

CYLINDRICAL VESSEL LIMIT LOAD ESTIMATION FOR OBLIQUE NOZZLE BY USING ANSYS AS ANALYSIS TOOL

¹DILIP M PATEL, ²Dr. BIMLESH KUMAR

¹Department of Mechanical Engineering D.N.Patel C.O.E, Shahada-420425

²Department of Mechanical Engineering C.O.E, Faizpur-425524

dilip1663@rediffmail.com

ABSTRACT: Pressure vessel contains with different inlet & outlet openings called nozzle or valves. The design parameter of these valves may differ in one pressure vessel. These valves cause geometric discontinuity of the pressure vessel wall hence stress concentration may occur around the valve or nozzle. Since due to the high stress concentration there may be the chances of failure of vessel junction. Hence detail stress distribution analysis needs to be done for pressure vessel.

Determination of limit pressure at different location on pressure vessel by using finite element method is less time consuming and it avoid complex mathematical work at difficult geometries. So, it is essential to validate the result. Experiments is conducted on oblique nozzle (45° with shell axis) & result obtained are used to validate the finite element results. Distortion measurement test by measuring change in diameter of vessel after vessel is pressurized using water. Twice elastic slope method & Tangent intersection method are used to find out limit load estimation of cylindrical vessel with oblique (45°) nozzle.

Key Words: Pressure Vessel, Limit Load, Nozzle

1. INTRODUCTION

Pressure vessel is a closed container designed to hold gases or liquids at a pressure different from the ambient pressure. It is applied with a differential pressure between inside and outside.

In the field of pressure vessel design, welded pipe nozzles and welded nozzles of vessels are generally subjected to high loads. Because of nozzle necessary for the exchange of fluid or gas causes high stresses at the edge of opening merely caused by operating pressure. This basic load is overlaid by additional loads due to connected pipe. These additional load results for example from restrained thermal expansions, vibration of pipes or shock pressures caused by the opening of valves.

Pressure Vessels are generally used in Storage vessels (for liquefied gases such as ammonia, chlorine, propane, butane, household gas cylinders etc.), Chemical industries (as a distillation tower, domestic hot water storage tank etc.), Medical field (as a autoclaves), Aerospace (as a habitat of spaceship), Nuclear (as a nuclear vessels), Pneumatic & hydraulic reservoirs under pressure, Rail vehicle airbrake reservoir, road vehicle airbrake reservoir, power, food and many other industries.

Opening in pressure vessel in the region of shells or heads are required to serve the following purpose

- Man ways for in and out of vessel to perform routine maintenance and repair.
- Holes for draining and cleaning the vessel.

- Hand hole openings for inspecting the vessel from outside.

Nozzle attached to pipes to convey the working fluid inside and outside of the vessel. Also, various failure mode of pressure vessel in industrial pressure vessels fail due to

- Excessive elastic deformation- induced stresses and elastic bending
- Elastic instability- column instability and vessel shell under axial load
- Plastic instability, brittle rupture, corrosion

Now, the pressurized cylindrical vessel with oblique nozzle is considered for experimental analysis. The three dimensional nonlinear elastic plastic finite element analysis is performed.

Also experimental hydrostatic test at increasing pressure in step is also conducted to validate with FEA result and hence present study provides some useful data to serve as a check on existing design method & basis for developing more accurate guideline.

2. METHODS TO CALCULATE LIMIT LOAD

There are different methods to calculate limit load and some of them are explained below

Twice Elastic Slope Method (TESM)

Twice elastic slope method is one which can be used with either experimental data or FE analysis. Limit Load can be found out from either distortion measurement test or Strain measurement.

By F.E. Analysis

FEM can be used for determination of limit load using twice elastic slope method. It requires non-linear analysis to find out limit load of structure. Non linearity is the material behavior as its young's modulus (E) changes from point to point shown in figure 1.

In order to define material behavior, different type of material models are used which are available in commercial FE packages of which multi-linear Elastic Isotropic Hardening, Bilinear isotropic hardening etc. After performing analysis, plot between load deformation & load strain is taken & procedure similar to experimental analysis method in order to obtain limit load.

