# "Structural Analysis of Nozzle Attachment on Pressure Vessel Design"

**Pravin Narale[1] , Prof. P. S. Kachare[2]**
[1](Mechanical Department, College of Engineering Pandharpur/ Solapur University, India
[2] (Associate Professor in Mechanical Department, College of Engineering Pandharpur/ Solapur University, India

## ABSTRACT

**In Pressure Vessels, Nozzles are required for inlet and outlet purposes. If these nozzle present on peak of the dish end do not disturb the symmetry of the vessel. However sometimes process requires that nozzles to be placed on the periphery of the pressure vessel. These nozzles disturb the symmetry of the vessel.**

**Geometrical parameters of nozzle connections may significantly vary even in one pressure vessel. These nozzles cause geometric discontinuity of the vessel wall. So a stress concentration is created around the opening. The junction may fail due to these high stresses. Hence a detailed analysis is required.**

**If nozzles are placed on the periphery of a pressure vessel, they disturb the axis symmetry of the system and cause eccentricity. Sometimes this cause generation of a couple & lead to a structural imbalance.**

**The presence of nozzle is going to hinder the radial expansion of the structure. Here we shall consider the one end of the nozzle fixed to an assembly. One Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Then to see what will be the effect on system stresses in that case.**

**So that it need to analyzed in FEA to understand effects of nozzle on Stress attributes of the vessel.**

**Keywords –** Hexahedron & Tetrahedron meshing, Non Linear Finite Element Analysis, Nozzle Junction, Monotonic Convergence, Pressure Vessel.

## 1. Introduction:

Pressure vessels are widely applied in many branches of industry such as chemical and petroleum machine-building, nuclear and power engineering, gas, oil and oil-refining industries, aerospace techniques, etc. As the name implies these are important components of processing equipment. Nozzles or opening are necessary in the pressure vessels to satisfy certain requirements such as inlet or outlet connection, manholes, vents & drains etc. Welded nozzles connecting a pressure vessel to piping can be placed both on the cylindrical shell and the heads of the vessel [1-2].

Geometrical parameters of nozzle connections may significantly vary even in one pressure vessel. These nozzles cause geometric discontinuity of the vessel wall. So a stress concentration is created around the opening. The junction may fail due to these high stresses. Hence a detailed analysis is required. One of the parts of overall structural analysis for nozzle connections is the stress analysis of two intersecting shells [4, 6].

Due to different loadings applied to these structures, a local stress state of nozzle connection characterized by high stress concentration occurs in intersection region. Internal pressure is primary loading used in the structure analysis for determination of main vessel-nozzle connections. However the effect of external forces and moments applied to nozzle should be taken into consideration in addition to the stresses caused by the internal pressure. External loading usually are imposed by a piping system attached to the nozzle. Values of the loads & moments are calculated by an analysis of piping system [9].

Many works including analytical, experimental & numerical investigations have been devoted to the stress analysis of nozzle connections in pressure vessels, subjected to different external loadings. The codes suggest a procedure to design the junction, but do not provide any methodology to calculate the extended and magnitude of these high stresses. The available analytical solution WRC-107 is limited to simple geometries [5-8]. So, there is need to carry out a detailed finite element analysis of the junction to calculate stresses at the junction & both in the vessel & in the nozzle. ANSYS package is used as a finite element tool [11] .

### 1.1 Scope for the work:

1) If nozzles are placed on the periphery of a pressure vessel, they disturb the axis symmetry of the system and cause eccentricity.
2) Sometimes this eccentricity can cause generation of a couple which can lead to a structural imbalance.
3) In a radial nozzle, the presence of nozzle is going to hinder the radial expansion of the structure. Here we shall consider the one end of the nozzle fixed to an assembly.

4) One Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Then to see what will be the effect on system stresses in that case.

