

STRUCTURAL ANALYSIS OF ITEMS IN THE PRESSURE BOUNDARY OF THE IEA-R1 RESEARCH REACTOR HEAT EXCHANGER

Gerson Fainer¹, Carlos Alexandre de Jesus Miranda¹ and Miguel Mattar Neto¹

¹ Instituto de Pesquisas Energéticas e Nucleares, IPEN – CNEN/SP
Av. Professor Lineu Prestes 2242
05508-000 São Paulo, SP
gfainer@ipen.br
cmiranda@ipen.br
mmattar@ipen.br

ABSTRACT

In the last years some actions have been taken in order to increase the power output of the IEA-R1 research reactor. Among these actions, we have the replacement of the primary heat exchanger. The thermal-hydraulic design of the new one was performed by IPEN and the mechanical design was developed by the manufacturer accordingly the TEMA requirements. It was verified the possibility to have some excessive vibration in the tubes between the tube-sheet and the first support plate near the inlet nozzle. So, it was necessary to adopt a non usual solution to avoid this problem. The adopted solution was to put the inlet nozzle in a structure which includes a cylinder and two radial rings welded to the cask. When doing so, there is an offset between the cask holes and the inlet nozzle. This solution was not a TEMA standardized one. This new solution was analyzed “by formulas” by the manufacturer. As this is an IEA-R1 research reactor important equipment, an additional structural analysis was performed at IPEN using the finite element method to analyze the modified items of the heat exchanger pressure boundary. The numerical results assessment shows the adequacy of the heat exchanger design and its structural sizing.

1. INTRODUCTION

IEA-R1 is a pool type research reactor, moderated and cooled by light water, and utilizing graphite as reflector. The construction is a Babcock & Wilcox design and the first criticality was achieved in September 16, 1957. After initial tests, the reactor started operating at 2 MW. Due to the growth in radioisotope demand, in 1997 after necessary modifications and upgrading process, the power is being increased to 5 MW.

More recently it was noticed that it was necessary to replace the primary cooling system heat exchanger in order to achieve the goal of an operation power of 5 MW. The thermal-hydraulic design of the new heat exchanger was performed by IPEN and the mechanical design was developed by the manufacturer accordingly the TEMA requirements [1].

It was verified the possibility to have some excessive vibration in the tubes between the tube-sheet and the first support plate near the inlet nozzle. So, it was necessary to make a structural modification to avoid this problem. The adopted solution was to put the inlet nozzle into a structure which includes a cylinder and two radial rings welded to the cask. When doing so, there is an offset between the cask holes and the inlet nozzle. Fig. 1 and 2 shows some details of these heat exchanger items.

This solution was not a TEMA [1] standardized one. This new solution was analyzed “by formulas” by the manufacturer. As this is an IEA-R1 research reactor important equipment,

an additional structural analysis was performed at IPEN using the finite element method to analyze the modified items of the heat exchanger pressure boundary and to assess structural behavior of the heat exchanger modified items. Details of the structural modeling and of the numerical simulation as well the results and assessment are showed and discussed.