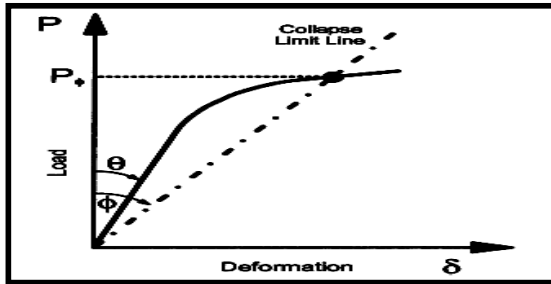


Figure 1. Twice Elastic Slope Method

Tangent Intersection Method (TIM)

Tangent intersection method is used to find out limit load of structure and requires non-linear analysis.

By FE Analysis

For determination of limit load using tangent intersection method, FEA is used. It requires non-linear analysis to find out limit load of structure. Non linearity is in the material behavior as its young's modulus (E) changes from point to point shown in figure 2.

In order to define material behavior different types of material models are used which are available in commercial FE packages out of which multi-linear Elastic Isotropic Hardening, Bilinear isotropic Hardening are mostly used for performing analysis.

3. EVALUATION OF LIMIT LOAD BY FINITE ELEMENT ANALYSIS

The model was simulated in Ansys and discretized with regular mapped meshing technique. A three dimensional iso-parametric type solid brick element

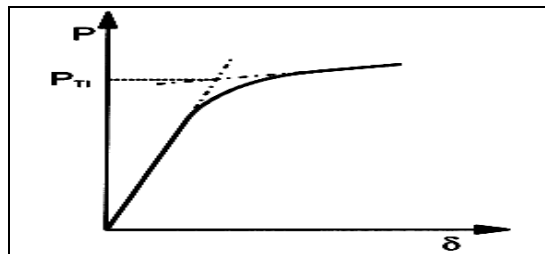


Figure 2. Tangent Intersection Method

Defined by eight nodes having 3- degree of freedom at each node is utilized for brick meshing.

Principally, the model size & the boundary conditions used for modeling a shell- nozzle intersection strongly influence the calculation results; the influence increases with increasing ratio of nozzle dia. to shell dia. and with decreasing wall thickness. Hence very fine mesh is maintained near to the area of shell to nozzle intersection to get accurate & precise results.

Dimensions of Model Vessels

The dimensions of vessel considered under analysis are as given in table 1.

Table 1 (Parameters of model vessel for analysis)

| Parameter | Dimension |
|---------------------|-----------|
| Mean Dia. of shell | 1012 mm |
| Mean dia. of nozzle | 213 mm |
| Thickness of shell | 12 mm |
| Thickness of nozzle | 8.20 mm |
| Length of shell | 2000 mm |
| Length of nozzle | 700 mm |

A three dimensional Iso-parametric type Solid brick element defined by eight nodes

Table 2 (Pressure vessel model definition)

| | |
|---------------------------------|---|
| Material of shell | 516 Gr.70 |
| Material of nozzle | 516 Gr.70 |
| Design code | ASME section VIII, div-I, Edition-2007 |
| Design pressure | Internal pressure=1.8 MPa External pressure = 0.1034 MPa |
| Max. allowable working pressure | 3.26 MPa |
| Weight of vessel | 900 kg |

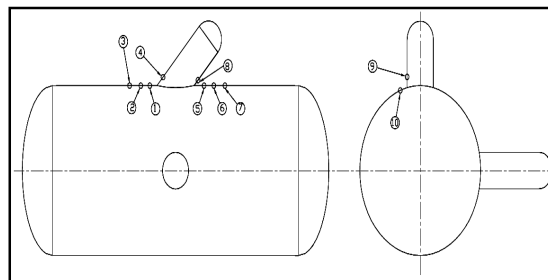
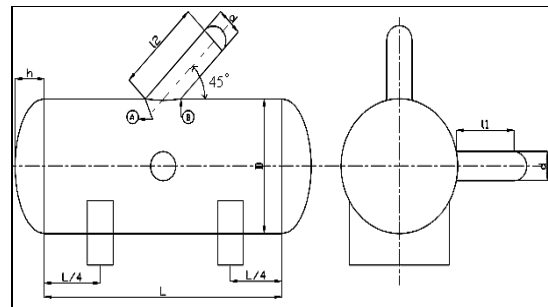


