STATIC STRESS ANALYSIS OF STEAM GENERATOR SHELL NOZZLE JUNCTION FOR LEAK BEFORE BREAK ANALYSIS

Mani N.[1]**, Thanigaiyarasu G.**[2]**, Chellapandi P.**[3]
[1]Research Scholar, Department of Mechanical Engineering, Anna University, Chennai, India
[2] Department of Mechanical Engineering, Anna University, Chennai, India,
[3]Safety Group, IGCAR, Kalpakkam, India
E-mail : [1]mani.mri@gmail.com

Abstract

The present power scenario in India is characterized by major contribution from fossil power stations. Under these circumstances, introduction of Fast Breeder Reactors (FBR) on commercial scale is possible only if their economic competitiveness is demonstrated viz-a-viz fossil power stations. For Sodium Fast Reactors, the Steam Generator is known to be a key component in terms of competitiveness and plant availability with safety implications through the sodium/water reaction. One of the critical locations in SG is the shell nozzle junction. This junction is subjected to an end bending moment and internal pressure. Since the shell nozzle junction is the critical location of SG a double-ended guillotine rupture will result in leakage of large quantity of sodium, which is not desirable. Hence safety requirements demand that LBB criteria with assumed initial flaw have to be demonstrated. For all these analysis, the basic requirement is to predict the state of stress precisely in the shell nozzle junction under various loading conditions. An efficient finite element modeling for shell nozzle junction has been presented in which shell elements are employed to idealize the whole region. The stress and deformation values are presented and compare with experimental study. Based on these analysis, the crack is initiated at the intersection of straight vertical shell and the cone ie. at the pullout region. These results are used for the analysis of leak before break concept.

Key words: Shell nozzle junction, steam generator, stress analysis, leak before break.

I. INTRODUCTION

The present power scenario in India is characterized by major contribution from fossil power stations. Under these circumstances introduction of Fast Breeder Reactors (FBRs) on commercial scale is possible only if their economic competitiveness is demonstrated viz-a viz fossil power stations. This requires a set of design features which lead to lower capital cost, reduction in construction time and improved capacity factor without compromising safety. In 500 MWe Prototype Fast Breeder Reactor (PFBR) the critical out-of-core components are main vessel, control plug, inner vessel, intermediate heat exchanges, steam generators(SG) and hot pipelines. Fig.1.1 shows the flow sheet of PFBR which indicates the critical components. Austenitic stainless steel type 316 LN is the main structural material for the out-of-core components and modified 9Cr-1Mo is chosen for steam generator.

Steam generator is an important component of Nuclear Reactor. Operating experience on steam generator has shown that this component plays an important role in influencing the plant capacity factor.

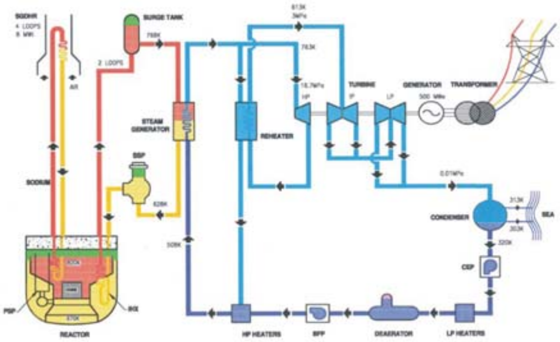

Fig. 1.1. PFBR flow sheet

The geometry of SG is a tall vertical once through shell and tube type exchanger supported at the middle. In PFBR, there are 8 steam generators, 4 in each of the two secondary loops. SG is once through integrated. Fig 1.2 shows SG configuration. The geometry of SG consists of two thick tube sheets connected by tubes and outer shell with bends near the bottom tube sheet. Sodium enters through a single inlet nozzle, flows upwards in the annular region and then flows down through the top inlet plenum where it is evenly distributed before entering the tube bundle. After flowing downwards on the outside of the tubes, sodium exits through the single outlet nozzle via the bottom outlet plenum. Feed water enters the tube side at the bottom, flows through the orifice incorporated for creating the desired pressure drop from SG stability consideration and flows upward in a counter flow direction to the down coming sodium.