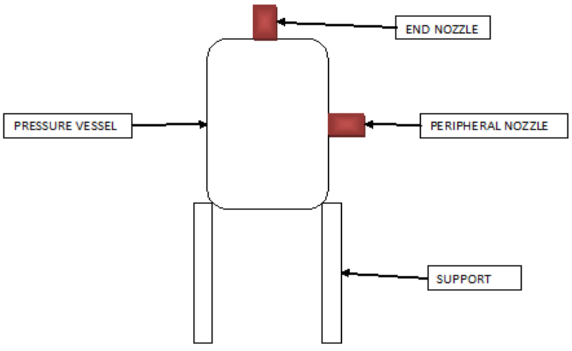



Fig.1.1 Sketch of Pressure Vessel

### 1.2 Need of FEA:

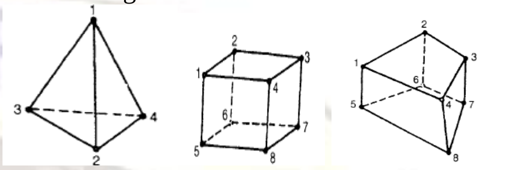
1) To prevent stress related vessel rupture and catastrophic failure, it is necessary to identify the main factors that contribute extensively to stress development in pressure vessels and how they can be mitigated. This work presents critical design analysis of stress development using 3D CAD models of cylindrical pressure vessels assembly and finite element engineering simulation of various stress and deformation tests at high temperature and pressure.
2) ASME code gives formulation to design pressure vessel with nozzles, however the Nozzles are at separate locations, such as dish end, side etc. However ASME code does not have clear guidelines for having Nozzles at the same height and how to counter the effect of the stresses.
3) Quench Nozzles specifically were patented in 2003, and hence they are a recent technology and this further means that standardized testing data is not available for them. Hence the only option is to use Design by Analysis approach prescribed by ASME.

### 1.3 Objective:

1) Finite element analysis of the junction *to* calculate stresses at the junction of pressure vessel and nozzle is performed using ANSYS software.
Analysis is done for to see effect of presence of nozzle on pressure vessel design.
2) Analysis is conducted for pressure vessel with presence of peripheral nozzle to determine the stress conditions with a peripheral nozzle & to determine the stress distribution in pressure vessel and nozzle.
3) To conduct structural analysis in ANSYS for pressure vessel by increasing number of nozzles.
4) Conduct a Study analysis, on the effects of increasing the number of Nozzles on the periphery until full symmetry is achieved.
5) Study effect of placement of nozzle, i.e. to check stress condition after changing angle between nozzles.
6) To find an optimum angle such that the stresses are maintained within limits .
7) To conduct Mesh Sensitivity analysis for all the models in FEA, so that the results can be validated. Mesh Sensitivity Analysis, is the analysis in which the Mesh Size of the analysis is changes to check variation in values of Deformation and Stress.

## 2. Basic element shapes

If the geometry, material properties, and other parameters of the body can be described by three independent spatial coordinates, we can idealize the body by using the three dimensional elements as shown in figure 2



Tetrahedron Rectangular prism Hexahedron

Fig.: 2.1 Three-dimensional finite elements.

For the discretization of problems involving curved geometries, finite elements with curved sides are useful. The ability to model curved boundaries has been made possible by the addition of midsized nodes. Finite elements with straight sides are known as linear elements, whereas those with curved sides are called higher order elements.

### 2.1 Size of Elements:

The size of elements influences the convergence of the solution directly and hence it has to be chosen with care. If the size of the elements is small, the final solution is expected to be more accurate.

### 2.2 Number of Elements:

The number of elements to be chosen for idealization is related to the accuracy desired, size of elements, and the number of degrees of freedom involved. Although an increase in the number of elements generally means more accurate results, for any given problem, there will be a certain number of elements beyond which the accuracy cannot be improved by any significant amount. This behavior is shown graphically in Fig.2.2

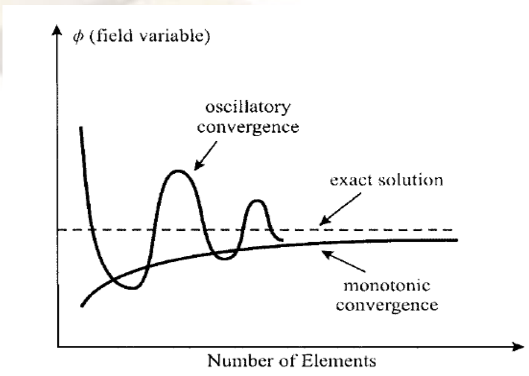



Fig.2.2 Relation between number of elements and the accuracy.