Figure 3. Various Location on Pressure Vessel

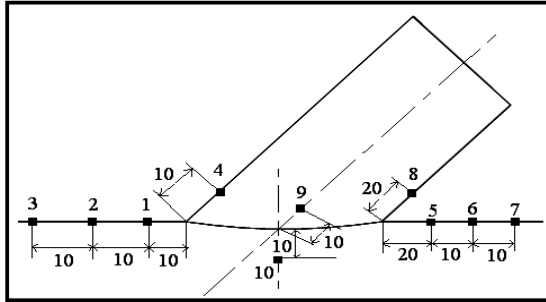


Figure 4. Distance (in mm.) from weld intersection

| SA-516 Gr. 70 | Chemical Composition % | | | | | E (N/mm ²) |
|---------------|------------------------|---------|----------|------------|-----------|------------------------|
| | C | Mn | Si | S | P | |
| | 0.2-0.31 | 0.7-1.3 | 0.1-0.45 | 0.035 max. | 0.03 max. | 2x10 ⁵ |

Boundary Conditions

- Hoop displacement and longitudinal displacement in nodes at one end of the shell on strained to zero.
- On the other end of the shell equivalent thrust (PD/4t) is applied. Similarly on the other end of the nozzle equivalent thrust (Pd/2t) is applied.
- Pressure was applied internally with increment steps.
- Symmetry boundary conditions in the plane along the longitudinal direction of the shell.

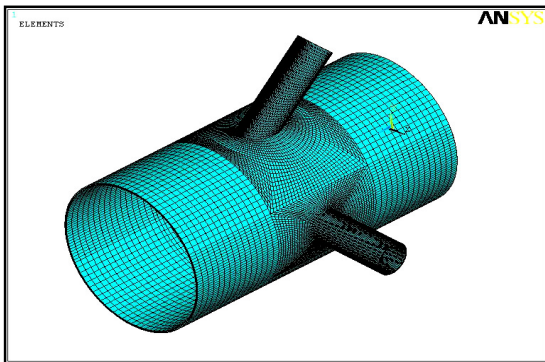


Figure 5. Mapped meshed model of experimental vessel

Loading Steps

Pressure is not applied in single step but applied in multiple steps in gradual incremental manner. The loading steps are reported in table.

Table 3 (Steps of pressure increment)

| | | | | | | | | | |
|-----------------------|-----|-----|-----|-----|-----|-----|----|----|--|
| Step No. | 1 | 2 | 3 | 4 | 5 | 6 | | | |
| Pressure (MPa) | 0.5 | 1 | 1.5 | 2 | 2.5 | 3 | | | |
| 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | |
| 3.5 | 4 | 4.5 | 5 | 5.5 | 6 | 6.5 | 7 | 8 | |

Material properties

Material used for analysis is SA 516 Gr.70 (low carbon steel). As the main purpose of this work is to find the limit pressure of the shell intersection, the yield, ultimate stress & stress – strain curve of the material are important parameters. The material properties and chemical composition are given in table 4.

Table 4 (Material properties & chemical composition)

| | |
|------------------------------|------------------------|
| Material | SA-516 Gr. 70 |
| Yield Strength | 360 MPa |
| Ultimate Strength | 543 MPa |
| Modular of elasticity | 180000 |
| Poisson's Ratio | 0.30 |
| Density ρ | 7833 kg/m ³ |

The material model used for analysis is multi linear isotropic hardening model which is described by seven points are considered to define material behavior which are noted in Table 5 & its plots in ANSYS is also shown in Figure 6.

