

STRESS ANALYSIS ON VESSEL/NOZZLE INTERSECTIONS WITH/WITHOUT PAD REINFORCEMENT FOR CYLINDRICAL PRESSURE VESSELS

Jorge R. Miranda, ricardomiranda@petrobras.com.br

Helder de Souza Werneck, heldersw@petrobras.com.br

PETROBRAS – UN-REGAP – Rod. Fernão Dias BR 381, km 427, Betim, MG, 323530-000

Carlos A. Cimini Jr., cimini@ufmg.br

Universidade Federal de Minas Gerais – Av. Antônio Carlos, 6627, Belo Horizonte, MG, 31270-901

Abstract. *The purpose of this paper is to evaluate the stress fields on the vessel/nozzle intersection of cylindrical pressure vessels using more realistic finite element models. ANSYS Workbench finite element tool was used as platform to develop three dimensional models with 20-node high order solid elements. Three models were developed, concerning the vessel/nozzle intersection: (i) unreinforced vessel/nozzle intersection, (ii) bonded pad reinforced vessel/nozzle intersection (pad is integrally welded on the vessel/nozzle intersection), and (iii) partially welded pad reinforced vessel/nozzle intersection (borders of the pad are considered bonded simulating welding lines and friction contact hypothesis is assumed in the pad/vessel contact surface). Pads were designed according to ASME Code Criteria – Area Replacing Method. Linear (models i and ii) and nonlinear elastic analysis (model iii) were performed. Results obtained correlated well with experimental results obtained from the literature.*

Keywords: *Finite element analysis, pressure vessel nozzle, cylindrical intersection, pad reinforcement, contact*

1. INTRODUCTION

Cylindrical pressure vessels are widely used in many engineering applications. Power and nuclear plants, and food and petrochemical industries are some examples of applications of such equipments. Particularly in the petrochemical industry, there are many cylindrical pressure vessels, such as distillation towers, heating exchangers and pipelines. Nozzles and shunts are normally used in pressure vessels with the purpose of fluids transfer and communication between vessels. Integral or welded, the nozzles are attached to the vessels, forming cylindrical intersections (Widera and Xue, 2004). Depending on the position of the axis, these nozzles can be radial, lateral or hillside.

Pressure vessel cylindrical intersections are defined as the transition between the nozzles and the shells, and constitute a critical area, normally the weakest part of the vessel. Due to geometric discontinuities, this region presents high stress concentration levels and it is where failure is more likely to occur. Therefore, this region is of primary interest for structural integrity and fracture mechanics analyses. In addition, due to difficulties in manufacture, these intersections are more suitable to fabrication imperfections, such as lack of weld deposition and/or weld penetration, which consequently cause crack propagation (Xue *et al.*, 2003).

For pressure vessels safety operation and long life warranty, equipment inspectors constantly monitor crack propagation and growing of internal failures. Through non-destructive techniques (X-ray, ultrasound scans, etc.), internal defects and failures of fabrication are regularly measured and controlled in order to avoid reaching a critical size. As failures frequently occur in the cylindrical intersections, the stress field in these regions must be thoroughly known so that a criterion for maximum defect size can be derived and implemented (Werneck and Tinoco, 2000).

Pressure Vessels Codes and Standards, such as the ASME Boiler & Pressure Vessel Code (ASME, 2006) or the BS 5500 Unfired Fusion Welded Pressure Vessels (BS 5500, 2004), among others, establish rules and methods for analyzing the pressure vessels and their components, and categorize internal stresses on them. Using analytical methods to perform stress analysis, one can calculate maximal and minimum stresses using formulas e graphics. Other way to evaluate the stress fields is to perform numerical analysis using computational tools, in particular, the finite element method (FEM).

Many studies and research work has been conducted on pressure vessel stress analysis using FEM. Innumerable papers have been published showing comparative results among experimental e analytical solutions, including the evaluation of Stress Concentration Factors (SCF) for cylindrical intersections. However, most of them considered intersections to be not pad reinforced (Wang *et al.*, 2006; Xue *et al.*, 2006; Liu *et al.*, 2004; Chen and Schnobrich, 1980; Dekker and Stikvoort, 1997; Natarajan *et al.*, 1987). And for those which considered pad reinforced intersections, the pad is assumed to be an integral part of the model, which do not represent the real condition (Zhang *et al.*, 2006; Taagepera and McKay, 2006). The reason for that is simplicity of modeling and less computer time for simulation.