### 3. Static structural analysis:

Static structural analysis is one in which the load/field conditions does not vary with time and the assumption here is that the load or field conditions are gradually applied. From a formal point of view, three conditions have to be met in any stress analysis, equilibrium of forces (or stress), compatibility of displacements and satisfaction of the state of stress at continuum boundaries. Various kinds of loads like force, pressure, inertia, thermal, or specified displacements can be applied. An important point to consider is that at least one of the displacements must be known before the rest can be determined. The known displacements are referred to as boundary conditions and are often times a zero value.

### 3.1 Analytical Procedure:

1) Static structural analysis of press. Vessel for different model is to be done in 'ANSYS WORKBENCH'.

Following is the list of model of pressure vessel with nozzle attachment for analysis:

Table No. 3.1 List of Different Model

| Model No. | No of nozzle | Angle between nozzles |
|---|---|---|
| 1 | 1 | - |
| 2 | 2 | $60^0$ |
| 3 | 2 | $90^0$ |
| 4 | 2 | $180^0$ |
| 5 | 3 | $60^0$ |
| 6 | 3 | $90^0$ |
| 7 | 4 | $60^0$ |
| 8 | 4 | $90^0$ |

2) Static structural analysis of press. Vessel by changing angular position between nozzles. In this we need to find an optimum angle such that the stresses are maintained within limits, because the closer the two nozzles get, the more interaction of their stress zones will take.

3) To get convergence or to increase accuracy of analysis all above analysis is repeatedly done by changing element size of vessel, nozzle & nozzle pad. i.e. we will get increase in no. of elements.

4) All above analysis for 4 different element size is to be done by two types of different meshing for each & every model by following all above mentioned condition. i.e. same analysis is conducted for two types of element shape (mesh)

i) Hex dominant meshing

ii) Tetrahedron meshing

Table No. 3.2 Element Size for parts

| No. | Element size for different parts ( mm) | | |
|---|---|---|---|
| | vessel | nozzle | nozzle pad |
| 1 | 100 | 30 | 30 |
| 2 | 75 | 25 | 25 |
| 3 | 60 | 20 | 20 |
| 4 | 50 | 15 | 15 |

.

**Total approx. number of analysis runs: 128**

### 3.2 Inputs Finite Element Analysis :

**A. Design Data:**

Design Code: ASME Section VIII Div. 2

Design Pressure: 0.23 MPa = $0.23 \times 10^6$ Pa = 2.3 N/m$^2$

Reference Temperature: $22^{0\,C}$

Material: structural steel

**B. Geometry:**

Vessel ID: 1150 mm

Shell Thickness: 75 mm

Length of Vessel: 6200 mm

Pad ID: 280 mm

Pad OD: 560 mm

Pad Thickness: 72 mm

Nozzle ID: 150 mm

Nozzle Thickness: 65 mm

Nozzle Height (total): 442.3 mm

Nozzle Head Dia.: 400 mm

Nozzle Head Thickness: 45 mm

**C. Material Data:**

Following are the Material properties for Structural steel:

Table No.3.3 Material Property

| Property | Value | Unit |
|---|---|---|
| Density | 7850 | kg/m$^3$ |
| Youngs Modulus | 2.00E+11 | N/mm$^2$ |
| Poissons Ratio | 0.3 | |
| Bulk Modulus | 1.67E+11 | Pa |
| Shear Modulus | 7.69E+10 | Pa |
| Coeff. Of thermal expansion | 1.20E-05 | /$^0$C |

**D. Mechanical Boundary Conditions**

1. Internal Pressure: 0.23 N/mm2
2. Displacement:

Table No.3.4 Displacement

| Axis Component | Left end | Right End |
|---|---|---|
| X Comp. | 0.m | Free |
| Y Comp. | 0.m | Free |
| Z Comp. | Free | 0. m (ramped) |

**3.3 Analysis of model of pressure vessel with number of nozzle=2 & angle=90$^0$:**

1) To conduct Structural Analysis and determine the Stress conditions in the press. Vessel & at the nozzle junction. (on the reinforcement pad)
2) To check convergence test for model, doing structural analysis by increasing number of elements.

**i) Analysis System:**

In our case we have to find stress in area of nozzle attachment along with overall stress in pressure vessel. Here, pressure applied is remains static. i.e. no variation with respect to time (df/dt = 0) and hence it can be considered as static structural (ansys) analysis system.

**ii) Model:**

We are interested in to calculate stress only in region of nozzle attachment with periphery of pressure vessel. We have different model of pressure vessel including variation in number & angle of nozzle.