Table 5 (Multi linear material model points)

| Strain (μϵ) | Stress (MPa) |
|-------------|--------------|
| 1 | 2000 |
| 2 | 37500 |
| 3 | 45000 |
| 4 | 64000 |
| 5 | 93000 |
| 6 | 16950 |
| 7 | 209000 |

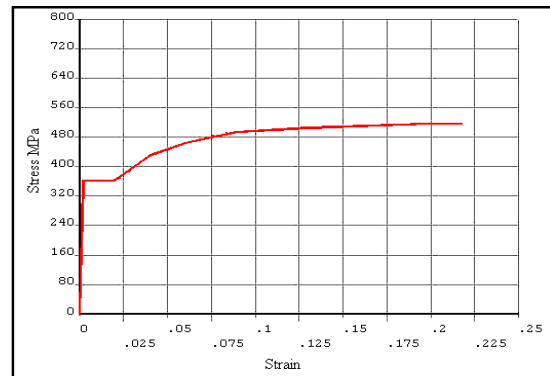


Figure 6. Material model in ANSYS

4. RESULTS AND DISCUSSION

Analysis resulted output is explained in different terms

Deformation

Deformation is found using finite element analysis, the obtuse side of nozzle deforms making the lateral angle between shell and nozzle axis to increase. The weld zone area shrinks in longitudinal section and swells in transverse section making the top portion of the nozzle to rise up as shown in Figure 7.

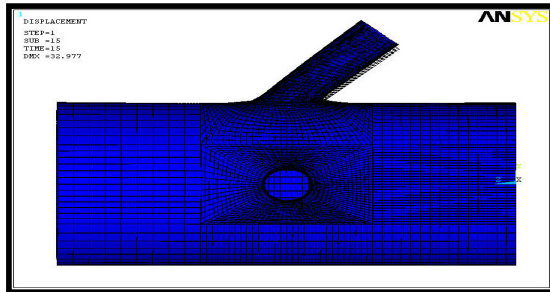


Figure 7. Deformation at 8 MPa Pressure

Stress Distribution

The Von-Mises Stress distribution is shown in Figure 8. Obviously the maximum stress originates at the acute side of the oblique nozzle making it highly stress concentrated area. The high stress pattern lines are in proximate around the junction area with reduction in strength of shell to nozzle joint vessel at high pressures.

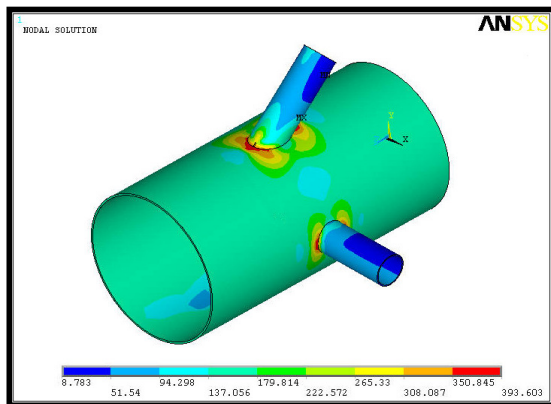


Figure 8. Stress distribution at 4 MPa pressure

Load Strain Plot

Strain plot shown in below figures and are the load-strain plots with limit load estimated with the help of same plots by using finite element analysis.