In order to consider a more realistic model of pad reinforced cylindrical intersections, it is necessary to model the welding line between the reinforcing pad and the nozzle/vessel walls and to assume contact between the external surface of the vessel and the internal surface of the reinforcing pad (Fig. 1). The problem becomes clearly of nonlinear nature and, consequently, the simulation is more complex and more time consuming. Thus, the purpose of this work is

to develop such more realistic finite element model and to perform the stress analysis in order to predict the stress fields on the cylindrical intersections (nozzles) with pad reinforcement. Results from the Finite Element Analysis (FEA) could then be used as guidelines for the PETROBRAS/REGAP Inspection Service.

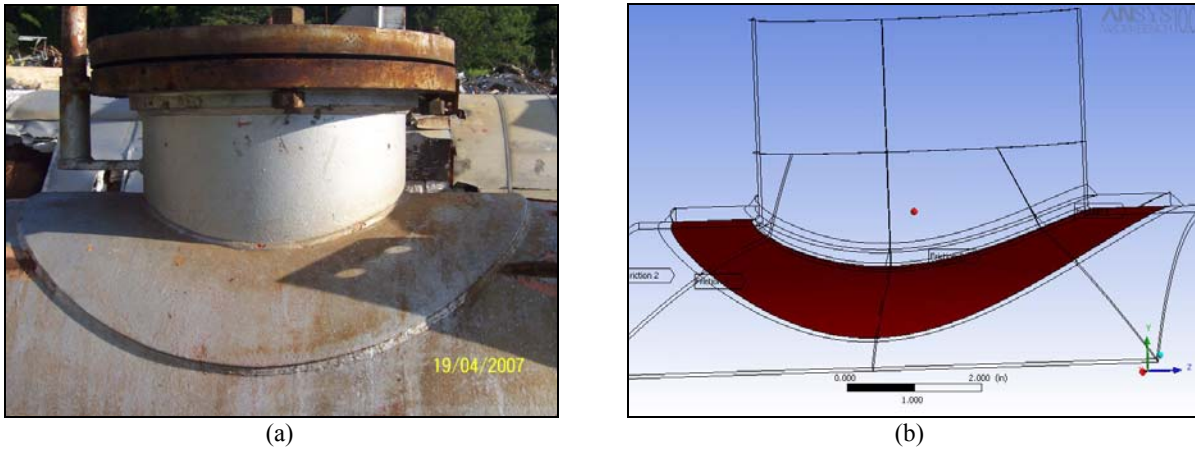


Figure 1. Vessel/nozzle intersection detail: (a) real picture and (b) geometric model

2. METODOLOGY

A finite element computational platform, ANSYS WORKBENCH, version 10 (ANSYS, 2005), was employed to perform linear and nonlinear elastic stress analysis of the vessel/nozzle intersection. Isotropic homogeneous material was assumed for all analyses.

The work consisted of two phases:

- Phase 1: validation of the computational platform and simulation parameters, and
- Phase 2: use of 3D finite element models to perform stress analysis on integrally and partially reinforcing pad vessel/nozzle intersection.

The objective of Phase 1 was to validate the computational platform. The work of Widera and Xue (2004) was used as validation baseline, since it included experimental results. The geometry definitions and recommendations regarding finite element modeling techniques were also adopted in an attempt to replicate the results. In this phase, models with shell elements, with 8-node 3D elements and with 20-node 3D elements were developed and the results compared with the experimental results (Widera and Xue, 2004).

Phase 2 was performed to access the pad reinforcement effect on the stress field of the nozzle/vessel intersection. After the model validation, pad geometry was added on the 20-node 3D element model generated on Phase 1. Stress analysis was then performed on the latter model for two hypotheses:

- bonded pad – pad is integrally welded on the vessel/nozzle intersection, and
- partially welded pad – borders of the pad are considered bonded simulating welding lines and friction contact hypothesis is assumed in the pad/vessel contact surface.

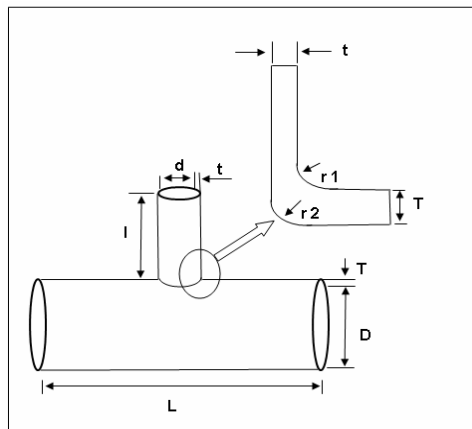


Figure 2. Geometry definition parameters

Table 1. Geometric parameters used

Description of parameter	Value	
	(in)	(mm)
Internal diameter of the vessel (D)	9.80	248.9
Thickness of the vessel (T)	0.10	2.5
Length of the vessel (L)	80.00	2032.0
Internal diameter of the nozzle (d)	4.90	124.5
Thickness of the nozzle (t)	0.05	1.3
Length of the nozzle (l)	14.50	368.3
Outer notch radius (r1)	0.05	1.3
Inner notch radius (r2)	0.05	1.3

2.1. Geometry definition

The geometry definition for the model, showed in Fig. 2, used the parameters listed on Tab. 1, resulting in the ratios $t/T=0.5$, $d/D=0.5$ and $D/T=98$.