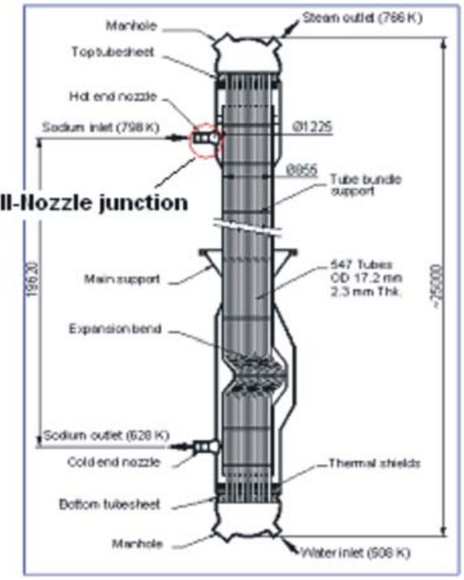

Fig. 1.2. Steam Generator

A. *Materials*

Modified 9Cr -1Mo (Gr.91) is the main Material of construction of the steam generator for components such as Tube, Tube Sheet, Shell, Thermal shield, Dished head and Flanges mainly due to its good high temperature mechanical properties ,resistance to chloride stress corrosion cracking and environment resulting from sodium water reactions and resistance to decarburization. Sodium inlet/outlet nozzles-trimetallic joints are made with (Gr91+Alloy 800+ 316LNSS). This material is used extensively as a structural material at elevated temperatures up to 873k in fossil fired power plants, petro chemical industries and as a material for tubing in the reheater portions and as thick section tube sheet material in the steam generators of FBRs. High thermal conductivity and a low thermal expansion coefficient coupled with enhanced resistance to stress corrosion cracking in steam water systems have favored the selection of this material for these applications. The material also possesses better monotonic tensile and creep strengths at elevated temperatures.

B. Objective

The objective of this project is, to evaluate the realistic state of stress in the junction so as to identify the critical location in the nozzle where notch is to be made for leak before analysis.

C. Details of the Shell Nozzle Junction

The sodium piping is made up of austenitic Stainless steel (SS316 LN) and the shell is made up of modified 9 Cr- 1 Mo materials. Since the thermal coefficient of expansion between the two materials is high, there is a trimetallic joint (Gr91+Alloy 800+ 316LNSS) between the piping and nozzle as shown in figure 1.3

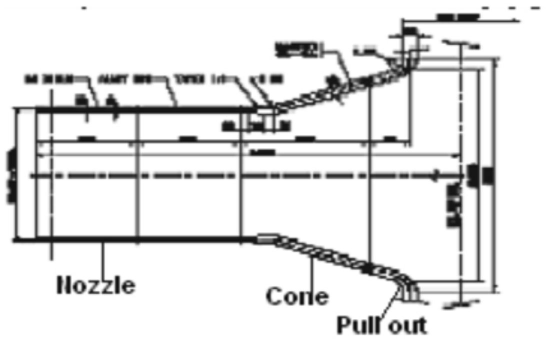

Fig. 1.3. Actual Shell Nozzle Junction

II. STRESS ANALYSIS OF SG SHELL-NOZZLE JUNCTION

A. *Introduction*

In this paper, scaled down model of actual shell nozzle junction is modeled and analyzed using ANSYS 10. Generally the pipe which carries sodium from surge tank to the SG nozzle causes bending moment on the shell nozzle junction. Stresses in the shell nozzle junction are analyzed to assess its structural integrity for the external BM which is caused due to vertical load at the end of the nozzle and for internal pressure.