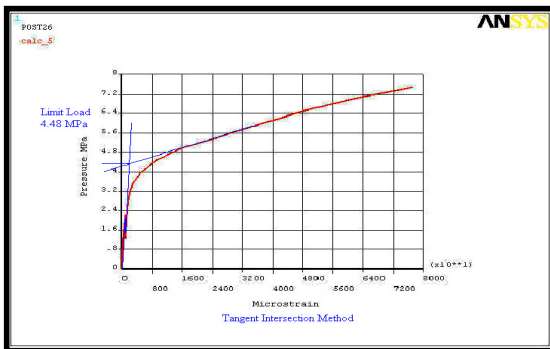


Figure 9. Load Strain Plot at node for Location No.1:
Limit Load for Node by TIM

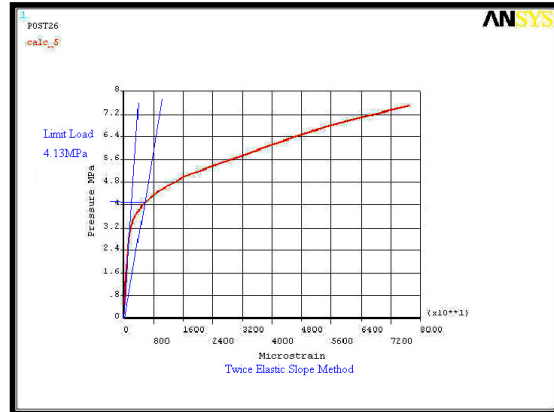


Figure 10. Load Strain Plot at node for Location No.1:
Limit Load for node by TESM

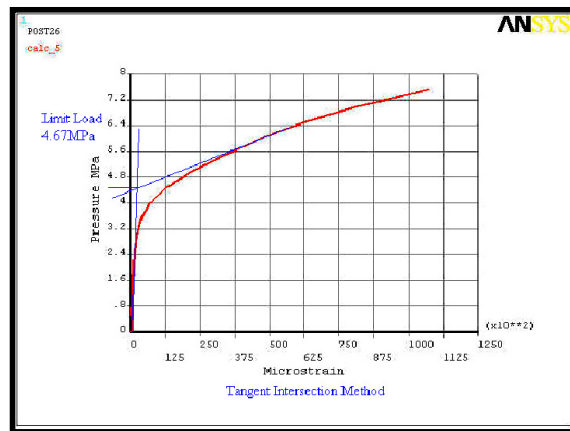


Figure 11. Load Strain Plot at node for Location No.5:
Limit Load for Node by TIM

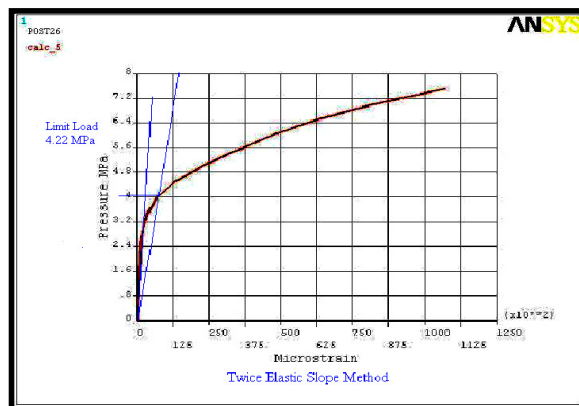


Figure 12. Load Strain Plot at node for Location No.5:
Limit Load for Node by TESM