2.2. Finite element modeling

2.2.1. Modeling and simulation parameters

High order 20-node volumetric 3D elements with 3 degrees of freedom per node were used in the nozzle, pad and vessel meshing.

The following recommendations regarding FEA modeling and simulation (Widera and Xue, 2004) were considered:

- about 100 elements around the circumference are desired for accurate determination of stress fields for pressure loading;
- about 3 elements through the wall thickness results in more accurate stress field prediction; however, on a practical point of view 2 elements through the wall thickness are sufficient.
- the finite element model should have a refined region at the intersection corresponding to $3.0\sqrt{RT}$ for the vessel and $3.0\sqrt{rt}$ for the nozzle;
- the element size at the nozzle/vessel intersection should be less than $0.02\sqrt{RT}$ for the vessel and $0.02\sqrt{rt}$, for the nozzle.

Figure 3 shows the finite element mesh used.

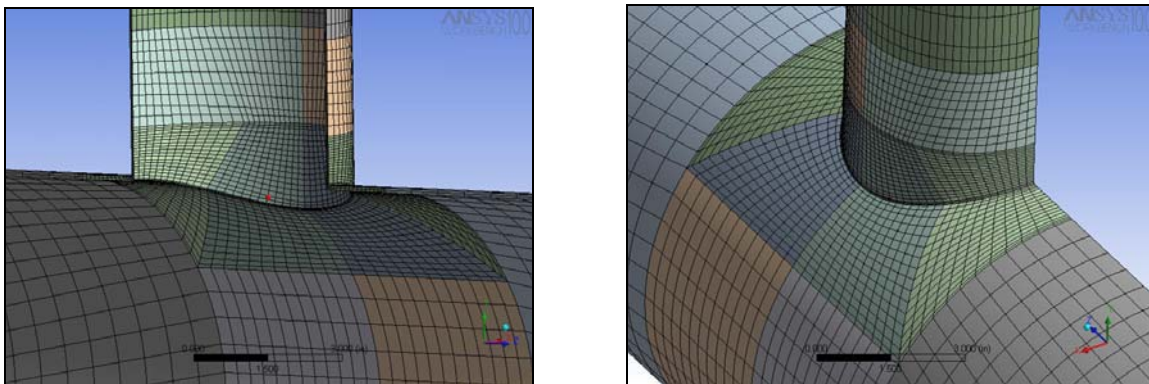


Figure 3. Vessel/nozzle intersection finite element meshing views

2.2.2. Boundary conditions and loading

Due to the symmetry of loading and geometry in the longitudinal plane, only one half of the vessel was modeled. Boundary conditions reflecting symmetry were imposed on all nodes located on the structure's symmetry plane. The left end of the vessel is assumed to have a fixed support while the right end is free. The nozzle end (top) is also free. In order to simulate the contained pressure, equivalent axial stresses were imposed as boundary conditions at both free ends of the model (vessel and nozzle).

Internal pressure of 0.35 MPa (50 psi) was applied at the vessel and nozzle and the influence of structure's weight was not considered in the simulation. Equations 3 and 4 show the axial stress imposed at the circular sections of the free ends (vessel and nozzle) when the internal pressure is applied.

At the vessel:

$$\sigma_z = \frac{P \times R}{2 \times T} = 8.45 \text{ MPa} = 1225 \text{ psi} \quad (1)$$

At the nozzle:

$$\sigma_y = \frac{P \times r}{2 \times t} = 8.45 \text{ MPa} = 1225 \text{ psi} \quad (2)$$

In order to obtain the stresses orientations, the coordinate system "S" was adopted. Figure 4 shows the longitudinal plane of the vessel/nozzle intersection. The radial stress is taken to be the stress in the S-direction, while the tangential stress is the stress perpendicular to the S-direction.