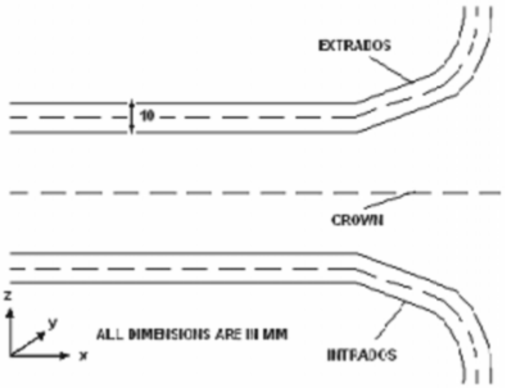

Fig. 1.4. Schematic representation of extrados, intrados, crown

B. *Geometrical details of SNJ*

Static FE analysis is carried out using ANSYS on half symmetric scaled down model of the shell nozzle junction. The internal diameter of the main tube is 1177 mm and that of the nozzle is 386 mm. The thickness of the main tube is 24 mm while that of the nozzle is 18 mm. The cylindrical shell has a pullout at 90° to its axis. This pullout will be welded to a conical reducer whose small end is attached to the connected piping. In order to reduce stress concentration, welding is avoided at the junction of cylindrical shell and the nozzle and the junction is made by pulling a portion of the shell in hydraulic press.

C. Material Property

The material properties taken for this analysis are given in table 1.

Table 1. Material property

Material type	Isotropic
Young's Modulus E	2.05E05 MPa
Poisson's ratio	0.3
Density	7850 kg/m^3

D. Boundary conditions and Loading

The flange plates at the top and bottom of the cylindrical shell are fixed.(all dof arrested). In the case of half symmetric models, appropriate boundary conditions were assumed to represent geometric and loading symmetries. Concentrated loads are applied at the end of the nozzle so as to develop the equivalent end moment. Pressure load is applied throughout the inner surface of the model, which includes the shell, the nozzle and the end plates. Three load cases were considered for an analysis of all the models. These are given in table.2. Fig.2.1 shows the boundary condition & loading details of shell nozzle junction.

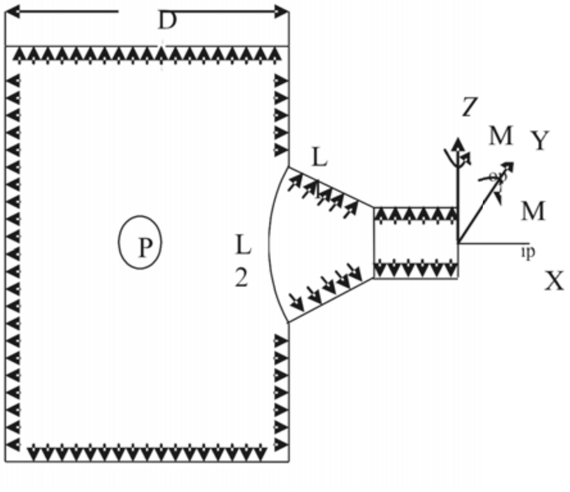

Fig. 2.1. Boundary conditions and loading

Table 2. Load case

Load case	Pressure MPa	Moment kN-m
1	0.5	0
2	0	45
3	0.5	45

E. Modeling and description of the analysis model

Scaled down model of the actual shell nozzle junction is created using ANSYS. First half shell model is created for analysis by taking advantage of symmetry. In the next step nozzle is created. The nozzles are assumed to be welded to the cylindrical shells with full penetration welds, however any effects of corner fillet welds are disregarded in the analysis. The element chosen for this analysis is SHELL 93.

F. Meshing

The size of the elements in the vicinity of the shell/nozzle was made sufficiently small(less than .25Ö(RT) where R,T are mean radius, thickness of the shell) to ensure that accurate results are obtained in this area. Away from the junction, the element size was gradually increased to account for their stiffness only. The meshed model of the pullout region is shown in fig.2.2 and shell Nozzle is shown in the Fig 2.3