**iii) Meshing Method:**

We are using both meshing method for model of press. vessel. i.e. for model conducted analysis by hex & tetra meshing by keeping same element size.

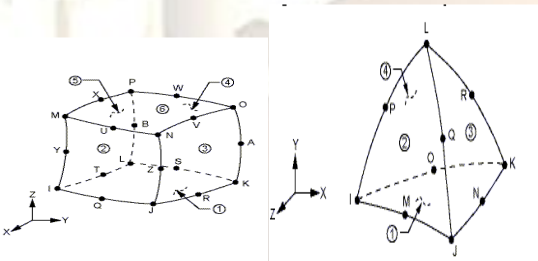


Fig.3.1 Hexahedron (Brick) elements & Tetrahedron elements

**v) Structural Analysis Setting:**

In this step, we have to give boundary condition (constraint) to model.
The Vessel is going to be tested at a pressure of 0.23 MPa, hence this was applied as the Internal pressure for the Vessel.

**4. Results & Discussion:**

The maximum stress values are 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 Pressure Vessel and the nozzle. High stress concentration is developed at this location due to abrupt change in the geometry and the consequent change in stress flow.

**4.1 Solution:**

Solution is followed by meshing & applying boundary condition. Total Deformation & Equivalent stress (Von-Mises stress) is calculated for overall pressure vessel, pad 1
& pad 2.

**i)Overall Max. Deformation:**

Max. Deformation in Pressure Vessel is 9.7518 × 10 $^{(-5)}$ m.
Min. Deformation in Pressure Vessel is 7.7107× 10 $^{(-8)}$ m

**ii) Overall Equivalent Stress:**

Maximum Equivalent Von Mises Stress in the Pressure Vessel is 6.2765 × $10^6$ Pa, which is approximately 6.2765 MPa.

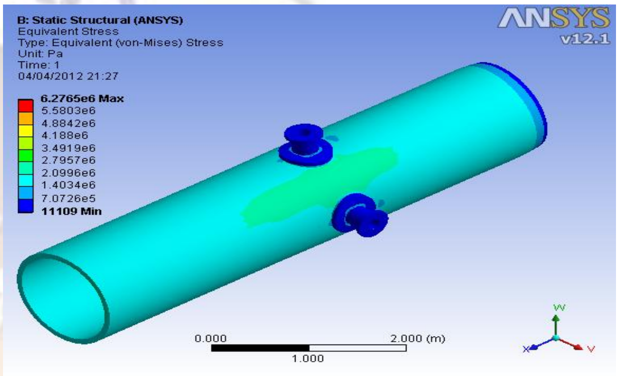



Fig. 4.1 overall max.equivalent stress

**iii) Equivalent Stress of Pad 1:**

Maximum Equivalent Von Mises Stress in the Pressure Vessel is 2.8314 × $10^6$ Pa, which is approximately 2.8314 MPa

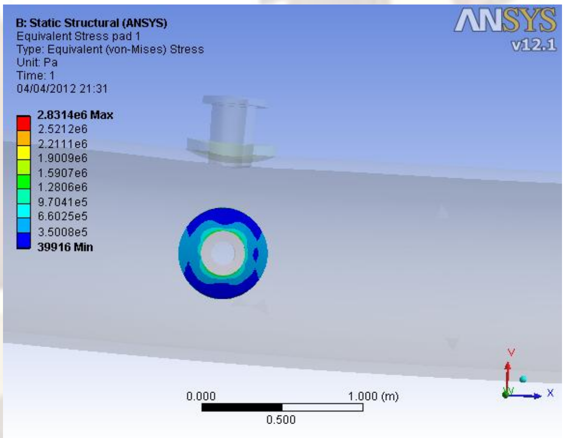



Fig. 4.2 equivalent stress of pad 1

**iv) Equivalent Stress pad 2:**

Maximum Equivalent Von Mises Stress in the Pressure Vessel is 3.0743 × $10^6$ Pa, which is approximately 3.0743 MPa

**v) Stress at area of nozzle attachment:**

From fig. we can see that there is maximum stress in the area of nozzle attachment.
Maximum Equivalent Von-Mises Stress is 6.2765 × $10^6$ Pa.

