STRUCTURAL ANALYSIS OF THE PRESSURE RETAINING PARTS OF A NUCLEAR POWER PLANT MECHANISM

H. Sanzi, ENACE S.A., L.N. Alem 712, 1001 Buenos Aires, Argentina

1. INTRODUCTION

The maximum stresses of the retaining parts of Nuclear Power Plant component were calculated for several loading condition according to the regulations of ASME Code, Section III, Subsection NB [1]. These are detected during normal operation and testing condition. In addition to the stress analysis, a fatigue evaluation and fracture mechanic with a circumferential crack were performed to show the integrity of the component. This component was idealized by means of finite element techniques.

2. DESCRIPTION AND DETAIL OF COMPONENT DESIGN

The drawing in Figure 1 shows a complete longitudinal section of the pressure retaining component. The material designation of each part component corresponds to austenitic material (flange, looseflange, tube and nozzle) and ferritic material (tubes, bolt and nut). The material gaskets depend on normal (austenitic) or testing (spiral-wound) conditions. The allowable stress and the limits of stress intensity are determined according to ASME III, NB 2000 for normal and testing conditions.

3. DESCRIPTION OF FINITE ELEMENT MODELS

The mathematical model was prepared to be used by Finite Element Program. The structural unit comprises the bolted joint, the pressure retaining tubes, the rings, flanges and looseflanges. The bolted joints have been modeled with 3-D isoparametric solid element. Only a sector with an angle of 18° was considered, being defined with symmetry condition. The pressure retaining tubes have been modeled with an axisymmetric 2-D isoparametric solid element.
We want to point out the following aspects:

* The bolt and nut were modeled by beam element with equivalent properties and the bolt-nut connection by the same element with infinitely rigid properties, but the results obtained with the last element do not represent the real behaviour, they are only used to accomodate load from the bolt and their actual stress states were calculated by other rules.

* The contact surfaces in the bolted joint contact were modeled with three-dimensional interface (GAP) and couple nodes in radial and/or axial directions, figure 1. A friction value of 0.2 and 0.3 was used.

* The length of each tube was defined according to the following expression:

\[ l \geq 2.5 \sqrt{\frac{R}{t}} \]

\[ t \] \[ m \]

\[ R \] and \( t \) are the mean radius and thickness respectively.

* The moment distribution force was represented by an equivalent axial load applied in the tube-end with a senoidal distribution in the circumferential direction (conservative assumption).

* The tangential stresses in the bolt due to tightening torque (prestress condition) were calculated separately by conventional rules.

* The gaskets used in normal (austenitic material) and testing conditions (spiral-wound material) were idealized by 3D-isoparametric elements with elastic properties.

* The initial clamping load (assembly preload) was calculated according to VDI 2230 and was represented in the model by initial strain. This strain was applied in the bolt and its value was corrected because of the bolted joint stiffness.

* A dense mesh with 20 nodes 3D-isoparametric elements was used in the vicinity of the circumferential crack.

4. **LOADINGS**

The load cases are classified into load case types. Each load case consists of several loadings depending on the load case type, as it is summarized in the following table:
<table>
<thead>
<tr>
<th>LOAD CASE TYPE</th>
<th>LOAD CASES</th>
<th>LOADINGS</th>
</tr>
</thead>
<tbody>
<tr>
<td>NORMAL CONDITION</td>
<td>Thermal load: Tube temperature distribution in radial and axial directions at 100% power.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Pressure load: 115 bar</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bolt preload: The bolt stress during tightening VDI 2230.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Loadings due to operational condition for all pressure retaining parts.</td>
<td></td>
</tr>
<tr>
<td>TESTING CONDITION</td>
<td>Test pressure load: 175.5 bar.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Bolt preload: VDI 2230.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Moment distribution load: Weight effect.</td>
<td></td>
</tr>
</tbody>
</table>

5. **STRESS ANALYSIS RESULTS**

The superposition was made according to loadings defined in Chapter 3. The maximum stresses occurred in Normal Operation and are shown in Figure 3.

6. **FATIGUE EVALUATION**

An analysis for cyclic service was required according to the criteria of ASME CODE. A representative pressure and temperature fluctuation of the complete transients of this component was calculated. A stress concentration factor was used in the bolt evaluation (shaft and thread part) [2].

7. **FRACTURE MECHANIC ANALYSIS**

The calculation of the stress intensity factor K is an essential step in the fracture mechanics analysis of cracks in engineering components. The K factor was obtained from the results of finite element analysis by correlating the predicted stresses adjacent to the crack tip [3,4].

The stress component in the crack tip stresses field is:

\[
\sigma = \frac{K}{\sqrt{2\pi r}} \cos \frac{\theta}{2} \left[ 1 - \frac{\theta}{2} \frac{\sin \frac{\theta}{2}}{\sin \frac{3\theta}{2}} \right](1)
\]
The stresses result close to the crack tip are inaccurate owing to the inability of the elements to represent the stress singularity. However the equation (1) may be used to plot the estimated value of $K$ as a function of $r$ by extrapolating the curve at $r = 0$. Then, we employed the Finite Element Method to estimate the $K$ factor for a length crack corresponding an angle of $12^\circ$ in circumferential direction and a width-thickness ratio of 0.3 (see figure 1). This method performs a good approximation of the $K$.

8. CONCLUSIONS

The present work describes the main design techniques used to demonstrate the structural integrity of a component under service and testing conditions. The work was carried out by using Finite Element Method and a stress, fatigue and fracture mechanic analysis were calculated. In all cases, the limits of Stress Intensities were found to be smaller than the allowables and the results for cyclic service verified the fatigue evaluation. The stresses for prestress load with a friction value of 0.3 have been higher (with respect to 0.2) in the bolted joint (about 20%). The $K$ factor calculated will forms the basis for subsequent development of fracture control plans, i.e., these results will provide an appropriate analysis to prevent fractures in the component.

REFERENCES

2. Stress Concentration Factors.
   R.E.Peterson, Ed. John Willey and Sons.
FIGURE 1: GEOMETRY DETAIL OF COMPONENT.

FIGURE 2:
FINITE ELEMENT MODELS: TOP AND PRESSURE RETAINING TUBE.
FIGURE 2 (cont.): FINITE ELEMENT MODELS: UPPER AND LOWER FLANGE.

FIGURE 3: MAXIMUM STRESS RESULT (STRESS INTENSITY).