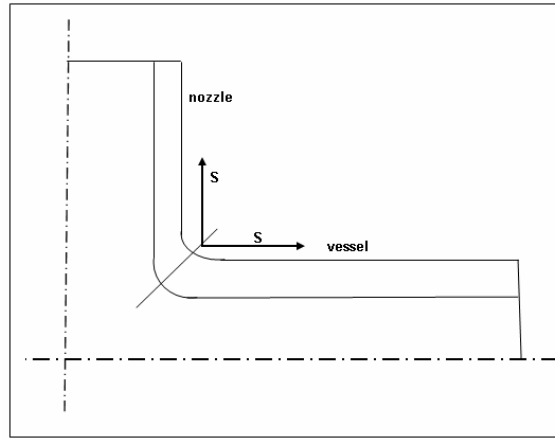


Figure 4. Definition of the coordinate S

2.2.3. Modeling the contact behavior

Contact is changing-status nonlinearity. That is, the stiffness of the system depends on the contact status, whether parts are touching or separated. Physical contacting bodies do not interpenetrate. Therefore, the computational platform ANSYS WORKBENCH establishes a relationship between the two surfaces to prevent them from passing through each other in the analysis. When the program prevents interpenetration, it said that it enforces contact compatibility (ANSYS, 2005). Simulation offers several different contact algorithms to enforce compatibility at the contact interface. In the meshing generation, elements called "CONTACT" and "TARGET" are created on the interface of pad reinforcement and shell, respectively.

ASME Code Criteria – Area Replacing Method (ASME, 2006) was used for designing the pad reinforcement. For modeling the bonded pad model it was used the specific case of "bonded" type of contact, the Multi-Point Constraint (MPC) formulation supported by ANSYS WORKBENCH. It internally adds constraint equations to "tie" the displacements between contacting surfaces. This approach is not penalty-based or Lagrange multiplier-based. It is a direct, efficient way of relating surfaces of contact regions which are bonded.

For modeling the partially welded pad model, it was assumed frictional contact hypothesis between the contacting internal surface of the pad and external surface of the shell. The formulation used is Lagrange multiplier-based supported by ANSYS WORKBENCH (ANSYS, 2005). The weld beads are modeled as MPC's lines at the intersection of the nozzle neck and the internal circumference of the pad and shell

3. RESULTS

Phase 1 (model validation analysis) was performed for the non-reinforced structure model only, using 3D 20-node elements. Results obtained in the analysis were compared to results available in the literature (Widera and Xue, 2004) for model validation. Widera and Xue (2004) also performed finite element analysis for a shell element and for a 3D 8-

node element models. Figures 5 and 6 present, respectively, this comparison for tangential stresses on the external surface of the vessel and nozzle along the longitudinal plane (coordinate S, Fig. 4). It can be seen that shell element, 3D 8-node and 3D 20-node 3D element models presented similar results. However, 3D 20-node element model could capture the peak stress area and consequent stress gradients of the experimental results and was validated to be used on the pad reinforcement simulation (phase 2).