**4.2 Result of Two Nozzle & Angle=90:**

Table No. 4.1 Result of nozzle.=2 angle=$90^0$

| Mesh type | No.of Element | σ max overall | σ max pad | σ PAD 1 | σ PAD 2 |
|---|---|---|---|---|---|
| Hex | 9163 | 6.28E+06 | 3.07E+06 | 2.83E+06 | 3.07E+06 |
| Hex | 13666 | 6.32E+06 | 3.12E+06 | 3.12E+06 | 2.87E+06 |
| Hex | 28188 | 6.23E+06 | 3.47E+06 | 3.47E+06 | 3.29E+06 |
| Hex | 62244 | 6.29E+06 | 3.97E+06 | 3.97E+06 | 3.82E+06 |
| Tetra | 40313 | 5.79E+06 | 2.83E+06 | 2.83E+06 | 2.62E+00 |
| Tetra | 74741 | 5.91E+06 | 3.11E+06 | 2.95E+06 | 3.11E+06 |
| Tetra | 168158 | 6.01E+06 | 3.09E+06 | 2.83E+06 | 3.09E+06 |
| Tetra | 322300 | 6.06E+06 | 3.33E+06 | 3.33E+06 | 3.32E+06 |

**4.2.2 Tetrahedron Meshing:**

**i) Overall Max. Stress (for tetra. meshing):**
Stress is Converging to a value of 6.0638 × $10^6$ Pa.

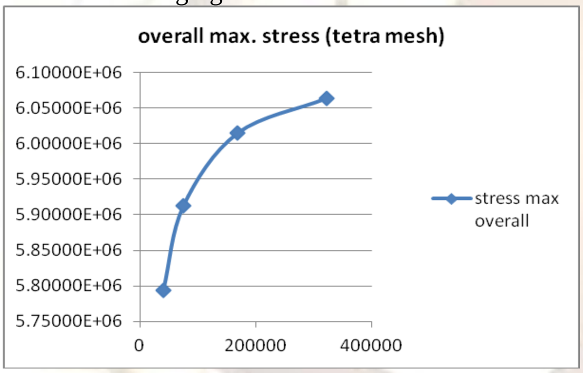



Fig.4.3 Graph of Overall Max. Stress (for tetra Meshing)

**ii) Max. Stress on pad 1 & pad 2 (for tetra. Meshing):**
Stress is Converging to a value of 3.332 × $10^6$ Pa for pad 1.
Stress is Converging to a value of 3.3242 × $10^6$ Pa for pad 2.

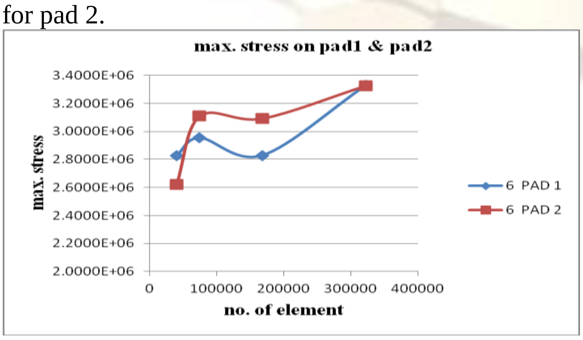



Fig. 4.4 Graph of Max. Stress on pad 1 & pad 2 (for tetra Meshing)

**4.3 Results of Peak Stresses from all cases :**

Table No. 4.2 Result of peak stresses

| No. | case | σ overall | σ pad max |
|---|---|---|---|
| 1 | Noz=1 | 6.68E+06 | 4.16E+06 |
| 2 | Noz=2 angle=60 | 6.19E+06 | 4.44E+06 |
| 3 | Noz=2 angle=90 | 6.32E+06 | 3.97E+06 |
| 4 | Noz=2 ang.=180 | 6.72E+06 | 2.89E+06 |
| 5 | Noz=3 angle=60 | 6.01E+06 | 5.20E+06 |
| 6 | Noz=3 angle=90 | 6.34E+06 | 4.31E+06 |
| 7 | Noz=4 angle=60 | 6.00E+06 | 5.80E+06 |
| 8 | Noz=4 angle=90 | 5.98E+06 | 3.16E+06 |

Following are results of peak stresses of overall pressure vessel & in the region of nozzle attachment i.e. pad from all cases.
From comparing results of peak stresses, stress is minimum for case $N_{2/180}$ & $N_{4/90}$ .