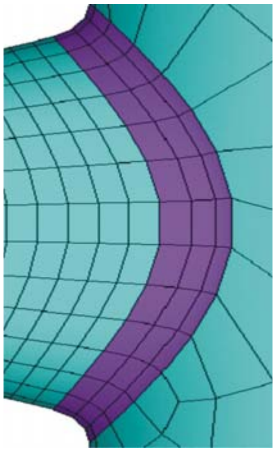

Fig. 2.2. Meshed model of pullout region

G. Finite Element Analysis

To analyze tubular joints using FEM, intersecting cylindrical shells are idealized by employing finite elements, which can approximate the in-plane behavior of the tubular members. Shell elements have been widely used to model tubular joints without cracks, but welds cannot be modeled properly using shell elements. Finite element analysis of shell nozzle junction specimen is carried using ANSYS software for healthy (uncracked) specimen. Linear static analysis of the shell nozzle junction has been carried out using a 2D model with eight-noded shell elements (SHELL 93) having six degrees of freedom at each node. Finite element modeling of the full model and half symmetric model has been made. The full specimen modeling was done mainly to establish the accuracy of the results with the half symmetry model. For half symmetry model, a coarse and a refined model were developed. Symmetry conditions were fully utilized in FE models to reduce the computing time. For in-plane bending cases, the nodes at the end of the SNJ were constrained through multi point constraint (MPC) option with in ANSYS. The flange plates at the top and bottom of cylindrical shell are fixed (all the dof arrested). In the case of half symmetry model appropriate boundary conditions were assumed to represent geometric and loading symmetries. Concentrated vertical load is applied at the nozzle end. Pressure load is applied throughout the inner surface of the model, which includes the shell, the nozzle and the end plates. Three load cases were considered for the analysis (given in the table).Convergence of the results was studied from these analyses. The refined half symmetric model was then selected based on convergence and analyzed for all the load cases.

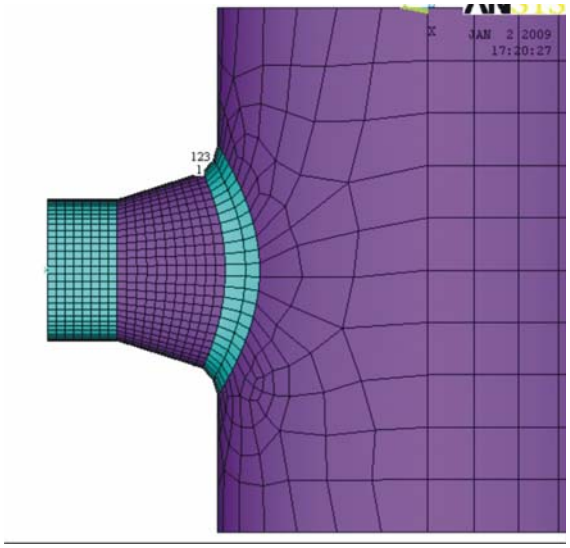

Fig. 2.3. Meshed model of Shell nozzle junction

III. RESULTS AND DISCUSSIONS

A. Introduction

In the Finite element analysis half symmetry of the shell–nozzle junction is modeled, with all applied boundary conditions and meshing discussed in the earlier chapters.

FEA was made for the following investigations:

i) Location of the peak stress due to bending load at the free end of the nozzle and internal pressure

ii) The variation of stress along two regions (extrados and intrados) and through the thickness of the shell

B. Von mises stress distribution

The FE model with node numbers is shown in the fig.2.3. The stress contours with von mises stress details for load cases 2 in fig.3.3. The deformation plots obtained for the load case 1 is shown in fig.3.1. Fig.3.2 shows displacement shape when only the moment is applied at the end of the nozzle. Fig.3.1 shows the displacement shape when only the pressure is applied all the inner surfaces of the shell nozzle junction. The maximum on mises stress is observed in the pullout region. There