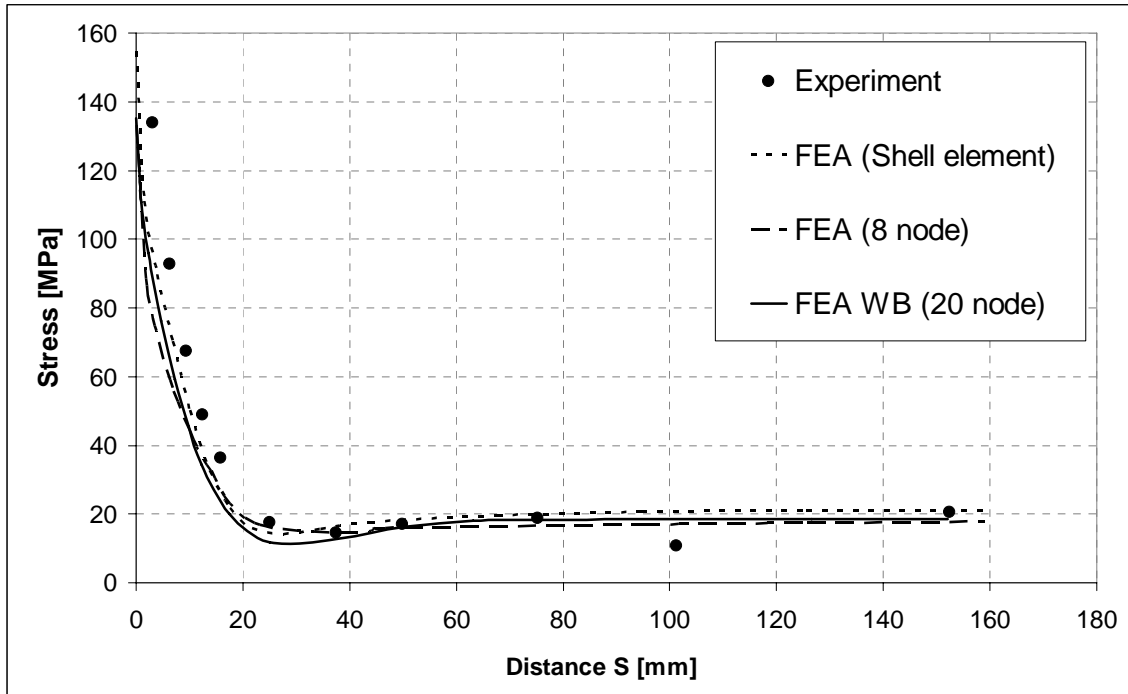


Figure 5. Tangential stress distribution in longitudinal plane of vessel

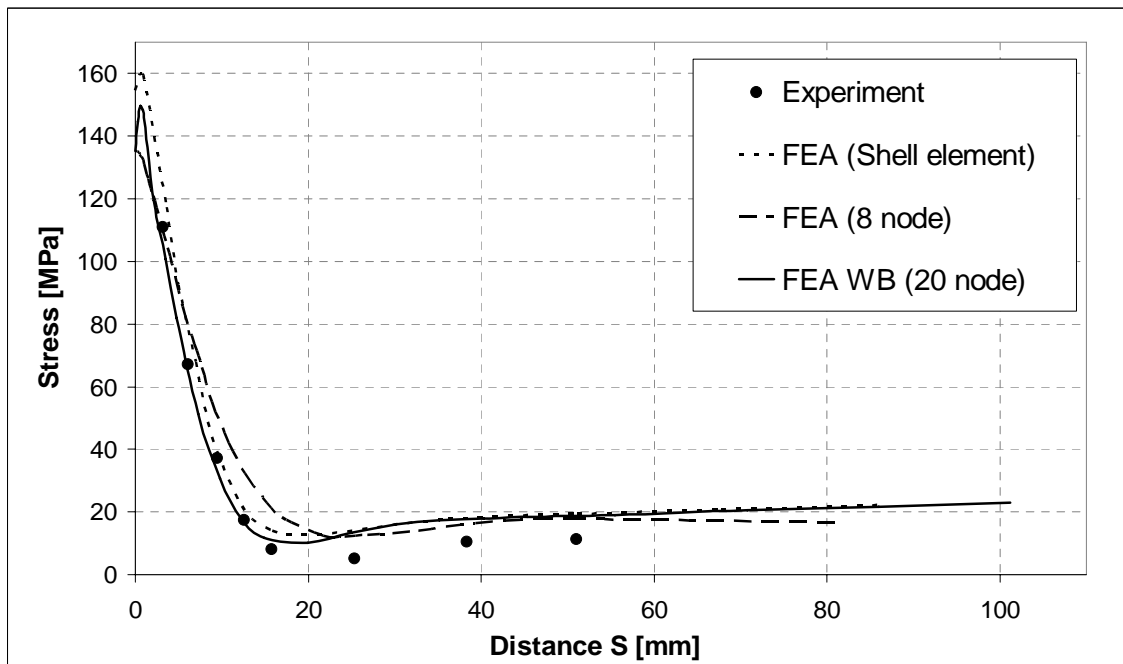


Figure 6. Tangential stress distribution in longitudinal plane of nozzle

Phase 2 analysis was performed on three models: non-reinforced, bonded pad reinforced and partially welded reinforced 3D 20-node element models. Results were plotted on Figs. 7, 8, 9 and 10.

Figures 7 and 8 show, respectively, the tangential and radial stress distributions on the external surface of the vessel, plotted together with experimental results for the non-reinforced structure. It can be seen that stresses significantly decrease for the models that include a pad reinforcement on the vessel/nozzle intersection. Decrease as large as 3 times are verified on the peak stress region, demonstrating the effectiveness of the pad reinforcement. Results for bonded pad and partially welded pad models differed around 20% for the tangential stresses on this region. However, radial stresses for the bonded pad model are substantially smaller than for the partially welded pad model, indicating the necessity of having a model that approximate to the real structure for adequate prediction.