**4.4 Nozzle Stress Increment Factor (N):**
The Stress increment factor for a particular case is defined as
N= (Stress peak for case considered / Stress peak for single nozzle)
e.g. $N_{2/60}$ =(4.44E+06) / (4.16E+06)
=1.0672

Table No. 4.3 Nozzle Stress Increment Factor

| No. | case | N (stress increment factor) |
|---|---|---|
| 1 | $N_{2/60}$ | 1.0672 |
| 2 | $N_{2/90}$ | 0.9534 |
| 3 | $N_{2/180}$ | 0.6936 |
| 4 | $N_{3/60}$ | 1.2493 |
| 5 | $N_{3/90}$ | 1.0344 |
| 6 | $N_{4/60}$ | 1.393 |
| 7 | $N_{4/90}$ | 0.7585 |

From observing table of nozzle stress increment factor we can conclude that stress increment factor for $N_{2/180}$ & $N_{4/90}$ is low.

## 5. Conclusion:

1. The accuracy of the FE model is highly dependent on the mesh employed, especially if higher order (cubic, quadratic etc.) elements are not used. In general, a finer mesh will produce more accurate results than a coarser mesh. At some point, one reaches a point of diminishing returns, where the increased mesh density fails to produce a significant change in the results. At this point the mesh is said to be "converged." Mesh convergence remains the most reliable means of judging model accuracy
2. For all of these models, convergence is clearly observed as the mesh is refined. From our result we are getting more reliable monotonic convergence instead of oscillatory convergence. In general by increasing number of nodes / elements it improves accuracy of results.
3. The maximum stress values are 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 Pressure Vessel and the nozzle. High stress concentration is developed at this location due to abrupt change in the geometry and the consequent change in stress flow.
4. Peak Stress for symmetrical nozzle attachment is lowest than other unsymmetrical cases.
5.Stress increment factor for symmetrical nozzle attachment is lower than others.
6. However, from our analysis we can conclude that symmetry is a far more important factor than number of nozzles.

## REFERENCES:

[1] Dekker, C.J. and Bos, H.J., "Nozzles-on external loads and internal pressure", *International Journal of pressure vessel and piping 72 (1997) I-18.*

[2] Fang, J., Tang, Q.H., and Sang Z.F., "A comparative study of usefulness for pad reinforcement in cylindrical vessels under external load on nozzle", *International journal of pressure vessels & piping 86 (2009) 273-279.*

[3] Amran Ayob, "Stress analysis of torispherical shell with radial nozzle", *Journal - The Institution of Engineers, Malaysia (Vol. 67, No. 3, September 2000*

[4] Skopinsky, V.N. and Smetankin, A.B., "Modelling and stress analysis of nozzle connections in ellipsoidal heads of pressure vessels under external loading'',*Int. Journals of Applied Mechanics and Engineering, 2006, vol.11, No.4, pp.965-979.*

[5] Porter, M.A., Martens, D. H. and Marcal,P., 1998, "On using finite element analysis for pressure vessel design", *PVP Vol. 368, ASME, New York, NY, pp. 139-146.*

[6] Hardenbergh Donald E.,"Stresses at nozzles connections of pressure vessels'',*SESA Spring Meetting,Washington ,1959.*

[7] Ha, J.L, Sun, B.C., and Koplik, B., 1995, "Local Stress Factors of a Pipe-Nozzle under Internal Pressure*," Nuclear Engineering and Design*, 157, Elsevier Science S.A., Lausanne, Switzerland, pp. 81-91.

[8] Wichman, K.R., Hooper,A.G., and Mershon, J.L., "Local stresses in Spherical and Cylindrical Shells due to external loadings'', *WRC Bulletin No.107, Aug.1965,revised Mar.1979.*

[9] Yun-Jae Kim, Kuk-Hee Lee and Chi-Yong Park, "Limit loads for thin-walled piping branch junctions under internal pressure and in-plane bending '', *International Journal of Pressure Vessels and Piping 83 (2006) 645–653*

[10] M. Qadir, D. Redekop,'' SCF analysis of a pressurized vessel–nozzle intersection with wall thinning damage'', *International Journal of Pressure Vessels and Piping 86 (2009) 541–549*

[11] Erdogan Madenci and Ibrahim Guven "The Finite Element Method and Applications in Engineering Using Ansys ", *Springer publication, ISBN-10: 0-387-28289-0.*