozzle and the Channel head, there is also a peak value at the inside of the head wall as shown in Figure 4. Figure 5 shown the stress distribution along the periphery of the junction inside the head.

(2) Numerical results for the primary side without the divider

At the junction of the tube sheet and the channel head, maximum stress σ_1 =473.05 Mpa as shown by the dash line in Figure 3. At the junction of the nozzle and the channel head, maximum stress σ_1 =262.2 Mpa as shown by the dash line in Figure 5. Comparing with the divider model, the corresponding stresses decrease about 22% and 18% respectively.

(3) Comparison with numerical results of axisymmetric model

The numerical results of axisymmetric model for the tube sheet-channel head-the cylinder of the secondary side are shown by the dot tash line in Figure 3. At section B, maximum stress σ_1 =469Mpa, approximately equal to the correspoding value of 3D FEM model without the divider. At the junction of the nozzle and the channel head, for axisymmetric model, the maximum stress at the inner side of the head wall is σ_1 =137.3 Mpa, which is almost equal the corrosponding value at a point (point 4 in Figure 5 σ_1 =136.9 Mpa) in 3D FEM, where the stress is least affected by the divider and other nozzles.

2. 3D finite element stress analysis of the tube sheet

2.1 Mathematical model

In general, the ligament stress and the hole edge the stress of the tube sheet with square array could be **evaluated** by the method described in Referece (slot, 1972), while in this report, 3D FEM is used. For considering the effects of different compoments, the segment from section A-A to section B-B in Figure 2 had been considered. For simplification, the following assumption were made:

(1) The central lane of the tube sheet is also drilled with holes of equal spacing, then the number of holes is increased from 5960 to 6228.

(2) The holes are approxinately axisymmetrically distributed irrespective of the square arrary.

Then three models, namely, 1/64, 3/128, 1/32 (5.625°, 8.4375°, 11.25°) had been considered. The tube sheet is divider into 5 layers by tridmensional isoparametric elements. The cicular holes are replaced by square hole of equivalent area. For more accurate ligament stress and hole edge stress, a segment is taken from the 1/32 mode to be dividered into finner meches, i.e., with 10 layers of the tube sheet and octagon holes instead of cicular holes.

2.2 Numerical results

(1) As shown in Fig.6, the results of the 3/128 model approximately coincide with the corresponding values of the 1/32 model. The maximum stress is located at the upper surface of the tube sheet near the center. For demonstrating the reliability of the 1/32 model, 1/32 and 1/8 models with clomped boundary condition with square arrayed holes were compared. The two models give comparable results (Fig.7), For conservatism the results of the 1/32 model is increaded by 10% to give the calculated stresses of the tube sheet.

(2) The ratio of the hole edge stress and the mean ligament stress approximetely equal to 1.2. The numerical values of and for 3D FEM and axisymmetric FEM are shown in Table 1.

Table 1

Location r(mm)		Ligament stress σ_θ (Mpa)	hole edge stress σ_θ (Mpa)
100	axisymmetric	229.3	278.7
	integral	204.7	248.8
	1/32(11.25°)	141.9	170.3
	1/8(45°)	156.1	187.3
500	axisymmetric	200.7	245.7
	integral	189.0	231.4
	1/32(11.25°)	119.4	143.3
	1/8(45°)	131.3	157.6

3. Conclusion

(1) The stress intensities at major locations are lower than the stress limits specified by ASME code.

(2) The stress field of 3D FEM properly shows the effects of different component of the primary side.

(3) The numerical results of axisymmetric model are conservative.

(4) Tube sheet with densely popularted holes of square array could be treated as axisymmetric model.

(5) The divider affects the stress distribution of the primary side

significantly and decreases the stress level. The stress at the center of the junction of the divider and the tube sheet approches the yield point, and elastic-plastic analysis should be performed

REFERENCE

ASME Boiler and Pressure Vessel code, Section Ⅲ, Rules for construction of Nuclear Power Plant Components. 1987 Edition

Slot T. Stress Analysis of Think Perforated Plates. Technomic Publication, 1972.