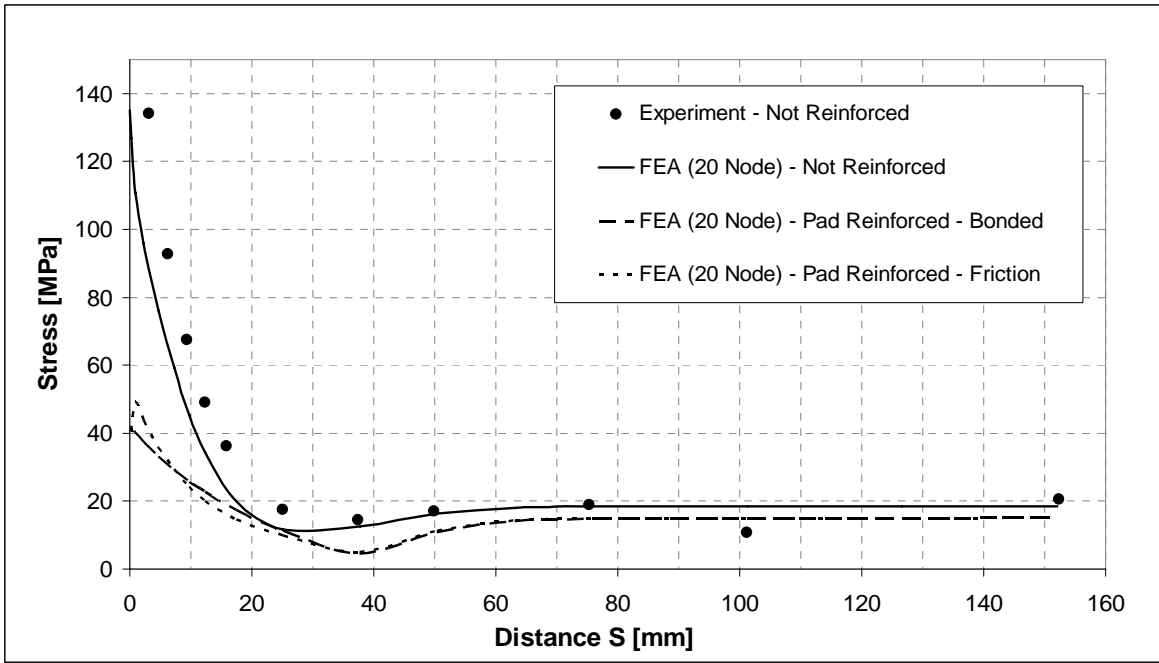


Figure 7. Tangential stress distribution in longitudinal plane of vessel with/without reinforcement

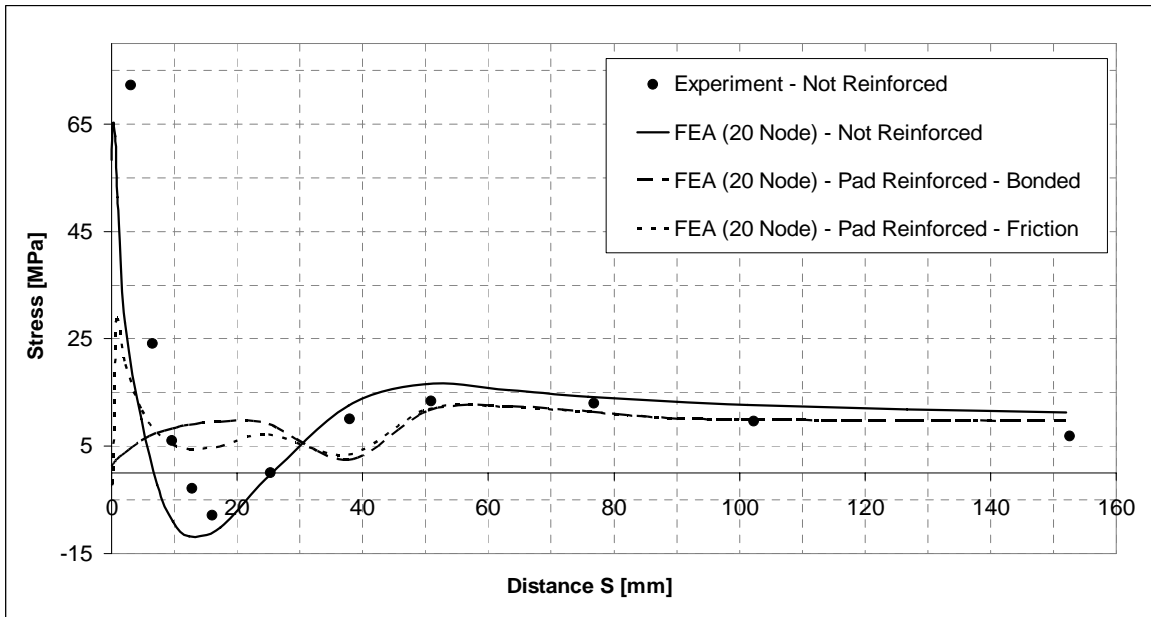


Figure 8. Radial stress distribution in longitudinal plane of vessel with/without reinforcement

Figures 9 and 10 show, respectively, the tangential and radial stress distributions on the external surface of the nozzle, plotted together with experimental results for the non-reinforced structure. Again it can be seen that stresses substantially decrease (over 3 times) for the models that include a pad reinforcement on the vessel/nozzle intersection. Differences between the bonded pad and partially welded pad models are minimal on the nozzle, and can be neglected.