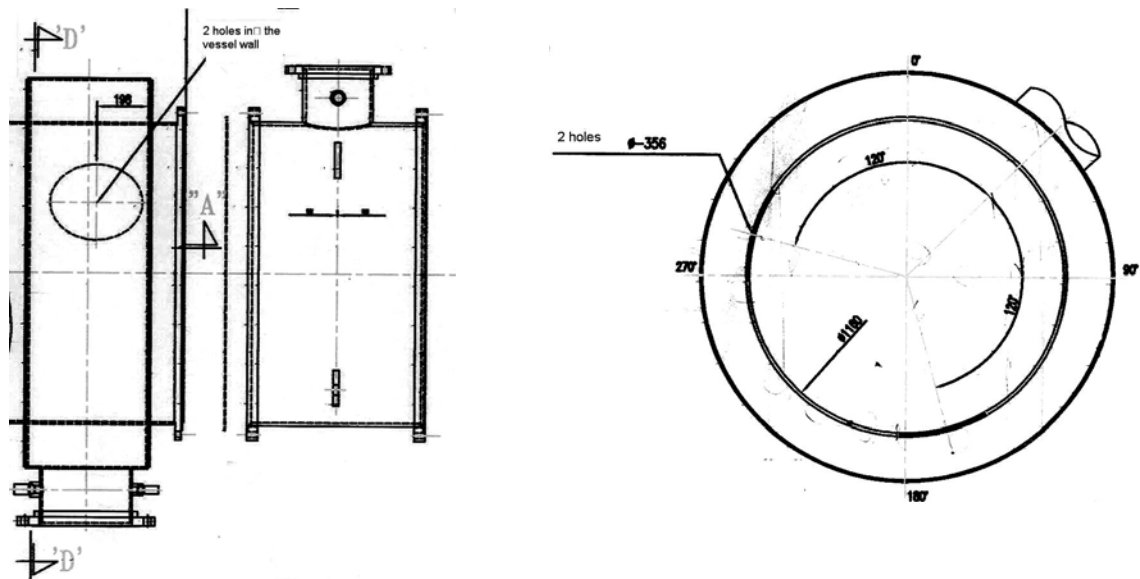


Figure 1: Heat exchanger – region under evaluation

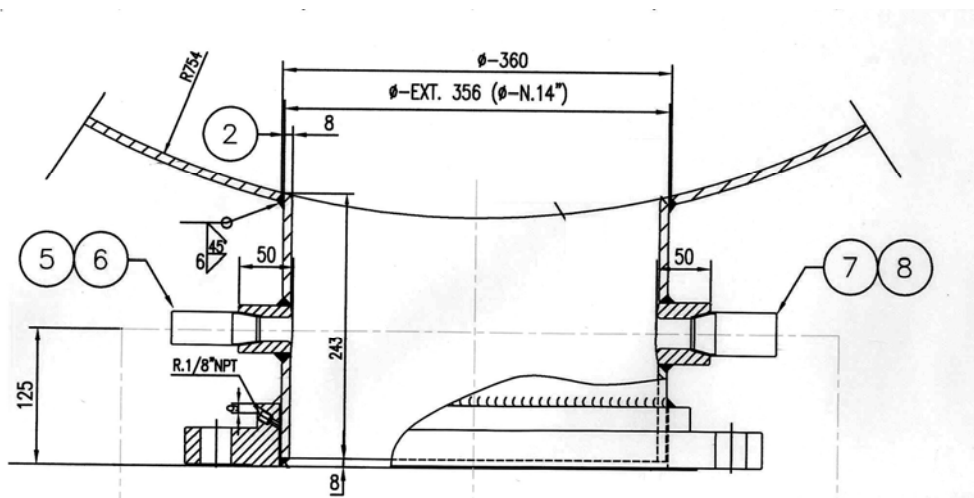


Figure 2: Heat exchanger – D-D view according to Fig. 1

2. FINITE ELEMENT MODEL AND ANALYSES

Based on the geometry showed in Fig. 1 and 2, a finite element model was built using three dimensional quadratic finite elements with three degrees of freedom per node (SOLID95 of the ANSYS [2] element library) to capture the welding connections in a more detailed and precise way. Notice that the small nozzles 5, 6, 7 and 8 and the flanged connection (see Fig. 2) are not included in the finite element model. Some views and details of the model and of

the mesh are showed in Fig. 3. It is important to notice that only a half of the structure is modeled due to the geometrical symmetry with respect to the plane YZ (see the coordinates system in Fig. 3(a)). The fillet welds were also modeled and adequate bounded contacts were defined to reproduce the actual geometry in their positions.