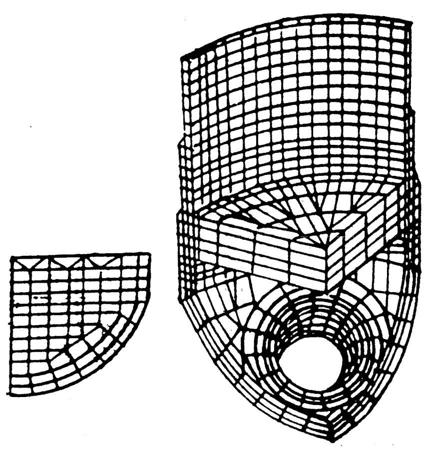

Fig.1 Element mesh of the primary side with divider.

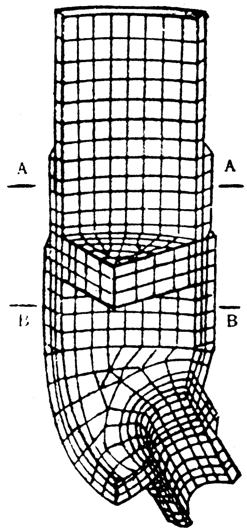

Fig.2 Element mesh of the primary side without divider.

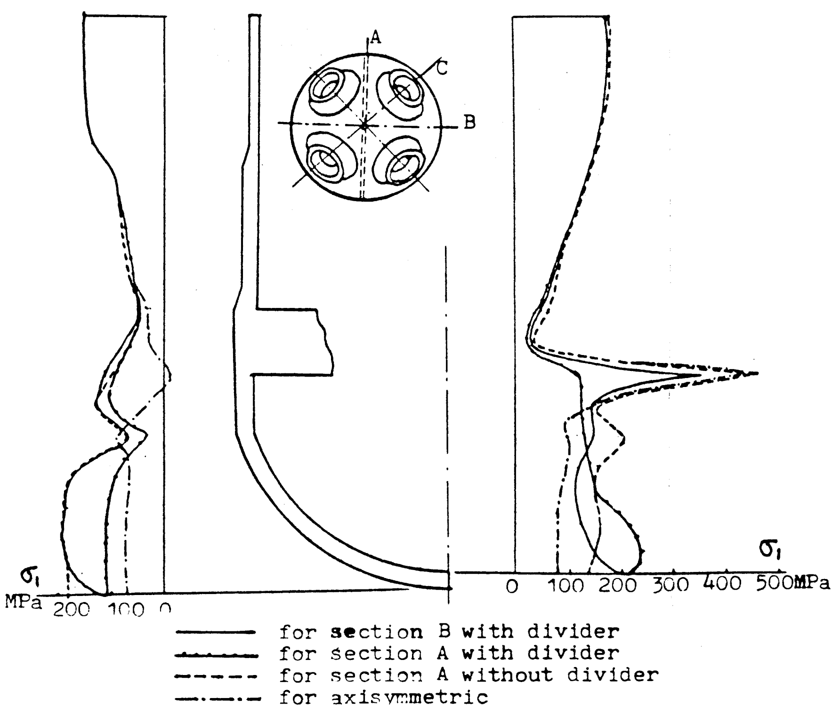

Fig.3 Stress distribution on A and B section.