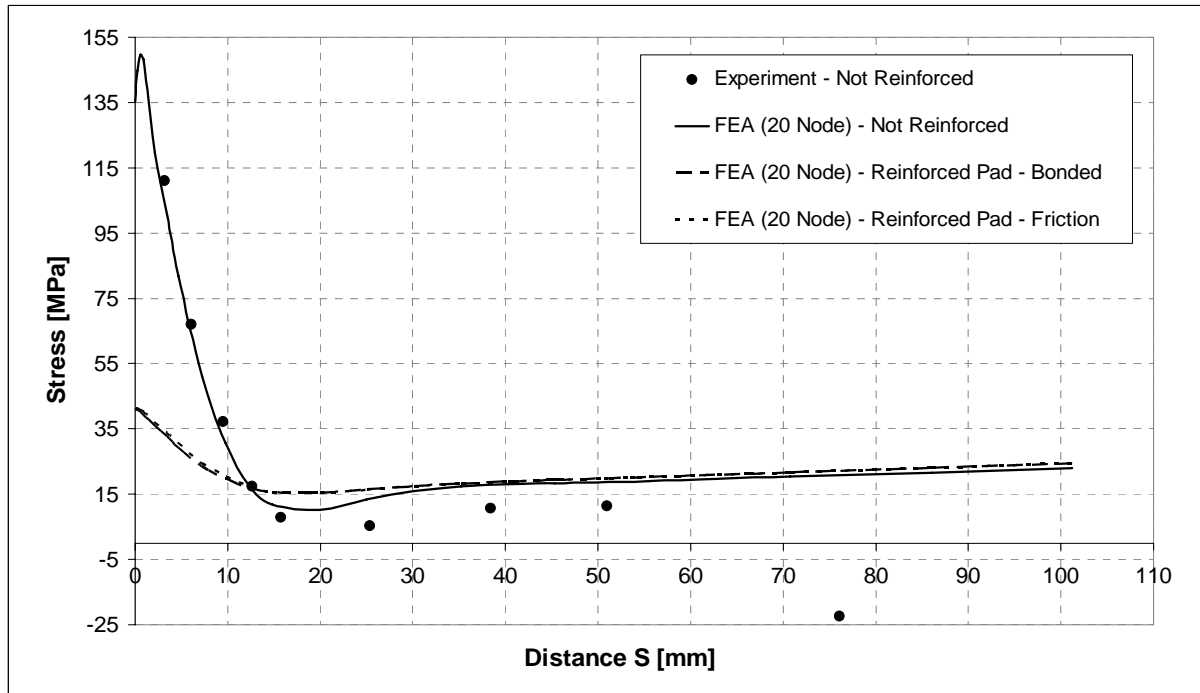


Figure 9. Tangential stress distribution in longitudinal plane of nozzle with/without reinforcement

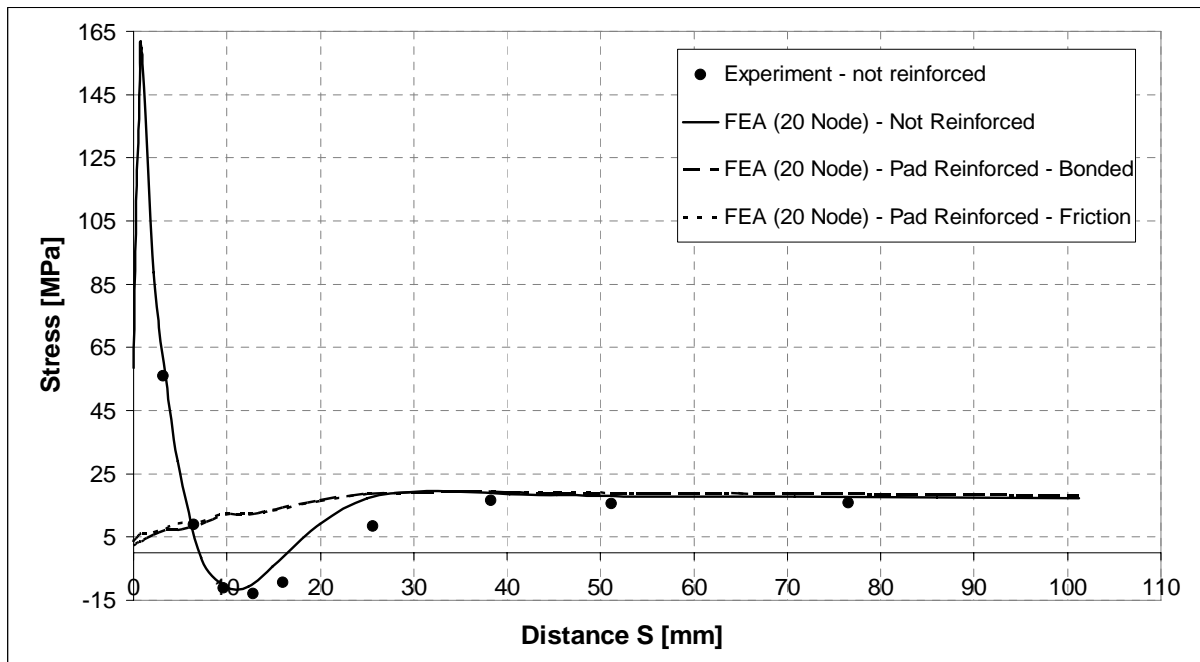


Figure 10. Radial stress distribution in longitudinal plane of nozzle with/without reinforcement