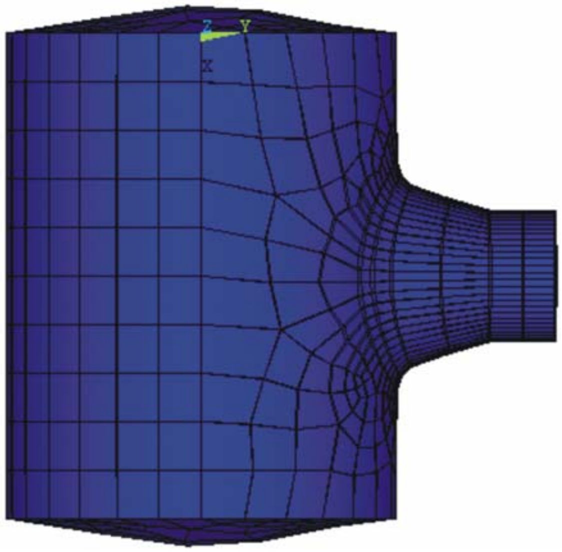

Fig. 3.1. Displacement shape for load case -1

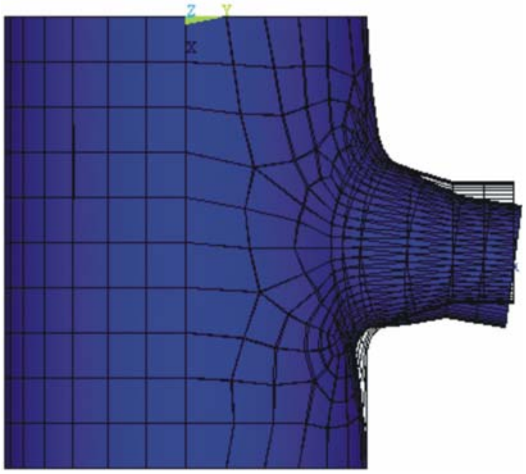

Fig. 3.2. Displacement shape for load case 2

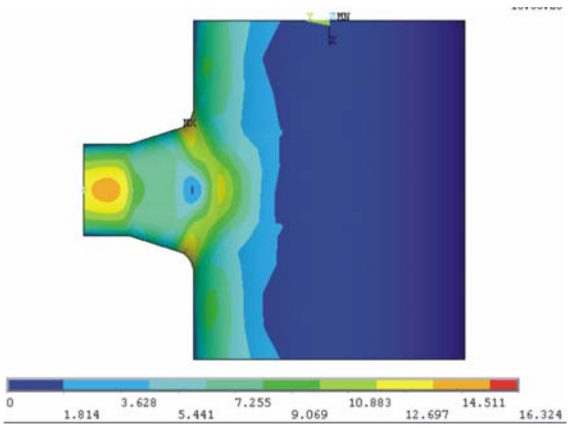

Fig. 3.3. Von-Mises stress plot for load case 2

is an equal and an opposite stress at the symmetry location along the extrados. It is also observed that there is another lesser peak stress at the junction of cylinder and cone portion.

C. Discussion

The shell nozzle junction model is analyzed with three load cases as given in table 2.2. Two locations are considered like extrados and intrados and for each location three layer stresses are calculated. The von mises stress values are plotted in the form of graph by taking von mises stress in y-axis and the distance from nozzle end to pullout in x-axis. In the following section will discuss in detail the stress values for all the load cases

In this load case 1 pressure is applied throughout the inner surface of the model. From the figure 3.4, it is observed that all the stresses are in tensile nature and is increasing linearly from starting to end of the nozzle. It starts decreasing in the cone potion and it is increasing in the pullout region.

In load case 2, Moment is applied at the end of the nozzle. Figure 3.5 shows the von mises stress values at each node along extrados for top, middle and bottom layers due to the application of end moment. It is observed that in the cylindrical portion of the nozzle it is following linear variation according to simple beam theory. In the conical zone, the variation is no more linear as the diameter is increasing and hence there is reduction in stresses. In the pullout zone, the shell theory predominates. . Since it is the bottom side of shell-nozzle junction the stresses are compressive in nature.

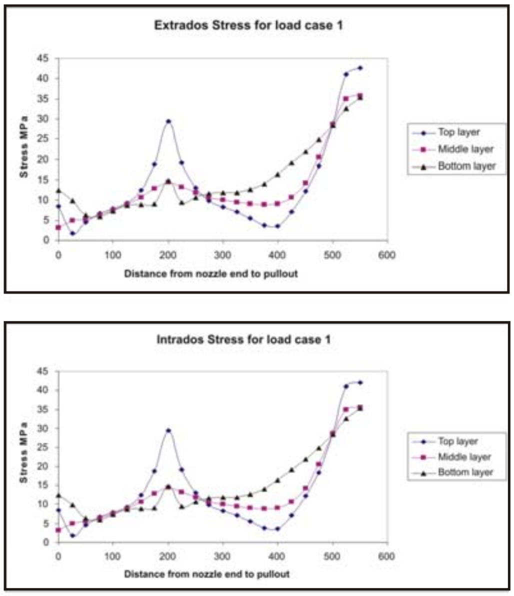

Fig. 3.4. Stress plot along the region from nozzle end to pullout for load case-1

In load case3, pressure is applied throughout the inner surface of the model and the moment is applied at the end of the nozzle. From fig.3.6, it is observed that in the cylindrical portion of nozzle it is following linear variation and at the junction of nozzle and cone intrados gives maximum stress values than extrados. In the conical zone the stress variation is no more linear and increasing towards the pullout. The maximum stress is obtained in the pullout region.