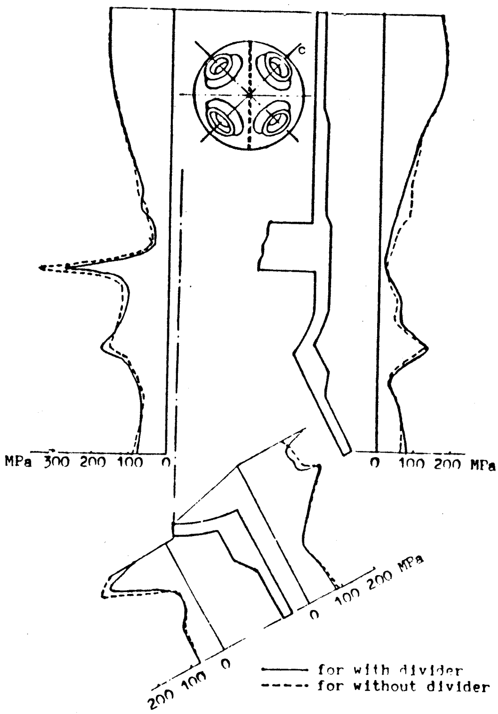

Fig.4 Stress distribution on C section.

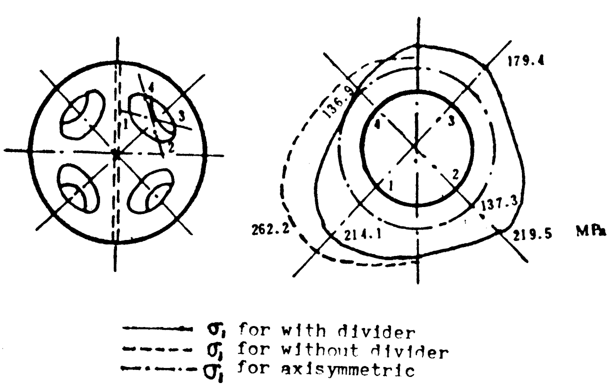

Fig.5 Stress distribution at the inside corner of the connecting region of the channel head and nozzle.

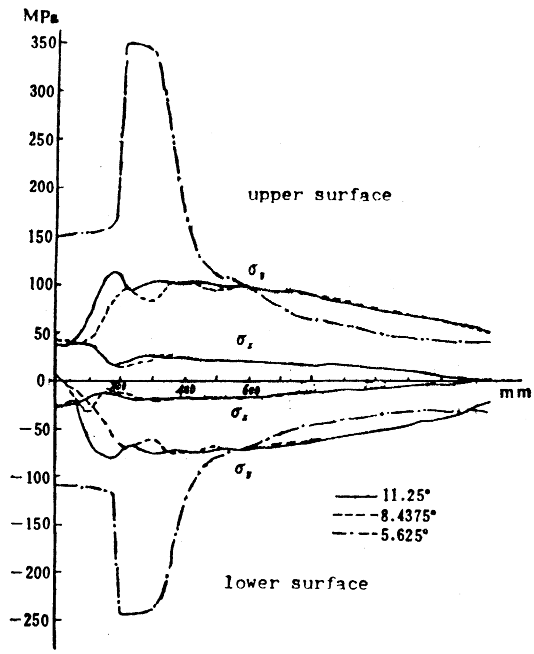

Fig.6 Distribution of radial stress for three type of models.

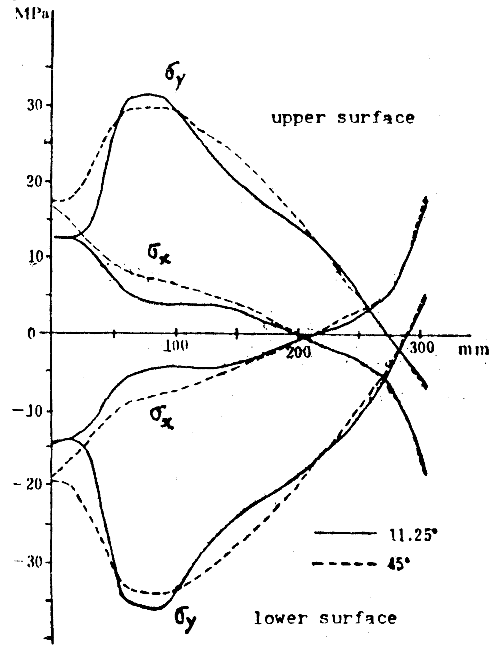

Fig.7 Stress distribution for 1/32 and 1/8 models.