4. CONCLUSIONS

Stress analysis on vessel/nozzle intersections were performed for cylindrical pressure vessels both non-reinforced and pad reinforced using finite element models. On phase 1, results for non-reinforced vessel models were compared to experimental data from the literature, validating the 3D 20-node element model.

Models representing two conditions of pad reinforcement (bonded pad and partially welded) were then analyzed and results were compared. Pad reinforcement significantly reduced both tangential and radial peak stresses on the vessel/nozzle region, corroborating the ASME Code Criteria – Area Replacing Method requirements for vessel/nozzle intersection reinforcement.

Although negligible for the nozzle region, bonded and partially welded pad reinforced models, however, presented higher differences on stress levels for the vessel region for both tangential and radial stresses. The partially welded pad reinforced model presented tangential stresses 20% higher and radial stresses were significantly higher than the bonded pad reinforced model. This fact suggests that a more accurate representation of the real configuration of the structure should be attempted in order to make adequate predictions. Results can be used by designers as guidelines for modeling reinforced cylindrical vessel/nozzle intersections.

5. REFERENCES

- ANSYS, 2005, “Ansys 9.0 Program Documentation”, SAS IP, Inc.
- ASME, 2006, ASME Boiler and Pressure Vessel Code, 2004, Sec. III, Div. 1, American Society of Mechanical Engineers, New York.
- BS 5500, 2004, Specification for Unfired Fusion Welded Pressure Vessels, 2004, British Standards Institution, London, UK.
- Chen, H.C. and Schnobrich, W.C., 1980, “An Elastic-Plastic Analysis of Normally Intersecting Cylinders”, Civil Engineering Studies, University of Illinois, 1980.
- Dekker, C.J. and Stikvoort, W.J., 1997, “Pressure Stress Intensity at Nozzles on Cylindrical Vessels: A Comparison of Calculation Methods, International Journal of Pressure Vessel and Piping, 74, pp. 121-128.
- Liu, Y.H., Zhang, B.S., Xue, M.D., Liu, Y.Q., 2004, “Limit Pressure and Design Criterion of Cylindrical Pressure Vessels With Nozzles”, International Journal of Pressure Vessels and Piping, 81, pp. 619-624.
- Natarajan, R., Widera, G.E.O., Afshari, P., 1987, “A Finite Element Model to Analyze Cylinder-Cylinder Intersections”, Journal of Pressure Vessel Technology, Transactions of ASME, Vol. 109, pp. 411-420.
- Taagepera, J. and McKay, M., 2006, “Nozzle Reinforcement: Constant vs. Variable”, ASME Pressure Vessels and Piping Division Conference, PVP 2006-ICPVT-11-93596.
- Wang, H.F., Sang, Z.F., Xue, L.P. and Widera, G.E.O., 2006, “Elastic Stresses of Pressurized Cylinders With Hillside Nozzle”, Journal of Pressure Vessel Technology, ASME, Vol. 128, pp. 625-631.
- Werneck, H.S. and Tinoco, E.B., 2000, “Avaliação de Trincas em Bocais de Vasos de Pressão”, VII ENCAT, PETROBRAS.
- Widera, G.E.O and Xue, L.P., 2004, “Guidelines for Modeling Cylinder to Cylinder Intersections”, Weld Research Council – Bulletin 493.
- Xue, L.P., Widera, G.E.O. and Sang, Z.F., 2003, “Influence of Pad Reinforcement on the Limit and Burst Pressures of a Cylinder-Cylinder Intersection”, Journal of Pressure Vessel Technology, Vol.125, pp. 182-187.
- Xue, M.D., Du, Q.H., Li, D.F. and Hwang, K.C., 2006, “A Stress Analysis Method for Cylindrical Shells With Nozzles Subjected to Internal Pressure, External Forces and Moments”, ASME Pressure Vessels and Piping Division Conference, PVP 2006-ICPVT-11-94053.
- Zang, W., Tiecheng Yang, X.C., and Yu, J.L., 2006, “Stress Analysis and Assessment for Nozzle Welding Area Enduring High Pressure and Temperature”, ASME Pressure Vessels and Piping Division Conference, PVP 2006-ICPVT-11-93440.

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.