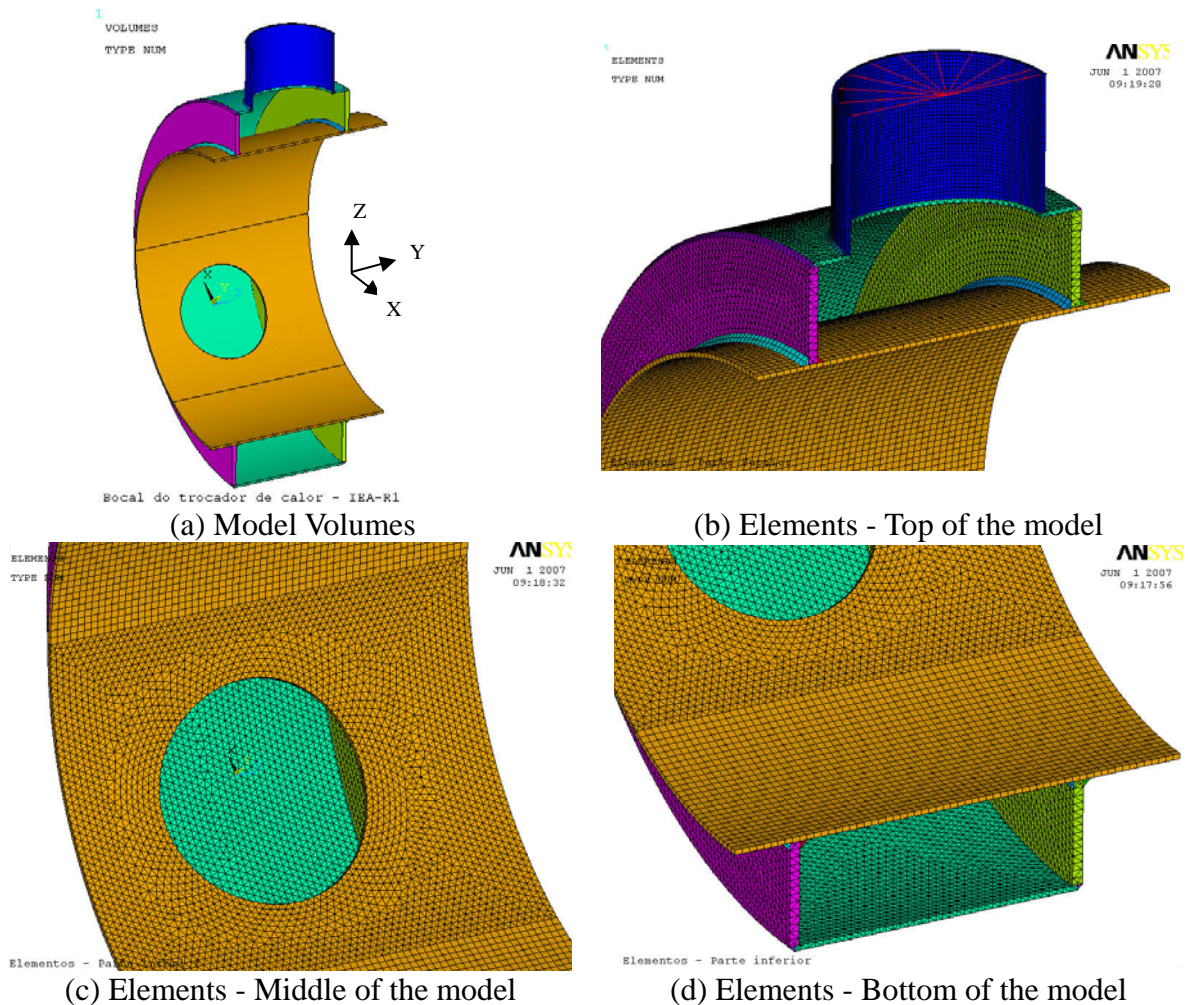


Figure 3: Finite element model

The heat exchanger material is the steel SA 240 TP304 and its mechanical properties, the Young's modulus - 200.000 N/mm^2 , the Poisson's ratio - 0,3, and the basic allowable limit S according to the ASME Code [3] - 130 N/mm^2 .

The applied loads are the heat exchanger design internal pressure and the nozzle piping loads (three forces and three moments). Three load steps were defined and the correspondent displacement constraints were applied to the model considering the geometrical symmetry and the symmetry or anti-symmetry of the loads according to the XZ plane (see Fig. 3(a)).

The structural analysis was performed in three load steps: 1st - internal pressure with symmetry constraints; 2nd - the symmetric nozzle piping loads with symmetry constraints; 3rd - the anti-symmetric nozzle piping loads with anti-symmetry constraints.

3. RESULTS AND DISCUSSION

The numerical simulations were performed with the program ANSYS [2]. The materials were considered elastic, linear and isotropic therefore the static and linear analysis option was used.

The obtained Tresca stress intensities (the biggest difference between the principal stresses) for the three load steps are showed in Fig. 4.

It is important to notice that the ASME Code [3] uses the called “Design by Analysis” where, under the design conditions, the allowable limit for generalized membrane primary stresses (P_m) is 1,0 S, for localized membrane primary stresses (P_l) and for primary membrane plus bending stresses ($P_l \pm P_b$) it is 1,5 S. If the maximum stress intensity is smaller than the allowable limit 1,0 S, that is the most stringent check, it is not necessary to perform a detailed stress analysis. If it is not the case, it is necessary to linearize and to separate the stresses in the ASME Code [3] stress categories, i.e., generalized primary membrane stresses (P_m), localized primary membrane stresses (P_l) and bending primary stresses (P_b) to do the stress analysis.