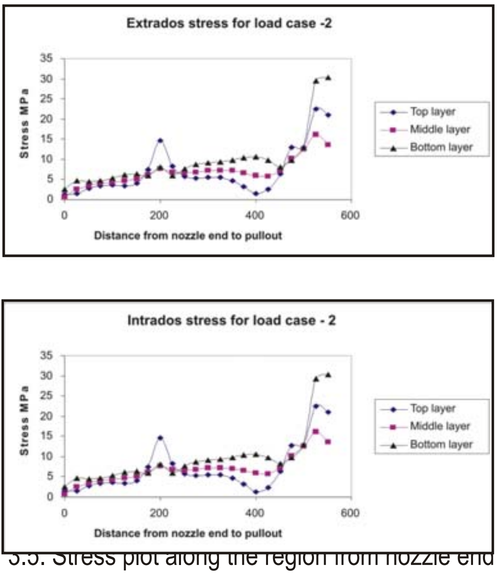

Fig. 3.5. Stress plot along the region from nozzle end to pullout for load case-2

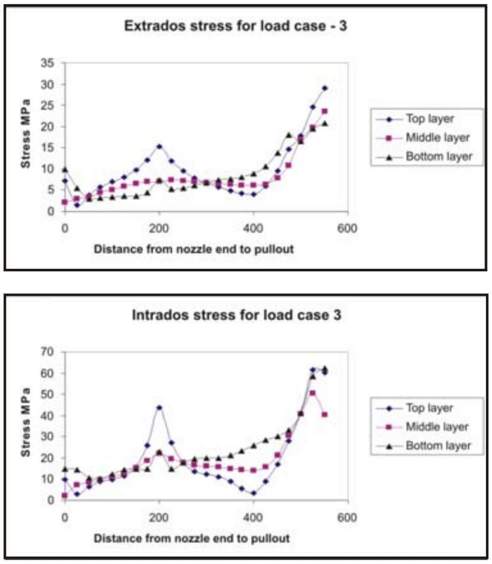

Fig. 3.6. Stress plot along the region from nozzle end to pullout for load case -3

IV. CONCLUSION

The maximum stress values obtained from the analysis for all the load cases. From the results of analysis, it can be observed that the maximum stress occurs at the junction of pull out region and the conical portion. High stress concentration is developed at this location due to abrupt change in the geometry and the consequent change in stress flow. In this location, the crack is formed and can do further analysis for leak before break concept.

REFERENCES

[1] D.S.Ramachandramurthy et.al, "Investigations on shell-nozzle junction of steam generator for LBB justification", Journal of Fatigue Fract Engng Mater struct., 2007, 30, pg.1203-1213.

[2] Ming-De Xue et.al, "Theoretical stress analysis of Intersecting Cylindrical Shells subjected to External forces on nozzle", Journal of Pressure vessel Technology, 2006, 128, pg.71-83.

[3] A.R.C.Murthy et al. "An efficient FE modeling strategy for Fracture Analysis of Tubular Joints", IE (I) Journal – CV, 2004, 85, pg.17-25.

[4] Bijlaard, P.O., "Stresses from local loadings in pressure vessels", Trans , ASME , vol .77, 1955 pp. 805-816.

[5] Zienkiewicz. O.C ., 1989, "The Finite element Method", McGraw –Hill, NewYork

[6] Mershon , J. L., Mokhtharian, K., Ranjan, G. V., and Rodabaugh, E. C., "Local stresses in cylindrical shells Due to External Loadings on Nozzles-Supplement to WRC Bulletin No .107, " WRC Bulletin No . 297, Aug.1984.

[7] Timoshenko, 1996, "Theory Of Plates and Shells" Tata Mc Graw –Hill

Mr. N. Mani is a Research Scholar at the Department of Mechanical Engineering, Anna University, Chennai and his field of research is "Fracture Mechanics". He has over nine years of teaching experience.