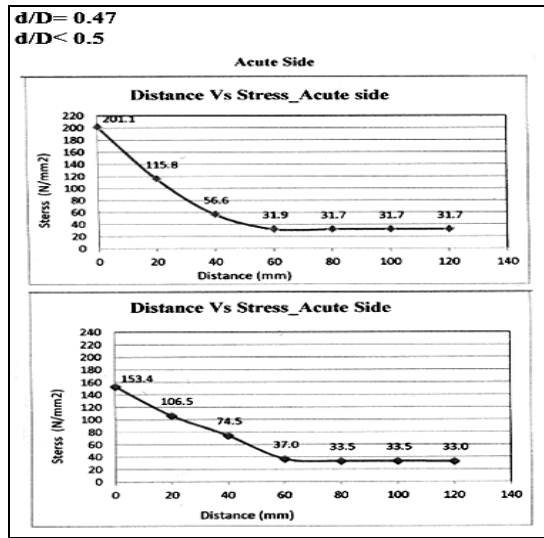


Figure 13. Stress Vs Distance plot

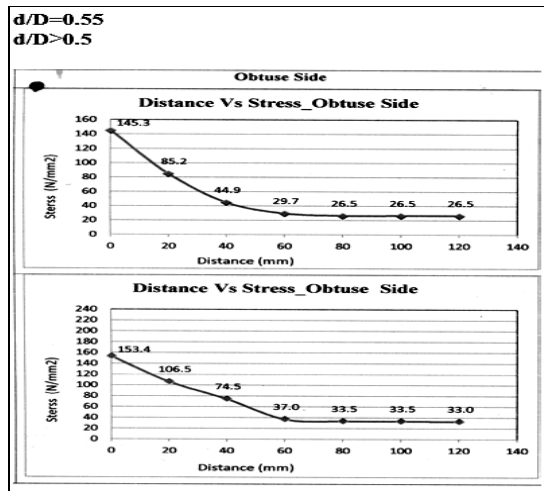


Figure 14. Stress Vs Distance plot

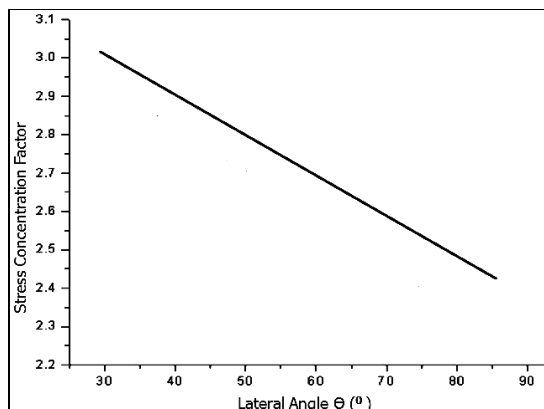


Figure 15. Ideal Curve for Lateral Angle Vs SCF

5. CONCLUSION

1. The value of limit load is more at acute side than obtuse side of inclined nozzle under internal pressure. As d/D ratio increases, the stresses at junction also increased and as inclination angle between shell axis and nozzle axis decreases the stresses at acute side get increase.

2. Definite deformation occurs at the intersection area of shell and nozzle, it result the intersection region shrinks in longitudinal section of the shell and nozzle, while bulging appears at transverse section.

Stress concentration factor (SCF) of the pressure vessel with inclined nozzle decreases with an increase in lateral angle θ for constant d/D , D/T , and t/T ratio.

3. Plasticity starts at the acute side of nozzle junction and smoothly grows near around joint across the obtuse side.

4. To evaluate the limit load using various methods TIM, TESH, PWC, the TESH & TIM is the method to estimate the lower value of limit load & is more effective for higher elastic slope of load V/S strain plot.

5. Finite element method is accurate and precise computational tool to simulate the model and predict the failure location of lateral (inclined) connection configuration and successfully interpret the result in required format.

6 REFERENCES

1. K.D. Clare and S.S. Gill, "Effect of vessel diameter /thickness ratio on the behavior beyond the elastic limit of flush nozzles in cylindrical pressure vessels Experimental investigation Journal mechanical engineering Science (1966) vol. – 8.
2. ASME Boiler and pressure vessel code section VIII Div.2, Appendix 4, and New York: American Society for mechanical Engineers; 2004.
3. K. Mokhtarian and J.S. Endicott, "Stresses in interesting cylinders subjected to pressure", WRC Bulletin 368.
4. Z.F. Sang, Z.L. Wang, L.P. Xue and G. o. Widera, "Elastic Stress of pressurized cylinders with hill side nozzle "Journal of pressure vessel and Technology, Nov.2006 – vol.128 pp625-331.
5. K. Naderan and M. Robinson. "Plastic limit pressures for neighboring radial nozzles in a spherical pressure vessel, journal of process mechanical engineering (1996) pages 75-78.
6. Fu- Zhen Xuan and Pei – Ning Li, "Finite Element - based limit load of piping branch junctions under combined loadings", Nuclear Engineering and Design (2004) Vol. – 231 pages 141-150.
7. Somnath Chattopadhyay, "Pressure Vessels Design and Practice", 2nd edition, CRC Press, 2005.