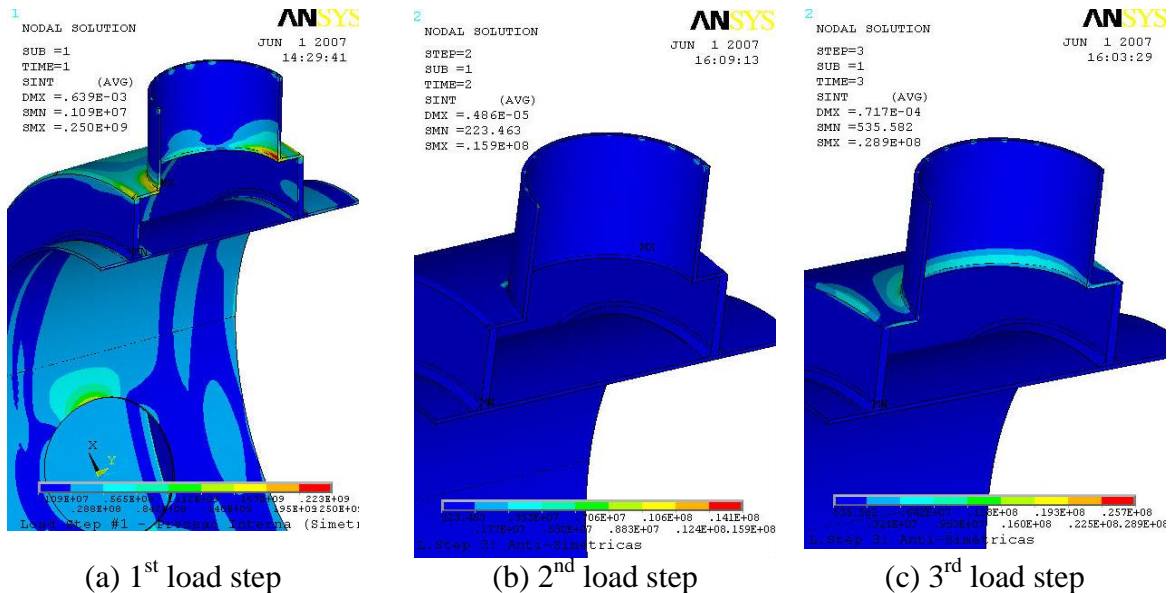


Figure 4: Tresca stress intensities

From Fig. 4, the maximum stress intensity is 250 N/mm², occurring in the vessel-nozzle transition weld. This stress is greater than $S = 130,0$ N/mm² and a more detailed stress analysis, based on the stresses linearization and separation, must be conducted. It also can be noticed that the greatest stress intensity values in the vessel-nozzle transition weld are produced by the internal pressure (1st load step) and the other loads produce almost no stress in this position. So, the next verification will be performed for the 1st load step, only.

The first step in the detailed stress analysis is the evaluation of the membrane and bending stresses in the critical positions. As indicated above, the critical point is the vessel-nozzle transition weld. Using the so-called “linearization by a line” procedure [4], available in

ANSYS [2] post-processing, the obtained stress linearization and separation is showed for four critical lines in this position and showed in Fig. 5. The critical lines are showed in Fig. 6.

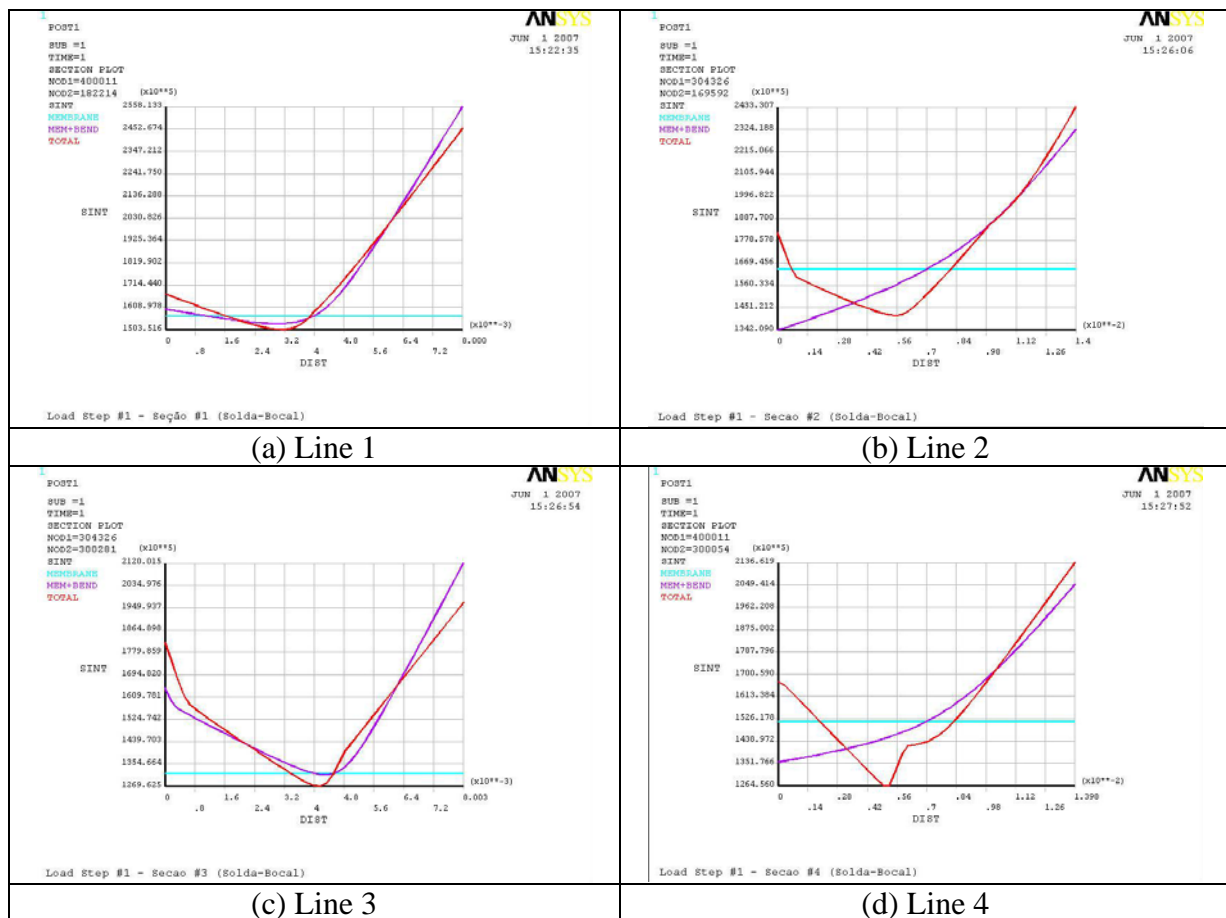


Figure 5: Stress linearization

Also, the ASME Code [3] indicates that in a cylindrical or spherical shell loaded by internal pressure, in positions remote from discontinuities, the membrane stresses shall be classified as P_m . In the other hand, in positions near a structural discontinuity, the membrane stresses shall be classified as P_1 . The bending stresses are practically negligible in positions remote from discontinuities and it shall be classified as secondary stresses (Q) near a structural discontinuity. In any shell, near a nozzle and loaded by its piping forces and moments, membrane stresses shall be classified as P_m and the bending ones as Q . It is important to notice that, according to ASME Code [3], Q are not related to the equilibrium of the internal and external forces and moments (the applied loads) it is developed by the constraint of adjacent parts or by self constraint of a structure. So, only the primary stresses must be verified under the design conditions.

From the results showed in Fig. 5 and following the ASME Code [3], the maximum membrane stress is $167,0 \text{ N/mm}^2$ (see Fig. 5 (b) Line 2) is classified as P_1 and it is smaller than the applied allowable limit $1,5 S = 195,0 \text{ N/mm}^2$.

4. CONCLUSIONS

The paper presents the numerical simulation using a three dimensional finite element model of items of the new heat exchanger of the IEA-R1 research reactor that have not a TEMA [1] standardized configuration.

The finite element model was developed considering the details of the welding connections. Only a half of the structure was modeled due to the geometrical symmetry. The structural analysis was performed in three load steps, considering the geometrical symmetry of the structure and the symmetry and anti-symmetry of the loads.

The obtained stresses were linearized and separated according to ASME Code [3] recommendations using ANSYS [2] and its post-processing procedure [4] to perform the structural assessment. The stress check shows that the applied stresses are below applicable ASME Code [3] allowable limits so the design of the IEA-R1 heat exchanger is qualified for its design load conditions.

REFERENCES

1. TEMA, Standards of the Tubular Exchanger Manufacturers Association, Eighth Edition, Tubular Exchanger Manufacturers Association, Inc (TEMA), Tarrytown, NY, USA, 1999
2. ANSYS, Inc., ANSYS Multiphysics 11.0 Release, Canonsburg, PA, USA, 2007.
3. ASME, The ASME Boiler and Pressure Vessel Code, Section VIII, Division 2, Pressure Vessels – Alternative Rules, The American Society Of Mechanical Engineers (ASME), New York, NY, USA, 2004.
4. Hechmer, J. L. & Hollinger, G. L., 3D Stress Criteria Guidelines for Application, Welding Research Council Bulletin # 429, Feb. 1998.